

## **Users Manual for AllenCAD Pro 7.0**

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# Chapter 1 Getting Started

## Using Help

### Finding Information

Help information can be searched for in many ways. In the left pane, you will see three tabs:



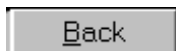
The **Contents tab** will display a list of topics and subtopics in a Table of Contents format. This will allow you to quickly locate your information by clicking on general headings and following sub-headings to the relevant topics. This can be especially useful when you are unfamiliar with terminology.

The **Index Tab** will display an alphabetical listing of key words that are contained within the topics. Clicking one of these will display a list of topics that contain this keyword. Clicking one of the topics on the list will bring the topic into the right pane of the Help window for you to read. The index is particularly helpful in locating information that may not be directly referred to in topic titles.

The **Search tab** will prompt you for a key word and then display an alphabetical list of topics containing the word or words entered in the keyword field.

**Note:** When working with AllenCad running, it can be helpful at times to be able to cross-refer the Help file. To make this simpler, right mouse click anywhere on the Help screen and select "Keep Help on Top".

### Displaying Previous Topics



Using the **Back** button will return you to the previously viewed topic. This button can be pressed repeatedly until you are returned to the Introduction topic.

### Printing Topic Information

Often, you will want to keep a printed copy of help information for future reference. To print your topic, you may right-click within the topic window and select print from the option dialog that is presented, or you can press the Print button that is located on the Menu Bar.

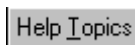


This command will offer you some print options, that you may select as is appropriate.

The options are "Print the selected topic" and "Print the selected topic and all subtopics". Generally, you will select the first option.

You can also print a topic from the left window by selecting the topic and right clicking within the left window similarly to the above description.

### Calling the Contents Window

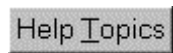


By pressing the Help Topics button on the Menu Bar, the Contents Window can be called at



any time, allowing you to switch to other topics for viewing.

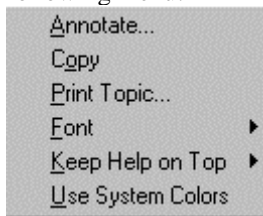
## Showing and Hiding the Contents Window



You may occasionally wish to maximize your topic screen or to keep a help topic visible while you continue to work in AllenCad. By pressing the Help Topics button on the Menu Bar, the left pane is toggled on and off, filling the window with the right (topic) pane. When used in conjunction with the program, this window can then be sized appropriately.

## Keeping Help "Ontop"

At times, you may wish to view Help information while you work in AllenCad. Rather than flip back and forth between applications, you can simply right-mouse click on on the Help screen and you will see the following menu:



Select "Keep Help Ontop" to keep it visible while you work in AllenCad.

## Browse and Topics Buttons



Due to the complexity of the information presented and relying on the facilities offered by the Table of Contents, Index and Search features available in this Help file, a particular browse order has not been provided for these Help topics. These buttons therefore do not fulfill a useful purpose but are a Help standard. As a result, error messages or unexpected results may occur if these buttons are pressed. These can simply be ignored.

## Getting Additional Help

Additional help is available from the following sources:

While you are using the program, AllenCad monitors your activity so that when you press F1 from within a command, system variable, or dialog box, you will be presented with help information directly relating to

your current activity. Pressing F1 at other times will summon the general Help file. If you need to access Windows Help, simply press F1 a second time.

Many of the dialog boxes contain a "?" button that will explain the entry options and requirements.

Simple identification of command icons is provided through the provision of ToolTips. Simply place your cursor over an icon and a ToolTip will appear that identifies the associated command.

More detailed information on commands and concepts can be found in the documentation found in the program's /Manuals directory after installation

This AllenCad help features the **Learning Center** serving as a "Quick Start" guide to AllenCad. The Learning Center includes a series of step-by-step tutorials and is included as a separate section of online Help. You can access the Learning Center at any time by pressing F1 or accessing it through the Help pull-down menu.

It is possible that the problem you may be experiencing has been resolved through program service packs, so be sure to check the website <http://adsi-usa.com> for the latest updates for your program.

Technical support and general information on the program is available at the following location: (www.ADSI.com). All AllenCad products offer free technical support for the warrantee period of the ADSI machine purchased with the cutter. After the warrentee period, support is available either on a payment per incident basis, or you may extend your warrentee. Full details on support programs are available on our website. Technical support requests should be directed to [support@adsi-usa.com](mailto:support@adsi-usa.com). Qualified and professional support staff are available from 8:30AM to 5:00PM Eastern Time. Additional Help is freely available to all users through the Online articles and the moderated User forum on the ADSI website.

When reporting technical difficulties, it is highly recommended that you provide a Log File of command and key sequences that lead up to the point of difficulty as well as a copy of the drawing file you are having difficulties with. These will enable our technicians to provide you with a speedy resolution to your difficulties.

## Log Files

AllenCad enables the user to create a log file containing a record of all issued commands and program responses, as they would appear in the text window. This log file can be of great assistance when program support is required or when writing macros.

Note: While commands and keyed inputs are recorded, coordinate values for points picked onscreen are not recorded in this file.

An operational mode controlled by the systems variable LOGFILEMODE enables logging and another systems variable, LOGFILENAME establishes the default filename and path for the generated file.

The commands LOGFILEON and command LOGFILEOFF begin and end log file recording, respectively and control the value setting for LOGFILEMODE.

## Program Functions

Many of your questions will deal with program commands and how they are used. Apart from links to specific command topics located throughout this help file, you can get to this information by:

1. Checking the Command Reference Chart for a listing of AllenCad commands gives a brief description of what the command does.

2. Going directly to the Command Reference section (below, in the table of contents for this Help file) and looking up the command in the alphabetical listing or
3. Checking the index for the command name

## Introduction To AllenCad

### Introduction



### AllenCad Pro

## Help System

**AllenCad™ ©2000-2002 ADSI Corp. All Rights Reserved.**

Portions of AllenCad ©1996-2002 Felix Computer Aided Technologies GmbH.

#### **PLEASE READ ALL CONTENTS OF THIS INTRODUCTION SECTION BEFORE USING THIS PROGRAM FOR THE FIRST TIME**

All AllenCad products are available for 30-day trial, allowing you to try before you buy. To permanently authorize AllenCad beyond the free trial period, you must register and pay the corresponding license fee by:

Contacting your dealer or agent

Calling our Toll Free (US/Canada) Order Desk at (800) 258-6360 (1-888-977-6937)

Ordering through our secure online store at [www.ADSI.com](http://www.ADSI.com)

#### **AllenCad Help Resources:**

##### **AllenCad Help**

This Help System contains your program User Guide that includes detailed reference to the concepts and commands used in AllenCad. It has been presented in this format to provide quick access to this information.

Please refer to the Table of Contents window on your left for a listing of help topics available, as well as the Index and Search windows for specific reference searches. Each Table of Contents section can be expanded using the plus (+) icon, or contracted using the minus (-) icon. Please refer to the "Using Help" section for further instructions on how to use Help.

AllenCad Help resources available to you also include the AllenCad Learning Center (incorporated into program Help). The Learning Center contains simplified instructions and tutorials designed to get you started with AllenCad right away. If you are unfamiliar with computer-aided design or drafting, this is where you should begin the learning process. Even if you are an experienced CAD user, the Learning Center provides insights on system flow and settings to optimize your design and drafting experience.

#### **Printed Documentation**

AllenCad documentation is included in the \manuals folder after program installation, You must have Adobe's free Acrobat Reader (version 4 or higher), (available at <http://www.adobe.com/products/acrobat/readstep.html>), to access the included PDF documents.

### **Free Extended Webhelp**

Extended online help, including free access to the AllenCad Online and user forums, is available 24 hours a day at [www.ADSI.com](http://www.ADSI.com).

### **Priority Support**

All AllenCad inquiries requiring an email or telephone require payment for each incident, or are available for 12-month periods by enrolling in one of our annual priority support programs. Qualified and professional support staff are available from 8:30AM to 5:00PM Eastern Time.

## **Authorizing AllenCad**

By default, AllenCad Pro is installed as 30-day "convertible demos. A startup screen will appear during the trial period allowing you to:

- Obtain a permanent authorization key or purchase other Power products
- Unlock the system for permanent use by entering your Product/CD key
- Contact ADSI by email or phone
- Start the program

Once you have authorized your system for permanent use, the startup screen will no longer appear. Additional links to the ADSI website are contained under the Help pull-down menu once the program is running.

## **AllenCad Pro**

### **Features of AllenCad Pro?**

AllenCad Pro use core technology, and offers exceptional power and value to simplify the computer-aided design process. It offers unrivalled compatibility with AutoCAD, by Autodesk Inc., with virtually identical handling of layers, text, fonts, styles, and advanced design components such as extended entity data.

**AllenCad Pro** has the ability to directly read *and write* both DWG and DXF files, created using AutoCAD or other CAD systems utilizing these industry standard formats. This provides the capability for file exchange between AllenCad Pro and these other programs. It can also read and write both DWG and DXF files. This permits file sharing with other cad programs.

**AllenCAD** offers complete 2D design processing, multiple desktop (user interface) configurations to suit your preference. For new CAD users, the default AllenCad desktop (menu, dialogs, and toolbars) offers a simple and intuitive environment in which to work. For AutoCAD-familiar operators, an AutoCAD emulated desktop can be selected, virtually eliminating the learning curve in a shared AllenCad/AutoCAD workplace. In addition, users can select between AllenCad and Autocad 15 emulation menus.

**AllenCad Pro** also features capabilities, to include advanced 3D processing and rendering, visual document management, batch plotting and conversion, full raster overlay capabilities, and complete system customization or extension with visual interface component editing and advanced programming using Lisp, C/C++, or Delphi. AllenCad Pro further enhances the professional designer's drawing experience with system extensions, including a selection of high quality, pre-drawn PowerSYMBOLS.

## Where Can I Get My Permanent Authorization Key?

AllenCad is often installed as free 30-day trial version after downloading from the ADSI website (<http://www.ADSI.com>) or by ordering a trial CD (service, shipping and handling charges apply). To obtain a permanent authorization key that will remove all time limitations on the use of this program, you must contact your selling agent, order online, or call ADSI direct at (800) 258-6360 (toll free US/Can) or (603) 216-6344.

## AllenCad Functions

### AllenCad Extended Feature List

#### AllenCad Extended Features

Drawing Navigator™ an explorer style window, which displays all the layers, line types, blocks, text styles, views, and even coordinate systems contained in a drawing.

Visual Block Manager lets you quickly create graphic block libraries (supports FLEXIBLE DRAWING EXCHANGE (FLX), DWG, and DXF blocks). With the click of a mouse or digitizer button, blocks can be graphically inserted at any scale or rotation you need.

AllenCad Pro's Image Library Tool lets you store multiple slides and bitmaps in a single library. Supports AutoCAD® slide (.SLD) and Windows bitmap (.BMP) formats.

AllenCad Pro's Hybrid Raster/Vector Image Overlay capabilities allow you to integrate raster bitmaps directly into vector drawings.

AllenCad Pro's Batch Processor lets you perform unattended batch plotting or file conversions.

Program customization and enhancement capabilities through LISP, C++ and Delphi programming support.

### Program Features

#### Feature

Short learning curve  
Context sensitive online help with print-ready documentation  
Online Learning Center  
Free access to Web help  
Metric or Imperial Units  
Fast pan and zoom based on an new 3D display list  
Options Menus and dialog box support in many commands.  
"Repeat previous command" function built-in.  
Cut and Paste between drawings and the clipboard  
Simultaneous opening of up to four drawings with up to four viewports of each  
A full range of object drawing and editing commands  
Context sensitive object editing (just pick an element and edit options are presented)  
Zoom back up to 20 steps  
Unlimited undo function  
Icon menu support  
Grouped functions on palettes, that can be called onto the drawing surface for quick access to commands  
User-definable Alias commands  
Drag and Drop user interface  
Easy to organize part (block) libraries.

Preview of drawings, blocks and symbols  
 Easy definition of complex objects (parts, blocks, symbols)  
 Visual Part and Symbol Library Manager (DWG and DXF files supported)  
 ( \* No DWG or DXF Export)  
 Support for Extended Entity Data (EED)  
 Externally referenced drawings (XRef's)  
 Sophisticated object grouping  
 Associated dimensioning and hatching  
 User-definable dimensioning styles  
 Layout mode (Paper space / Model space)  
 ( \* Model Space elements can be imported and viewed only)  
 Enhanced high speed printing/plotting with user selectable pen assignments and full plot previews  
 Batch printing/plotting and file conversion  
 Detailed plot menu with multiple configuration options  
 User-definable coordinate systems (UCS)  
 Autorendering based on Microsoft OpenGL graphics  
 Fast hidden line removal  
 Interface for hybrid raster/vector applications  
 Digitizer support/Tablet menu support  
 Thickness command for common elements (3D entity extrusion)  
 Raster file import, manipulation and editing  
 Direct Windows Metafile (WMF) and Bitmap (BMP) Export  
 AutoPort™ utility library automates the conversion of AutoLisp®/ADS® programs, menus, and dialogs written for use with AutoCAD®

## **AutoCAD Compatibility**

### **Feature**

Direct reading and writing of AutoCAD DWG and DXF files up to Release 2000  
 Transparent execution of all compatible AutoCAD functions  
 Industry standard layers, line types, blocks, fonts, and line types  
 AutoCAD DCL Dialog Converter  
 Options to load AutoCAD emulated menus (R14 and R2000), toolbars, command aliases and "hot key" assignments  
 Direct insertion of DWG or DXF drawing parts (blocks)  
 Direct reading of AutoCAD 1,0 and 1.1 SHX fonts, linetypes and Hatch Patterns (SHX/SHP, LIN and PAT files) - Including Bigfont support.  
 WCS/UCS coordinate systems

## **Warnings Trademarks and Disclaimers**

## Warning

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AllenCad Pro includes an Advanced Programming Interface (API), supporting limitless system customization using Lisp, Visual C++, or Delphi™ programming languages. AllenCad Pro's unique AutoPort™ translation tool set allows quick and painless "porting" of existing code written in AutoLISP® or ADS®. Menus, dialog boxes, and palettes created for AutoCAD® use are automatically converted, saving end users thousands of dollars per license for each CAD system purchased.

If you are a commercial, corporate, or government developer, AllenCad Pro™ is available as PowerENGINE™. This is a completely scalable and flexible Software Development System. Organizations of any size can embed all or selected portions of PowerENGINE™ into solutions of their own design. For more information on how to join the Power Developer's Network please visit <http://www.ADSI.com>.

**AllenCad Pro Customization and Lisp Programming Guide** (see corresponding PDF file)  
Guide to customizing AllenCad Pro using the Desktop and Menu Editor (DME™), and the Flexible Lisp (FLISP™) application programming interface (API).

**AllenCad FDT Programming Guide** (see corresponding PDF file)  
Advanced guide to customizing AllenCad Pro using the AllenCad Flexible Development Toolkit (FDT).  
For experienced Visual C++ or Delphi programmers only.

## AllenCad Pro General Features

### AllenCad Pro™

*CAD That Works The Way You Do*

**AllenCad Pro™** is one of the world's best professional CAD values, offering power and flexibility nearing AutoCAD® by Autodesk Inc. Fast and easy to use, AllenCad Pro™ conforms completely to industry standards, and is optimized for the latest Windows® operating systems.

**AllenCad Pro™** offers exceptional compatibility with AutoCAD®, the world's leading CAD system with over 4 million users. AllenCad Pro™ has over 300 AutoCAD® compatible commands that require no relearning, allowing for a shared AutoCAD®/AllenCad™ workplace. AllenCad Pro™ operates directly with industry standard DWG® and DXF® files (up to R2000 format), with round-trip retention of critical R2000 drawing elements.

**AllenCad Pro™** is affordable. For the price of a single AutoCAD® 2000i license, you can equip about eight AllenCad Pro™ workstations.

**AllenCad Pro™** incorporates a completely open architecture design, with drag-and-drop placement of system icons. Three different desktops (user interfaces) are included with the system, making AllenCad Pro™ instantly usable by AutoCAD® familiar and non-AutoCAD® familiar operators. AllenCad Pro's visual menu, dialog, and toolbox editor allows technicians of any skill level to instantly customize the desktop for maximum user productivity.

**AllenCad Pro™** has all of the drawing and editing commands most CAD professionals expect. Advanced 2D drawing commands, such as trim, fillet, or chamfer, complement sophisticated functions such as associative dimensioning and hatching. AllenCad Pro™ offers extensive 3D element processing and creation (apart from solids modeling, planned for future releases), allowing the user to work in the plane of choice. Rendering and line/face hiding is based on the latest Open GL™ standards, reducing the time needed to perform complex imaging tasks.

**AllenCad Pro™** includes an Advanced Programming Interface (API), supporting endless system customization using Lisp, Visual C++, or Delphi™ programming languages. AllenCad Pro's unique

AutoPort™ translation tool set allows quick and painless "porting" of existing code written in AutoLISP® or ADS®. Menus, dialog boxes, and palettes created for AutoCAD® use are automatically converted, saving end users thousands of dollars per license for each CAD system purchased.

If you are a commercial, corporate, or government developer, AllenCad Pro™ is available as **PowerENGINE™**. A completely scalable and flexible **Software Development System**, organizations of any size can embed all or selected portions of PowerENGINE™ into solutions of their own design by becoming a member of the Power Developer Network. More information is available at [www.ADSI.com](http://www.ADSI.com).

## AllenCad Pro Extended Feature List

**Drawing Navigator™** an explorer style window, which displays all the layers, line types, blocks, text styles, views, and even coordinate systems contained in a drawing.

**Visual Block Manager** lets you quickly create graphic block libraries (supports FLEXIBLE DRAWING EXCHANGE (FLX), DWG, and DXF blocks). With the click of a mouse or digitizer button, blocks can be graphically inserted at any scale or rotation you need.

**AllenCad Pro's Image Library Tool** lets you store multiple slides and bitmaps in a single library. Supports AutoCAD® slide (.SLD) and Windows bitmap (.BMP) formats.

**AllenCad Pro's Hybrid Raster/Vector Image Overlay capabilities** allows you to integrate raster bitmaps directly into vector drawings.

**AllenCad Pro's Batch Processor** lets you perform unattended batch plotting or file conversions.

### Additional Program Highlights:

- Short learning curve
- Context sensitive online help with print-ready documentation
- Online Learning Center
- Free access to Web help
- Metric or Imperial Units
- Fast pan and zoom based on an new 3D display list
- Options Menus and dialog box support in many commands.
- "Repeat previous command" function built-in.
- Cut and Paste between drawings and the clipboard
- Simultaneous opening of up to four drawings with up to four viewports of each
- A full range of object drawing and editing commands
- Context sensitive object editing (just pick an element and edit options are presented)
- Zoom back up to 20 steps
- Unlimited undo function
- Icon menu support
- Grouped functions on palettes, that can be called onto the drawing surface for quick access to commands
- User-definable Alias commands
- Drag and Drop user interface
- Easy to organize *part* (block) libraries.
- Preview of drawings, blocks and symbols
- Easy definition of complex objects (parts, blocks, symbols)
- Visual Part and Symbol Library Manager (DWG® and DXF® files supported)
- Support for Extended Entity Data (EED)
- Externally referenced drawings (XRef's)
- Sophisticated object grouping
- Associated dimensioning and hatching

- User-definable dimensioning styles
- Layout mode (Paper space / Model space)
- Enhanced high speed printing/plotting with user selectable pen assignments and full plot previews
- Batch printing/plotting and file conversion
- Detailed plot menu with multiple configuration options
- User-definable coordinate systems (UCS)
- Autorendering based on Microsoft OpenGL® graphics
- Fast hidden line removal
- Interface for hybrid raster/vector applications
- Digitizer support/Tablet menu support
- Thickness command for common elements (3D entity extrusion)
- Raster file import, manipulation and editing
- Direct Windows Metafile (WMF) and Bitmap (BMP) Export

#### **AutoCAD Compatibility Highlights:**

- Exceptional AutoCAD® compatibility
- Direct reading and writing of AutoCAD DWG® and DXF® files up to Release 2000
- Transparent execution of all compatible AutoCAD® functions
- Industry standard layers, line types, blocks, fonts, and line types
- AutoCAD DCL® Dialog Converter
- Options to load AutoCAD® emulated menus (R14 and R2000), toolbars, command aliases and "hot key" assignments
- Direct insertion of DWG or DXF drawing parts (blocks)
- Direct reading of AutoCAD 1,0 and 1.1 SHX fonts, linetypes and Hatch Patterns (SHX/SHP, LIN and PAT files) – Including Bigfont support.
- WCS/UCS coordinate systems

#### **Programming and Customization Highlights:**

- True 32-bit CAD
- High-Speed Performance
- Optimized for Windows 95/98/NT/ME and 2000
- Network-ready
- Customizable toolbars
- Open software architecture
- Visual Menu, Dialog, and Tool box editor
- Lisp Interpreter with code protection option
- C/C++ Application Programming Interface (API) similar to AutoCAD
- Delphi™ Advanced Programming Interface
- Dynamic Link Library support with DDE
- Callback and Notification functions
- SQL2/ODBC Support
- True Type Font support

AutoPort™ utility library automates the conversion of AutoLisp®/ADS® programs, menus, and dialogs written for use with AutoCAD®

## **General Remarks**

AllenCad runs on a Microsoft Windows® desktop.

All essential program steps function therefore, according to Windows protocols. This is true for pull-down menus, dialog boxes, icons, scroll bars, etc. The desktop provides a series of standard elements whose function and mode match the corresponding elements of the Windows desktop. For example, there

are palette buttons for opening new windows, as well as for changing window sizes. Use of these "standard" Windows control elements therefore has not been explained in detail neither in the documents nor in the online help.

The use of this program requires at least a minimum of Windows knowledge. If you have any questions that concern Windows control elements, the answers can be most often be found in your Windows manual, Windows Online Help, or other literature on the subject of Microsoft Windows.

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## **System Requirements**

AllenCad runs on machines using a 32 bit Windows operating system. The following system should be considered a practical minimum for use with this program:

- A Pentium class workstation
- 32 megabytes of ram (64 megabytes of ram or more would be desirable)
- 60 megabytes of Disk storage to accommodate the program, plus storage for additional support and drawing files.
- A CDROM drive (CD version only)
- Internet Explorer or Netscape version 4 or above

## **Installing AllenCad**

### **PLEASE READ ALL CONTENTS OF THIS INTRODUCTION SECTION BEFORE USING THIS PROGRAM FOR THE FIRST TIME**

AllenCad products are installed as 30-day "convertible demos, allowing you to try before you buy. To

permanently authorize AllenCad for operation beyond the free trial period, you must obtain a product key number by registering as a user and paying the corresponding license fee. Product authorization keys can be obtained by:

- Contacting your dealer or agent
- Calling our Toll Free (US/Canada) Order Desk at (800) 258-6360 (1-888-977-6937)
- Ordering through our secure online store at [www.ADSI.com](http://www.ADSI.com)

### **AllenCad Help Resources:**

This Help System is your detailed quick reference guide to the concepts and commands used in AllenCad Professional.

Please refer to the Table of Contents window on your left for a list of help topics available. Each section can be expanded using the plus (+) icon, or contracted using the minus (-) icon. Refer to the "Using Help" section for further instructions on how to use help.

AllenCad help resources available to you include:

#### **- AllenCad Learning Center (incorporated into program Help)**

The AllenCad Learning Center contains simplified instructions and tutorials designed to get you started with AllenCad right away. If you are unfamiliar with computer-aided design or drafting, this is where you should begin the learning process. Even if you are an experienced CAD user, the Learning Center provides insights on system flow and settings to optimize your design and drafting experience.

#### **- Printed Documentation**

AllenCad Professional includes electronic manuals in Adobe PDF file format, designed to complement this online help. Complementary PDF documentation is included in the \manuals folder after program installation,

You must have Adobe's free Acrobat Reader (version 4 or higher), (available at <http://www.adobe.com/products/acrobat/readstep.html>), to access the included PDF documents.

#### **- Free Extended Webhelp**

Extended online help, including free access to the AllenCad Online and user forums, is available 24 hours a day at [www.ADSI.com](http://www.ADSI.com).

#### **- Personal Support**

AllenCad Professional includes support during the warrentee period of the ADSI equipment purchased with AllenCAD. After the warrentee period support is available at our current costing rates. Qualified and professional support staff are available from 8:30AM to 5:00PM Eastern Time.

## **The Program's User Interface**

### **The Program's User Interface**

AllenCad's user interface is designed to be as flexible and useful as possible, both for experienced and inexperienced CAD operators. AllenCad includes hundreds of system functions (commands) for general-purpose design and drafting.

Each command has a variety of options, which are presented in a popup option menu or in the option bar, depending on how your system is configured (see Option Menu or Option Bar, and Desktop

Configuration sections). System variables (sysvars) can be used to alter how commands or the system itself operates.

From the outset, AllenCad has been designed with a traditional style of interface for computer-aided design processing. This means:

Virtually all system functions can be run by typing the appropriate command

Virtually all system functions can be run by picking the appropriate selection from the pull-down menu

Virtually all system functions can be run by picking an icon from a desktop toolbar or floating toolbox

This user interface allows you to run the same function using three different methods, which can be a little confusing to new CAD operators. Don't let it confuse you! In just a little while, you will begin to see why this three-way interface has been embraced by millions of computer-aided design users around the world as the most productive way to create or modify your designs and drawings.

The trick to this 3-way interface is to experiment with each command, and how each command can be processed. Experienced, or "power" CAD users, traditionally combine typed entry of commands (e.g. type the command "line" to draw a line), with left mouse button selection of drawing elements (e.g. pick the end of an existing line) or complementary system functions (e.g. pick a snap setting for the immediate command).

This 3-way interface conforms largely to the way AutoCAD®, by Autodesk Inc., and MicroStation®, by Bentley Systems Inc. operate. If you are familiar with either of these leading CAD systems, you should be comfortable with AllenCad right away. If you are not, you may want to consider taking an AutoCAD course, or purchasing one of the many books or online courses available for learning AutoCAD.

### **Repeating Commands:**

All commands are repeated (if no command is presently active) or executed (if command processing is currently underway) by clicking the right mouse button or pressing the Enter key. This takes some getting used to if you are not familiar with traditional CAD operation. Commands can be cancelled by pressing the Esc key.

Although unconventional to standard Windows® style operation (where menu selections or icons must be picked each time), this repeating method of command execution is extremely productive once you get used to it.

**Note:** The right mouse button can be used in place of Enter or <Enter> in this or any other system documentation.

### **Macro Control of Commands:**

AllenCad Pro and its developer specific cousin PowerENGINE™, allows macro "scripting" of commands, whereby often used command sequences can be saved to a macro file for repeated operation. See the MACRO command later in this help guide or in the documentation for more information on creating and running macros.

### **Programming Control of Commands:**

AllenCad Pro and its developer specific cousin PowerENGINE™, allow complete programming control of system commands using the Lisp, Visual C++, or Delphi application programming interface (API). Refer to the programming and extension techniques outlined in this help, and more particularly to the Programming and Customization documents, and the Flexible Development Toolkit (FDT) document.

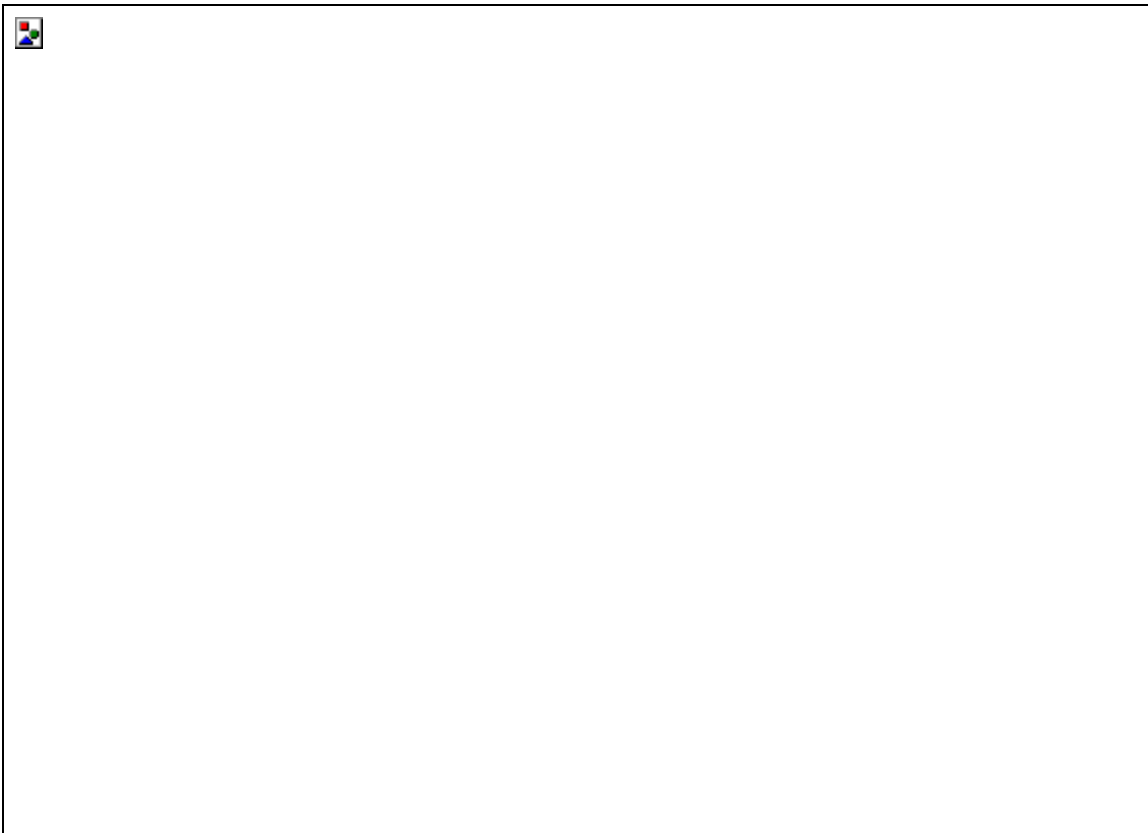
## **The Program's Desktop**

Desktop refers to the user interface components you see and use when you are running AllenCad. This includes:

- drawing title bar
- the pull-down menu
- the horizontal toolbar
- the properties bar
- the drawing area
- the Control Panel (Left Sidebar)
- the coordinate symbol
- the cross hairs
- the command line
- the status line

In addition, the desktop may also contain several floating toolbars, or palettes.

Click anywhere on the image below to get a description of desktop components.



AllenCad ships with a default desktop, plus a number of AutoCAD® emulated desktop components that can be used in place of the default components if you are an AutoCAD familiar operator (see AllenCad for AutoCAD Users section).

All portions of the desktop can be toggled on or off to maximize or minimize the amount of drawing area you have available as you work. The AllenCad command line may be configured at the top of the desktop, the bottom of the desktop, or for experienced users, turned off completely.

Refer to the Desktop Configuration (deskconf) section for details on how desktop components can be altered from the default configuration.

**Warning:** DO NOT turn off all desktop components, or you will be left with a blank drawing area with no menus, etc. The only way to recover from this mistake is to type deskconf (even though the command line is not visible), which recalls the desktop settings popup menu for recovery.

## Desktop Configuration

### Command DESKCONF

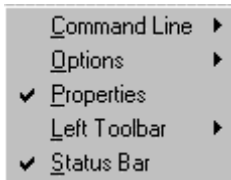
#### Alias DSK

##### Located:

1. Pull-down Menu: Tools – Desktop Manager
2. Or Enter Command or Alias Directly at the Command Line

## Modifying the Desktop Layout

The DESKCONF command's pop-up menu enables you to re-arrange the desktop to suit your needs. This pop-up menu allows you to set various layout options for the programs user interface, including the display of the Command Window, Options Menu or Options Bar, Property Bar, Left Sidebar, and Status Line.



The Desktop Layout pop-up menu

**Caution!** The following sections describe how to turn off desktop elements. The user should exercise caution in doing so, however, as these actions will remove visual references to command inputs. If all desktop elements are turned off and the user wishes to reinstate them, the DESKCONF command will need to be retyped and executed with the relevant options selected. Pressing F2 will call the text window where the user can see the command prompt, as needed. Likewise, online Help can be called as needed, by pressing the F1 key.

### Turning desktop items on and off:

To achieve more space to display drawings on the desktop, or as an advanced user, who is familiar with the names and options of the commands, you may want to turn off certain items of the desktop. You can turn off and on (at runtime) the display of the command line, the Options Menu or context toolbar, the property bar, the left toolbar, and the status bar as the figure above shows.

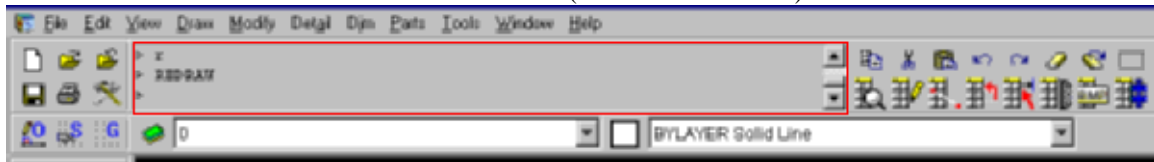
#### How to open the Desktop Configuration pop-up menu:

You can call the *Desktop Configuration* menu by typing the DESKCONF command or selecting the Desktop Configuration selection on the Tools pull-down menu.

You can also open the *Desktop Configuration* pop-up menu by clicking the right mouse button in a "non-functional" area of the program's desktop. Such area is either the background area upon where the drawing viewports are displayed or areas in the property bar, Options Bar, or toolbars that are not covered by buttons or drop-down lists. If necessary, reduce the size of the current drawing(s), cascade the drawing(s), or minimize the drawing(s) to get some free space of the drawing area background. Right-click to this area to open the *Desktop Configuration* menu. You exit this pop-up menu by left clicking to some other area or component on the desktop.

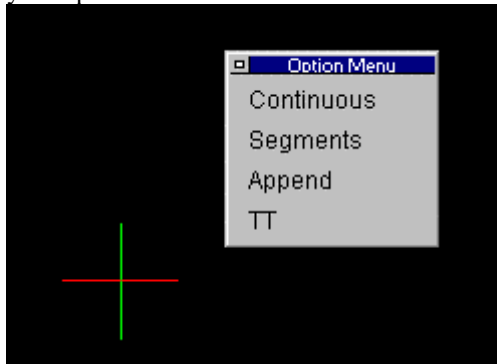


The **Command Line** can either be placed at the top of the screen (below the menu bar as pictured below), or in the default location at the bottom of the screen (above the status line).



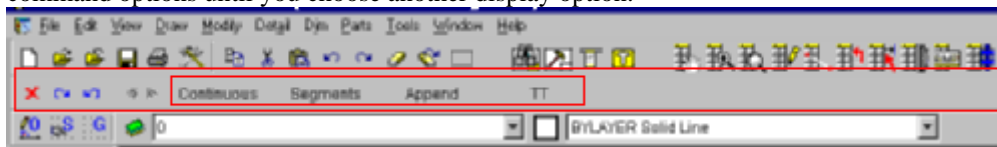
Where you place the bar is strictly a matter of personal preference, as there are no specific advantages in choosing either location. If you are an advanced user, who is familiar with the program's command set, you might even turn the command line off to provide a larger drawing edit window. In this instance, you can always check for program messages or inputs by toggling the text window on/off using the F2 key.

The items of the **Options** sub-menu allow you to determine whether you want to be prompted for command options via an *Options Dialog Menu* or within a *Context Toolbar*. By default, command options are displayed in a pop-up menu that appears in a dialog window in the edit window. These command option dialogs (as pictured below) will disappear or change to offer new options, once you have selected your option.



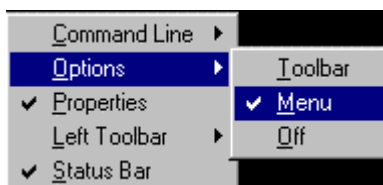
An Options Dialog Menu - The benefits to this option are that it places the options right on the editor where you are working, and that it doesn't reduce the overall size of your drawing edit window. The drawback is that you sometimes may have to move (drag and drop) the dialog if it is in your way as you draw.

If you choose to display options on an *Options Bar*, you will see a new toolbar inserted between the horizontal toolbar and the Properties Bar, as pictured below. This toolbar will remain open to display command options until you choose another display option.



The Options Bar - Note the options presented on the bar.

Note: You can recall the display of the context pop-up menu by selecting the option *Options>Menu* from the *Desktop Configuration* pop-up menu.

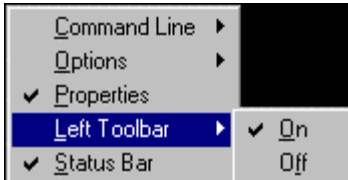


Both the **Properties Bar** and the **Status Bar** can be toggled off and on by clicking on the selection in the

Desktop Configuration dialog. When toggled on, the selection will appear with a check mark as shown below:



The **Left Sidebar** can be switched off and on by clicking on the on or off options that appear in a secondary window when you click on the Left Toolbar option in the Desktop configuration dialog, as shown below:



See Also: Emulating the AutoCAD Desktop

## Drawing Title Bar

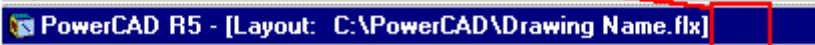
Model Mode - Drawing Window 3



Window Number (0,1,2 or 3)

Layout Mode

No Window Identifier



In Model Mode, each drawing window has at the top a Title Bar. The name of the file will be indicated in this Title Bar. In front of the file name are the complete specifications of the path, with information about the drive, the directory, and any subdirectories in which the file was saved.

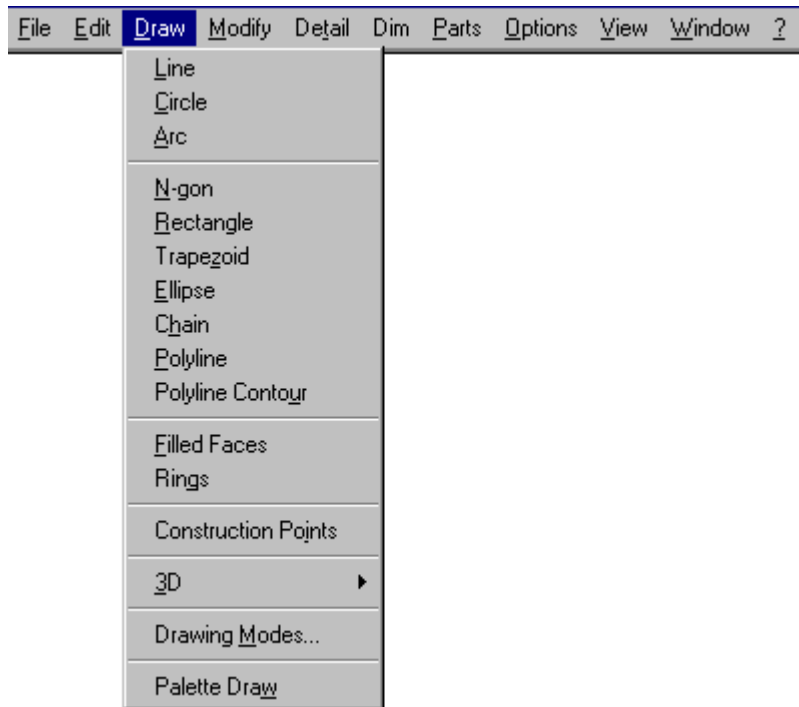
If there are several views of one drawing, the separate drawing windows are marked by continuous numbering. A maximum of four windows can be open at any time and are numbered 0,1,2 and 3. This label will appear behind the drawing name, and will be separated from the name by a vertical line.

When a drawing window is maximized, the described specifications will be found in the Title Bar of the desktop.

If the user has opened several files simultaneously, the specification of the file name will facilitate having an overview of the drawings that are working at the time.

When you change from Model to Layout Mode, all references to drawing window number disappear from the title. In addition, the filename is prefaced by the Layout designator, indicating that you are in Layout mode.

## Menu Bar



Commands (and their options) can be selected using pull-down menus. The menus and commands may also be issued by key sequence (for example, ALT + F, N for new file) or by clicking the proper option field within its menu.

For some commands, submenus are available. Submenus are indicated wherever you see the "right arrow" symbol, as with the 3D entry on the above sample. These submenus offer further commands or variations of the command.

A description of each menu command can be found in the Users Guide Section.

## Horizontal Toolbar



When you install *AllenCad* out of the box, by default, below the menu bar there is a button lines divided into a left and a right area. This component of the desktop layout is called Toolbar.

Depending upon your selected preference for command options, there may also be another bar below, called the Options Bar.

### System Buttons



A group of six buttons is located on the left side of the toolbar. These buttons cover commands from the *File* menu that refer mainly to the opening, saving and output of drawings, and to general program settings:



**Starting a New Drawing**



**Opening an Existing Drawing**



**Closing the Current Drawing**



**Saving the Current Drawing File**



**Printing the Current Drawing File**



**Configuration: General Settings for the Program**



Another group of symbols is arranged on the toolbar, to the right of the systems buttons. These icons represent commonly used drawing management commands.

Among the symbols you find, are those for customization of the program's environment and the arrangement of the drawing windows.



On the right side of the toolbar is a group of icons that call the palettes onscreen. These palettes are like toolbars in that they contain groups of associated command icons, but unlike the toolbar itself, these palettes are called onto and can be moved around on the drawing window, to be available where and when you need them. When finished with them, palettes can be closed until called again.

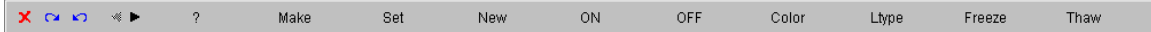
## The Options Bar

As an alternative to the Options Menu, you can configure to utilize an Options Bar, or Options Bar, instead. This bar will not be visible unless this option is selected.

The Options Bar is a special area of the desktop located underneath the command line. (where command line is located at the top of the screen).

It contains context-sensitive option buttons.

Using those buttons enables you to call options when a command is performed.



The Options Bar is arranged as follows:

A group of three initial buttons on the Options Bar pertain to the execution of commands.

These are (from the left):

1. ▶ Cancel the execution of the current command (Escape)
2. ▶ Nullify the withdrawing of the last work step (Redo)
3. ▶ Withdraw the last work step (Undo)

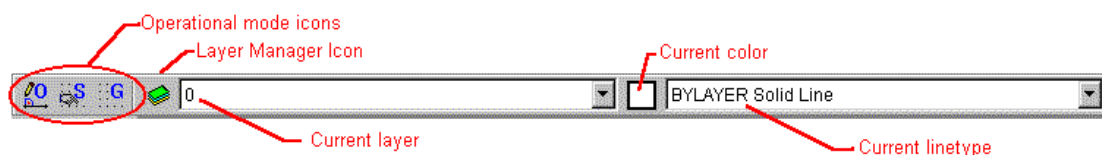
Two arrows in the Options Bar serve to scroll the available options.

A number of command prompts come up with such a number of choices that cannot be displayed all over the available space of the Options Bar. In such situation, one (or both) of the arrows will be available.

After a command has been invoked, the options currently available are displayed in the option bar. Clicking on the desired option will select that option. Hotkeys may also be used, if they are assigned to the options.

The option fields will be changed to correspond with the current status of the drawing. Doing this keeps the number of options shown manageable. If there are no options to the command, these option fields are empty. The option field may change as often as after each step in the drawing.

## The Properties Bar



The Property Bar gives you a quick and comfortable way of setting drawing modes as well as some main properties for further construction and drawing.

The bar's visibility is controlled by the variable DTPROPBAR and indirectly by the DESKCONF command.

The first three buttons in this area serve to directly control the use of the Orthogonal mode as well as the Snap and Grid status. These modes (O = Ortho, S = Snap, G=Grid) can be activated, or disabled, by clicking on the appropriate button. The active respectively non-active condition is indicated by the appearance of the button. Explanations about these drawing tools are found in the section *Drawing with Precision*.

The further elements of the property bar allow setting current entity properties for further drawing: Layer, Color and Linetype.

The first button of this area opens the *Layer Manager* dialog box that permits to create new layers, to specify their properties, to rename them, and so on. Beside this button a drop-down list is found containing all the layer names of the current drawing. From this list you can choose the layer you want to set current.

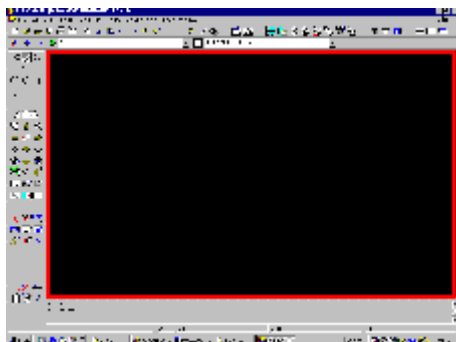
Next, a square displays the color of the current layer, or, if set so, the current explicit color. You can click to this square to open the *Color* dialog box for setting the color applied when creating new entities.

Another drop-down list contains the linetypes defined in the current drawing. From this list you can choose a running linetype for further drawing. Please notice, that usually it is a wise practice to set both the current color and the current linetype to the logical BYLAYER property. This means that the corresponding properties associated to the current layer are applied.

You will find detailed references on layer concepts, and the linetype and color properties in *Users Guide – The Basics of Drawing - Layer and Object Properties*.

You can turn on and off the Property Bar at runtime. See the section *Users Guide – Configuration – Desktop Layout Configuration*.

## The Drawing Area



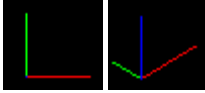
The Drawing Area is the part of the desktop in which drawing entities are created and modified. This area is highlighted in red in the above desktop image.

Up to four drawings with four viewports each can be opened simultaneously in the Drawing Area. Each drawing and each view will be represented in its own window. The individual windows can be tiled, cascaded, or enlarged to fill the Drawing Area.

If more than one window is opened, only one will be the active window. The active window may be recognized by the color in its Title Bar. Only drawings in the active window will be working. A window is selected as the active window by clicking with the mouse or the digitizer on any part of the window or (if there is full screen view) via the pull-down menu *Window*.

Outside of the drawing area, the cursor assumes the typical Windows pointing shape. Within the drawing area, the cursor appears as cross hairs.

## Coordinate Symbol

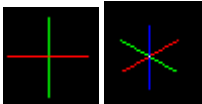


Normally, the program puts a coordinate symbol in the lower left corner of the Drawing Area. This symbol indicates the position of the axes of the coordinate system and provides orientation of the alignment of the current coordinate system.

The user can classify the individual axes of the coordinate symbol by different colors that increase the clarity of this symbol.

In addition, the coordinate symbol can be faded in and out by the transparent command TICON (Toggle Icon).

## Crosshairs



The crosshairs indicate the current cursor position in the drawing. The crosshairs move with any mouse or digitizer pointing device movement. As with the coordinate symbol, the crosshair symbol changes to reflect viewpoint changes. In the above images, the 2-dimensional standard planview image changes to something similar to that on the right, when a 3-dimensional view of the drawing model is selected.

By Pointing or Identifying in a running command, you can determine points, distances and angles.

To set up the crosshair size, type in the name of the system variable CURSORSIZE.

## The Control Panel (Left Toolbar)

A control panel is situated on the left side of the screen. This side bar includes a permanent series of command or function buttons.



The bar's visibility is controlled by the variable DTTOOLAREA and indirectly by the DESKCONF command. Several optional displays are controlled by the DTTOOLAREASTYLE variable.

The symbols arranged there can be subdivided into groups as follows:

**Drawing tool symbols** - Explanations for the adjustment of these functions and their use in drawing may be found in the section *Users Guide – The Basics of Drawing – Precision Aids*

**Symbols to control the drawing views** - The explanation of these commands are given in *Users Guide – The Basics of Drawing – Viewing the Drawing*.

**Commands to control User Coordinate Systems and Layout Mode** - See the section *Users Guide – The Basics of Drawing – Coordinate Systems* for information on the usage of User Coordinate Systems and *Users Guide – Layout Mode / Paper Space* for information on the Layout Mode feature.

**Detailing Functions: dimensioning, crosshatching, text** - Dimensioning is discussed in the section *Users Guide – Detailing the Drawing – Dimensioning Variable Summary*, hatching in *Users Guide – Detailing the Drawing – Crosshatching*, and text editing in *Users Guide – Detailing the Drawing – Text*.

**Inquiry Commands** - The explanation of these commands is given in *Users Guide – Obtaining Drawing Information*

## Command Line and Text Window



The Command Line, or the Text Window, serves as communication between the program and the user. The command line's visibility and position is controlled by the variable DTCMDLINE and indirectly by the DESKCONF command.

In its default condition, the program uses a Command Line situated directly underneath the Menu Bar that looks like this.

The button on the horizontal toolbar will allow the user to switch between this mode of display and a larger Text Window, placed freely on the drawing field.

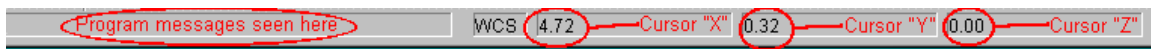
The default Command Line is fixed in its position on the desktop. It is possible, depending on DTCMDLINE setting, to change this. With the help of the scroll bar on the right side of the Text Window the registrations can be "rolled" up and down.

This field allows the user to:

- ▶ Specify points and values, select objects, and confirm inputs;
- ▶ Confirm the selection or input steps by the user (for example, *4 selected*);
- ▶ Receive messages, indications or warnings from preceding steps user (for example, *command not found*).

At the same time, the Text Window is fulfilling a recording function, the user can read or reconstruct all the program steps of the working session including all inputs made. Again, with the help of the scroll bars on the right side of the Text Window or with the keys Page Up or Page Down the inputs can be "rolled". When performing this function, the Text Window has the advantage of broader visibility.

## The Status Line



The Status Line is situated at the lower side of the visible desktop.

The line's visibility is controlled by the variable DTSTATUSLINE and indirectly by the DESKCONF command.

It contains a series of information fields. Brief "help texts" will appear in these fields. This line also contains the coordinates of the current cursor position.

The first information field from the left serves two functions:

When the Cursor is moved over the symbols of the symbol bars or palettes, a brief description of the commands and functions that will be executed by the chosen symbol is indicated. Information on the function of a command selected from any of the pull-down menu items is also displayed in this field. This information field will in addition, display the current distance and angle between the starting point drawing function and the current cursor position. This is helpful to the user when he or she is using the cursor to input the coordinates.

The next information field shows the currently used coordinate system.

WCS = World coordinate system

UCS = User coordinate system

The remaining three information fields specify the X-, Y- and Z-coordinates of the current Cursor position, in that order.

## Learning Center



## Introduction



# AllenCad 7.0 Learning Center

This series of "**Learning Center**" tutorials is designed to introduce you to some of the basic concepts of CAD drafting as well as getting you up and running with the program. If you are a more experienced user, you might want to skim through the topics to familiarize yourself with certain features that are unique to AllenCad software.

The tutorials introduce these concepts, through the use of brief explanations, examples, and graphics. Technical assistance is also available to registered AllenCad users through the company website at [www.ADSI.com](http://www.ADSI.com). As well, we welcome all comments and suggestions that will help us make this program more useful to you, the user and we have provided a submission form on the Tech Support pages for your convenience.

Wherever you see highlighted green or blue text such as in the previous paragraph, you have only to click on the text using the left mouse button to call the information onscreen. Text highlighted in blue is specifically designed to link you to Help sources on the Internet. If you have Internet access **and are currently online**, clicking on blue text will therefore link you to these information sources.

## Basic Concepts

### Configuration

#### Hardware Configuration

One of the first things you should ensure after installing the AllenCad program, is that your hardware settings are optimally configured for the program. AllenCad Help has a complete section devoted to Hardware Configuration. In addition, there are also sections on AllenCad Program Configuration and Drawing File Configuration.

Please click on the links provided, to access this information.

#### Program Configuration

Before beginning to work with AllenCad, you should ensure that the program is configured with

your drawing needs in mind. AllenCad Help has a complete section devoted to AllenCad Configuration and Concepts. Please click on the link to go to this section. Additional sections are also dedicated to Hardware Configuration and Drawing File Configuration.

## Drawing Setup Parameters

As you begin a new drawing, you will want to confirm the drawing parameters that will apply to that drawing. In addition to Hardware Configuration and AllenCad Program Configuration AllenCad Help has a section dealing with the concepts and commands used in Drawing File Configuration.. To access the information in these sections, please click on the links provided.

## General Concepts

### Use of Mouse Keys

In general, your mouse keys function more or less the same in AllenCad as they do in other Windows applications, except that a double click is not required to execute a command from a button or menu selection. A single left mouse click will do. Command initiation, onscreen point picking and entity selection then, is done with the left mouse button.

In general, also, a right mouse click is the same as pressing a confirming Enter key. Doing so, both indicates acceptance of a default value or "Yes" response to a prompt, as well as repeating the last command call if command completion was the last action performed. When "Mouse Context " is switched on, right-clicking on an object selects it for modification.

If you use a wheel mouse, the wheel will allow you to scroll through the text screen entries, both when the text screen is opened, and at the command line, when it is closed. In the drawing window, all panning and zooming is controlled by the Pan and Zoom commands. The wheel is therefore inoperative in the drawing window.

Users with three button mice may find that certain center button command assignments that have been configured in Windows, may function within AllenCad. This will be true only if the particular function is supported by the program.

## Opening and Saving a Drawing

To open a new drawing, you use the NEW command. This will present you with a blank drawing, upon which you may start to create your drawing "model". When opening a new drawing file, consider using a template drawing, whenever you are beginning a new drawing with standard drawing parameters.

To open an existing drawing file, you use the NEW command. Using this command, you can open existing .flx, .dwg , or .dxf drawing files.

When you OPEN a new drawing, you give the new file a name. Whenever you want to save the drawing elements you have created, you must use the SAVE command.

Often though, you will want to save the current drawing under another filename, format, or location. For this, you use the SAVEAS command. Note, however, that there are other formats in which drawing information can be saved. The following formats have their own specific file saving commands:

Windows Metafile format; Windows Bitmap Format; Slide format and saving selected entities.

## Command Entry

For most commands, AllenCad offers several methods of command entry. Commands may be initiated by selecting an option on a pull-down menu, selecting an icon on a toolbar or palette or directly entering the command name at the command prompt.

A series of commands can also be recorded in Macros (also called Script files) or programmed in LISP routines for automated execution.

See Also: Using Relative Coordinates

## Input Methods and Keywords

Coordinates and angles are specified in AllenCad in the same way as in AutoCAD. Similarly, object snaps and object selection made during command execution, are performed in the same manner as in AutoCAD. AllenCad also provides the same XYZ point filters for coordinate input.

Overall, AllenCad accepts the same special keywords you know from AutoCAD. The following table summarizes these methods and keywords:

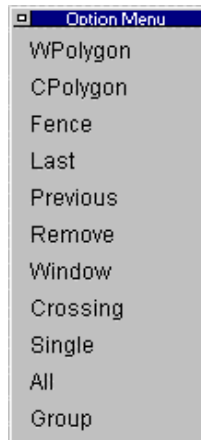
### Input Methods and Keyword Similarities Between AutoCAD and AllenCad

Input Method or Keyword	Comments
Press Spacebar or Enter	At the > command prompt, the spacebar or Enter key recalls the previous command. Within a command, the keystrokes accept the default option. In some cases, the keystrokes terminate the command.
@ symbol	The @ sign references the coordinates of the last point.
From	Establishes a temporary reference point for specifying a subsequent point. The From keyword is used in combination with other object snaps and relative coordinates.

Input Method or Keyword	Comments AllenCad Menu	ACAD Equivalent Menu
Object snap options:		
Int	<u>I</u> ntersection	Intersection.
App	I <u>m</u> plied Intersection	Apparent (implied) intersection.
App	<u>E</u> nd Point	Endpoint.
Mid	<u>M</u> id Point	Midpoint.
Cen	<u>C</u> enter Point	Center.
Qua	<u>Q</u> uadrant	Quadrant.
Per	<u>P</u> erpendicular	Perpendicular.
Tan	<u>T</u> angent	Tangent.
Ins	I <u>n</u> sertion Point	Insertion point.
Node	P <u>o</u> int Entity	Node (point entity).
Nea	<u>N</u> ext Point on Entity	Nearest.
	S <u>i</u> ze Snap <u>B</u> ox...	

Object selection options:

Wp  
Cp  
F  
L  
P  
R  
W  
C  
Si  
All



Window polygon.  
Crossing polygon.  
Fence.  
Last.  
Previous.  
Remove.  
Window.  
Crossing.  
Single.  
All objects in drawing.

Point filters:

X  
Y  
Z  
x,y  
y,z  
x,z  
x,y,z

References an existing point location and filters out unwanted X, Y, or Z-coordinate values.

Note that there is a small difference: In AllenCad, you do not type the point prefix; instead you type x, y, or z and (optionally) the numerical coordinates, as shown in the following examples:

From point: 5,y  
Y-Coordinate: 6  
To point: x,y,z  
X-Coordinate: 1.9  
Y-Coordinate: 2.8  
Z-Coordinate: 3.7

## Command Options


Users familiar with AutoCAD will feel immediately "at home" with most of the options that AllenCad presents for commands.

The primary difference from AutoCAD is the context-sensitive pop-up menu, called the *option menu* that displays the options available for the command. As an alternative, you can set up the *option bar* on which to display your options instead of the Options Menu.

## Canceling Commands

Sometimes you want to cancel a command before it has been completed. This can happen if you start a command that was activated by accident, or if the command automatically repeats itself.

To cancel the execution a running command:

- Click the symbol  (if you have the Option Bar configured) or
- Press the Esc key or
- Enter a new command.

## Selection Sets

When you need to identify objects for modification, AllenCad will prompt you to "Select

Objects". If you need to select a number of objects, individually picking them can sometimes be time-consuming. AllenCad offers you a dialog that lets you select multiple objects at once.



You can select from the dialog by picking the option with your mouse, or else typing in the capitalized letters (usually the first letter of each command). You will be offered an opportunity to make multiple option selections. After each option completes, the prompt Select Objects: reappears. The selection process terminates and all identified objects are added together on a "selection set", when this prompt is answered with a right mouse click, or pressing the Enter key,

**Note:** Be careful to clear any active Osnaps before using the options, as results might otherwise be unpredictable.

#### A. "Window" Selections

**Window** selection allows you to select all objects *that lie entirely within* a rectangular area that you will define onscreen by identifying two opposite corners of the rectangular area. The convention is that you will pick the left corner of the rectangle (either upper or lower) before dragging across to the diagonally opposite corner on the right. Picking and dragging in this fashion without making a dialog selection is assumed to be a Window (rectangular) selection.

A similar command for selecting objects lying *entirely within* an irregular area is the **Wpolygon** (Window Polygon) command. To define the polygon boundary with this option, you will pick a point on the boundary, then trace the boundary line, picking corners, in a clockwise or counter clockwise direction. Pressing Enter (or your right mouse button) will complete polygon definition and all objects within the boundary will be added to your selection.

#### B. "Crossing" Selections

Sometimes you will want to ensure that you are selecting *everything* within your rectangular window or polygon area, including objects that may lie only partially within the boundary.

**Crossing** (rectangular boundary) and **Crossing Polygon** (CPolygon - irregular boundary) are your options to Window and Wpolygon, to accomplish this. For crossing, be sure to select your right corner before selecting the diagonally opposite left corner.

#### C. "Fence" Selections

The "Fence" option selects all objects that cross a temporary polyline that you will be asked to draw onscreen. This option can be particularly helpful when trimming multiple objects.

#### D. "Last" Selections

The "Last" option automatically selects the last object that you created in the drawing.

#### **E. "Previous" Selections**

This option recalls the last defined selection set of objects used either in a previous command, or else created previously using the SELECT command. As noted above, you will be able to add additional objects to this previous selection by choosing subsequent options.

#### **F. "Remove" Selections**

Occasionally, we may make mistakes and include objects that we do not wish to include in our selection set. After all objects that we want to include have been selected, we can select the Remove option to deselect those objects that we don't want. Just as we can use different options in "Select mode", we can also use options in "Remove mode". The selection set assembly is considered by the program to be complete once all removals have been registered. Note that the Undo command does not function to deselect objects that have been selected. You must either **Remove** these objects or restart the command again from the beginning.

#### **G. "Single" Selections**

You might find an occasion where you want to pick only one object. Although you will seldom use it, this option ensures that you can pick only a single object at a time.

#### **H. "All" Selections**

Using this option will put all objects in the drawing (including those on locked or frozen layers) in your selection set. You can use this option in conjunction with the Remove option to achieve an "all except" selection set.

#### **I. "Group" Selections**

Where objects have been previously given a group identity using the GROUP command, these objects can be selected, using the group option and supplying the group name when it is prompted for.

### **Reversing Commands**




No matter how careful you are, you will find the Undo command a very good friend! Knowing that it is available may also give you the reassurance that you may need to experiment a little while you work.

If you recognize that you made a mistake, you can reverse the effect of (undo) the command. AllenCad keeps track of all steps you've taken during the current drawing session. This archive is called the undo list.

#### **Reversing a Command**

The UNDO command reverses each step sequentially. You can even keep using the UNDO command until you get back to the beginning of the drawing session. At that point, AllenCad warns, "Undo list is empty!"

To activate the Undo command, either:

1. Click the Undo button .
2. Select Undo from the Edit menu or
3. Type U at the keyboard and press Enter.

Some commands are not stored in the undo list. These include commands that control the viewport

(Zoom, Pan, and Window), set parameters, and other transparent functions. These steps cannot be reversed by the Undo command (however they have another command Zoom Previous to accomplish this task).


The undo list is kept only with the current session of the program. It is not saved with the drawing.

### "Undoing the UNDO (Redo)



If you undo too far, you can use the Redo command. It reverses only the last Undo operation, however. The drawing is restored to the last state prior to the Undo command – and this is only possible if there has been no other command executed between the Redo and Undo commands.

While running the Undo command, you can call for a Redo by either:

1. Clicking the symbol .
2. Selecting Redo from the Edit menu;
3. Or entering the Redo command at the command line.

### Transparent Commands

A series of commands and functions that control your view of the drawing or that provide information about the (about to be) selected objects are at your disposal even when another command has been called and is being executed. These commands are called transparent commands.

They include commands and functions for:

1. Changing the configuration of the drawing area (for example, 'ZOOM, 'PAN) or the window functions (for example, 'WCASCADE, 'WTILE);
2. Controlling precision ('GRID, 'SNAP, 'ORTHO, Object snap functions);
3. Switching between Command Line area and Text Window ('TSCREEN);
4. Loading or closing palettes ('PALMAN);
5. Calling the info functions ('EINFO, 'AREA, 'DIST); and
6. Calling the Help function ('HELP or '?)

After the execution of the transparent command, the program continues to execute the original command.

If transparent commands are called using the Command Line area, an apostrophe must be placed before the command for it to be treated transparently (for instance, 'ZOOMOUT).

Otherwise, transparent commands can be called as usual by clicking the proper option in the pull-down menus or by choosing the proper symbol, where the apostrophe is implied.

Transparent commands cannot be executed if:

1. The program is performing input of a sequence of characters; or
2. A dialog or selection window of another command is opened.

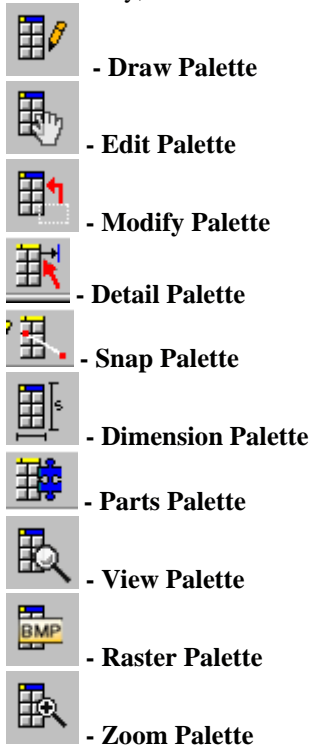
### Using Palettes

Palettes are callable toolbars that contain groupings of command icons that generally provide related functions. These palettes can be called onto the drawing window from the pull-down menus, called by the PALMAN command, or simply selected from the group of palette icons (shown below) that are located on the top toolbar:

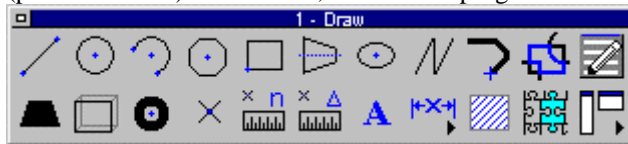


As with all desktop icons, holding a mouse pointer over them momentarily, can identify these icons, after which a tooltip will appear with the identifier for that icon.

Individually, the icons and corresponding palettes are as follows:



Each palette contains a group of related command icons, which can be called onto the edit window for ready access to the commands when you are performing related drawing duties. The Draw palette (pictured below) for instance, contains all program commands that relate to drawing geometrical objects.



This palette can be called whenever you require repeated access to these various commands and it can be dragged and dropped in any convenient location, as you move about and work on your drawing. When finished with it, you simply click in its close button on the top left corner of the palette header, and it closes.

## Zoom and Pan

As you begin to use the program, you will want to become familiar with The **Zoom** and **Pan** series of commands, as you will use them regularly when working on your drawings. The **Pan** commands are used to move the visible portion of your drawing in the editor. Pan enables you to view portions of the drawing that may currently lie outside the edit window. The Zoom commands allow you to "zoom in" or "zoom out" for a closer or more distant view of your drawing. Besides the specific Zoom and Pan commands listed below, a **ZOOM** or **PAN** command issued by itself will result in an options box being placed onscreen, allowing you to select a specific option. Both of these commands are Transparent commands.

### 1. Pan Commands:



As mentioned, the Pan commands allow you to move your view of the drawing in the edit window. These commands can be run from the View pull-down menu, the command icons on the left side panel or from the command line. The icons and command descriptions are as follows:



**PANUP** – Moves the drawing view upward.



**PANLEFT** – Moves the drawing view to the left.



**PANRIGHT** – Moves the drawing view to the right.



**PANDOWN** – Moves the drawing view downward.



**PAN** – Moves the drawing view in the direction and by the distance specified by two left mouse button clicks.



**DPAN (Dynamic Pan)** – This command changes the cursor icon to a hand image and allows you to repeatedly move the drawing view using the left mouse button to "grip and drag" the drawing.



**ZOOM BACK (Previous)** – Although technically not a Pan command, this command will return the drawing view to its previous condition if a Pan command is issued in error.

All of these commands with the exception of DPAN terminate after one execution and return to any previous command that may still be active. A second Pan command may be given as needed or by right-clicking with the mouse to repeat the command. DPAN commands may be terminated either by pressing Esc, clicking Exit in the option box that opens on your drawing editor or by issuing another command.

## 2. Zoom Commands:

Where Pan commands allow you to move your view about on a drawing, the zoom family of commands allows you to obtain a closer or more distant view of your drawing. As with Pan commands, Zoom commands are also transparent commands and are often used in conjunction with Pan commands.

The Zoom family of commands is available from the View pull-down menu, the Left Side Bar icons or through direct command entry at the command line. Note that if you only enter ZOOM at the command line, common options appear for selection in a box that appears on the desktop. You may select from this list, enter a zoom factor, enter a "p" (for previous), an "e" for extents, a "d" for dynamic, or simply specify a window using two mouse clicks on the edit window. The ZOOM commands include:



**ZOOMWIN (Zoom Window)** – This command zooms you in to a view that you define by picking a window using two left mouse button clicks.



**ZOOMFIT (Zoom Extents)** – This command zooms you out so that your view contains every entity in your drawing



**ZOOMIN** – This command zooms you in to 90% (a factor of 0.9) of your previous view. The new view is centered on the old view, so the part of the old view that you lose, is around the outside periphery.



**ZOOMOUT** – This command is the reverse of the ZOOM IN command. It zooms you out to 110% of the old view (a factor of 1.1). Again, the new view is centered on the old, so the additional view coverage is around the outside periphery of the old view.



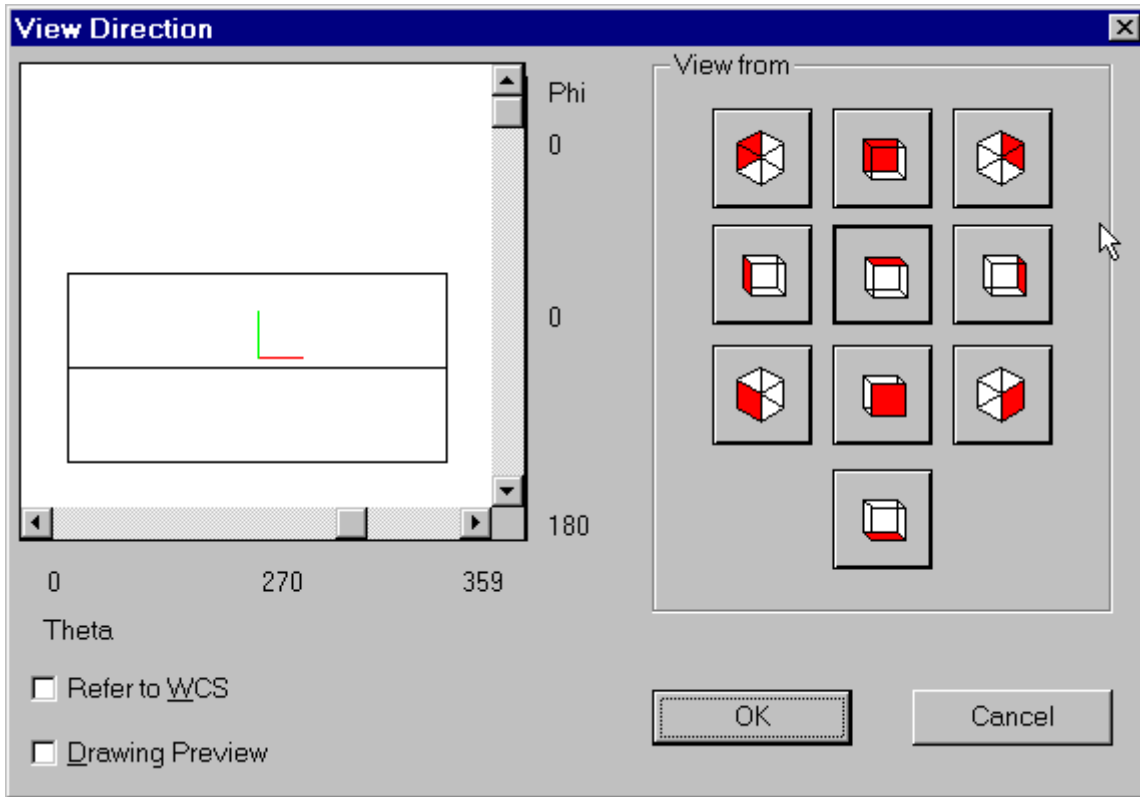
**DZOOM (Zoom Dynamic)** – This command allows you to enlarge or reduce the view coverage by an extent that is dictated when you click your left mouse button on the desktop, and holding down on the button, you drag in either an up or down direction. Dragging up (in the positive direction) zooms in on the drawing and dragging in a downward direction zooms out, with the new view centered on the old.



**ZOOMBACK (Zoom Previous)** – This command allows you to return to the previous view. When used repeatedly, you can step back through several previous views

### 3 Dimensional Views

By default, you view your drawing in "**Plan View**", or in other words, looking down on your drawing from above. From this vantage point, you are unable to visualize the 3-dimensional effects of your drawing as the third dimension extends into or out of your screen. To obtain another perspective, you can select the 3DView option on the **View** pull-down menu, and select one of the following presented options:



The red-shaded side on each option indicates the angle that you will be viewing your drawing **from**. To return to plan view, simply select **"Set Plan View to Construction Plane"** that is also located on the **View** pull-down menu.

You may use the sliders in this dialog to set view values for theta (Rotation IN the X-Y plane) and phi (rotation TO the X-Y plane) to achieve views other than the specific ones offered in the icon menu.

## Working With Layers

In complicated drawings particularly, it is usually desirable to group drawing entities on different layers. This allows them to be handled as a group.

Only one layer can be "current" at any one time and all objects drawn will be drawn on the current layer. Although default settings can be overridden, colors and linetypes are assigned to specific groups, so that entities drawn on that layer automatically inherit the assigned color or linetype specified for that layer.

Layers can be turned off or on to make their contents temporarily visible or invisible. They can also be frozen/thawed when longer-term visibility issues warrant. These capabilities can be very useful when editing a drawing, or when you wish to produce different drawings from a single drawing file. A drawing of a building, for example, may have individual floor and lot details arranged on different layers, so that only those drawing entities needed for specific drawings are visible when the drawings are printed. The ability to "mix and match" layers in this way also means that you only have to draw entities once, and they will be available for use in multiple

drawings.

Although you can move and copy entities between layers, the user is advised to have his layer use planned in advance, before commencing a new drawing.



On the Desktop above the drawing editor, you will notice the above bar. This is called the Properties Bar. Clicking the small green box icon on the left will open the Layer Manager dialog, where layers can be created and configured. To the right of this icon we have boxes that indicate the current layer (2, in this case), the default color of entities drawn on this layer and the default linetype that will be used when drawing entities on this layer. By clicking on the arrow boxes to the right (or on the color box itself), it is possible to change the current layer, color or linetype to be used.

## Orthogonal Mode



Orthogonal Mode limits the movement of the cursor so that it will only move parallel to the axes of the current coordinate system. This makes it much easier to design and place parallel, or "running", lines or objects.

In the standard World Coordinate System (WCS) the orthogonal lines will run horizontally or vertically from the base point.

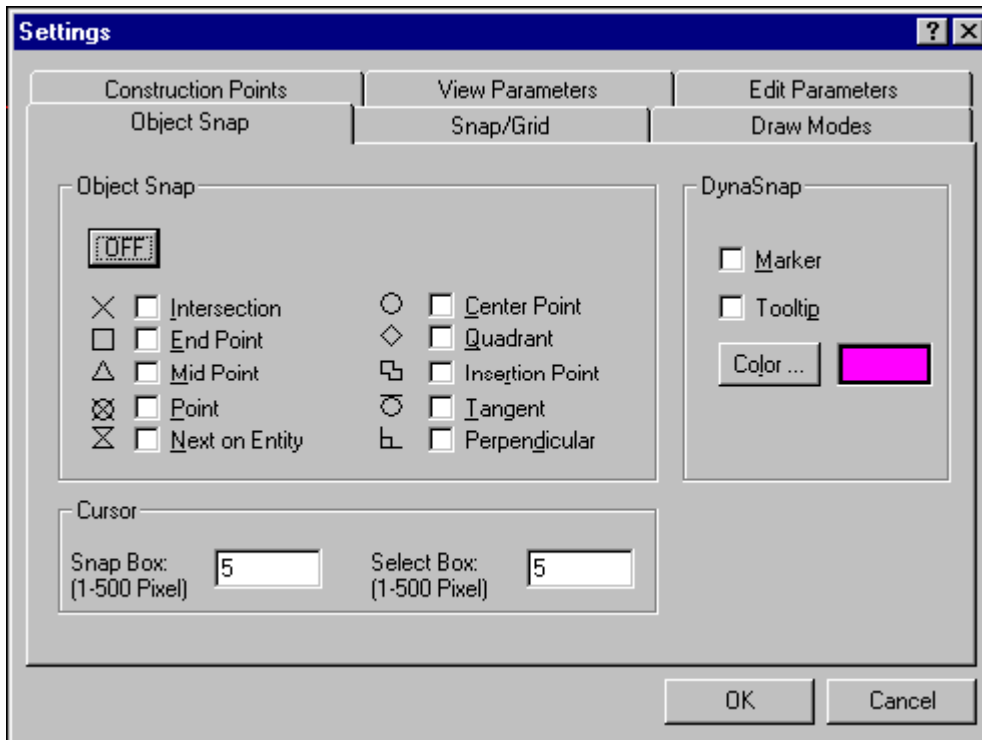
If Orthogonal Mode is activated, it is only possible to indicate points that are located on a line running parallel to one of the axes of the coordinate system. Such a line is also called an "orthogonal" line.

**Note:** Orthogonal Mode is effective only when "picking" points using the Cursor. Input of non-orthogonal points is possible by keyboard input while the orthogonal mode is activated.

## Object Snaps

Drawing accuracy depends upon how closely we can select the proper location for our drawing objects. When drawing these objects, we can often specify location by entering co-ordinate values for location, however, these coordinates are often unknown and it would take some investigative work to determine what the values might be. Very often too, objects need to be drawn in a location that is relative to other objects in the drawing. A line needs to be drawn from the center of a circle or to the midpoint of another line, for instance. To make drawing related objects simpler, AllenCad provides the user with precision tools known as Object Snaps (Osnaps for short).

Osnaps can be made active for use in repetitive command use or they can be called as required for a single application. To set them for repetitive use, you would select the Object Snaps tab on the Settings dialog after issuing the command SETTINGS. Alternatively, you can select "Settings" on the Options pull-down menu. The dialog that appears will be as follows:



Here, you can activate the osnaps that you will need by selecting the adjacent box or typing in the underlined letter in each snap name. The Off button will deselect all active settings. When osnaps are active, a pick box will be present on your cursor. This will typically appear after you have issued a command for which points must be input. Box size can be adjusted on this dialog as well. If you have the Dynasnap Marker selected, you will see the symbol for the snap point closest to your cursor position, when you have a drawing command active. This will indicate the snap that is closest to the present cursor position, and that is currently selectable.

There are also short-form option commands that will activate an osnap for a single use. You could type in "Cen" before picking a circle, if you wanted to specify the center of that circle, for example.

Object Snap Input Abbreviations:

1. Endpoint - **end**
2. Mid point - **mid**
3. Center point - **cen**
4. Quadrant - **Qua**
5. Intersection - **int**
6. Implied Intersection - **imp**
7. Perpendicular to - **per**
8. Tangent to - **tan**
9. Next point on entity - **nxt**
10. Insertion point - **ins**
11. Point entity - **pnt**

Command input for drawing a line from the center of a circle to the midpoint of another line would look like:

> LINE

Cen (To activate the center osnap)  
(Pick a Circle)  
Mid (To activate the midpoint osnap)  
(Pick a line segment)

That is all there is to it. You may wish to note these option commands for reference, until they have been memorized. Another point to observe is that **an option command will over-ride any osnap settings made on the settings dialog**. Thus, if the mid-point of an object and a perpendicular to that object appear to be very close together and both osnaps are active, you can simply use the option "Mid" to ensure that it is the midpoint that is picked.

## The Concept of Units

Before you begin a drawing, you want to consider both the measurement units that your drawing will be labeled with, as well as the measurement units you will be using for drawing input. Decimal mode will allow you to start with the assumption that one drawing "unit" or "measure" would be one foot, meter, or whatever you wished your labeled dimensions to represent. Another is feet and inches. Be careful not to mix two different units types, however. If you have some data measured in feet and other data to be used measured in meters, you may wish to convert one set of data to the other before beginning your drawing. Another option might be to have separate drawing files for the metric and English items. You can then apply a scale factor to one of the drawings to convert the units, before merging the drawings together.

Other unit formats available include Scientific (IE. 1.55E+01), Engineering (IE. 1'-3.50"), Architectural (IE. 1'-3 1/2") and Fractional (IE. 15 1/2). These formats should be specified from the outset and used consistently throughout your drawing development.

Units are specified, using the SETUP command.

## Using Blocks or Parts

Parts (or Blocks as they are sometimes referred to) are simply external drawings or internal collections of drawing entities within the drawing that have been given a part definition. After creation, these may be inserted repeatedly in a drawing, as may be required. This ability saves much repetitive drafting and significantly speeds up the drawing process. You may wish to consider saving drawn symbols, title blocks, etc. that you regularly use, in a separate drawing collection or library for future use with other drawings.

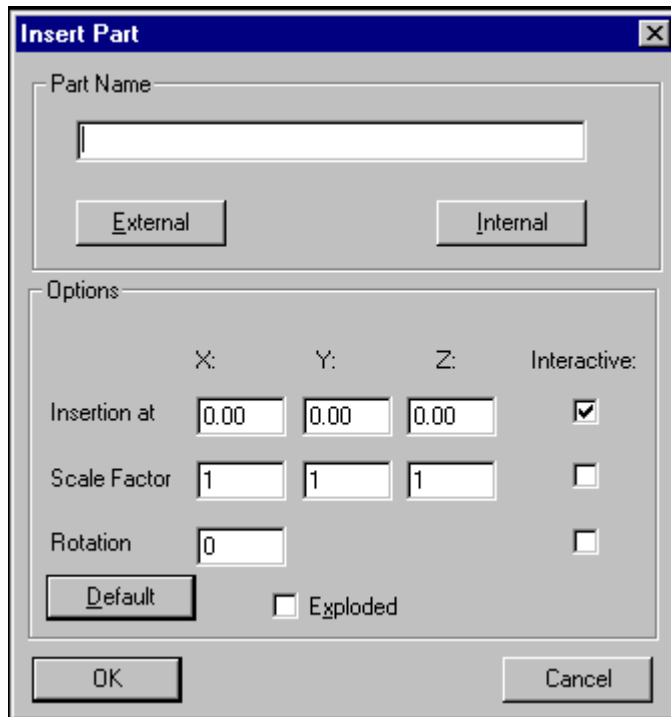
Blocks are treated as separate entities by the program and the component entities cannot be edited after they are inserted into a drawing unless they are first "exploded". Exploding a block causes each of the entities within the block to return into their original single entity status. After editing the entities, the same entities can be used to create a new block with the same name as the old block. This allows the user to automatically change every existing block with that name in the drawing, to the new format.

Note: Exploding a block does not change the block definition within the drawing. You can therefore insert blocks after exploding copies of the same block that were inserted earlier.

### Importing a Block

To import either an external block or drawing file, or an internal block that has been defined in the drawing, select Insert Part... on the **Parts** pull-down menu. You will be presented with the

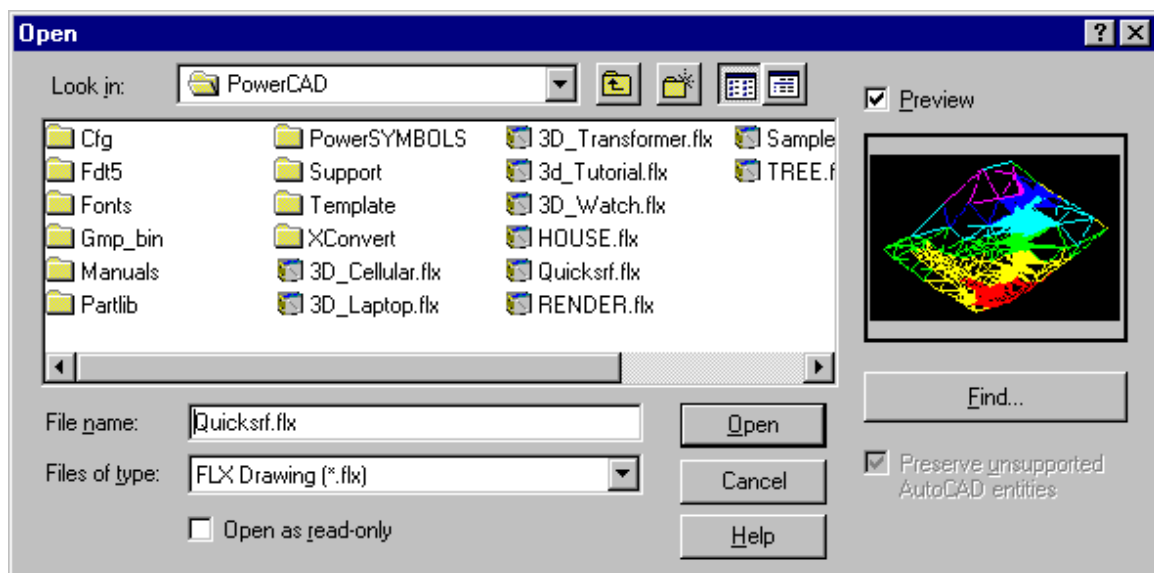
following dialog box:



In it, you will notice a space to enter the part's filename, as well as buttons to indicate whether it is an external file or an internal block that is defined in the current drawing file.

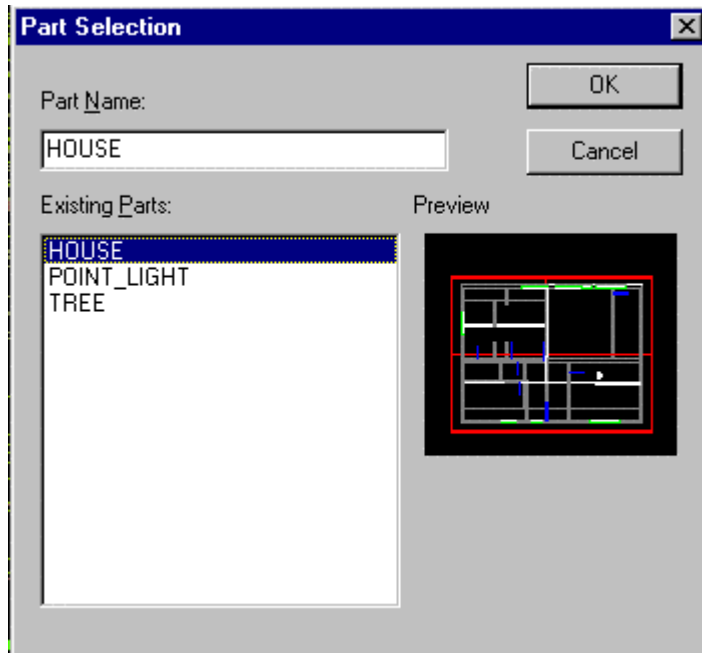
You can also change default values for insertion point, scale factor and rotation, as well as making the selection interactive during the insertion process. The last option, "exploded", allows you to explode the block as it is inserted.

If you have chosen External, you will be presented with the following dialog:



This dialog allows the user to browse for a specific file, as well as specify the file type being listed.

Choosing Internal, on the other hand, brings up the following box:



All parts that presently exist within the drawing are listed. Selecting one of them from the listing will produce a thumbnail picture of the block as shown. This is very helpful in confirming that you have selected the correct block, prior to insertion.

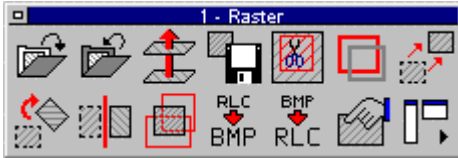
## Raster and Vector Graphics

In the program documentation, you will often see the terms "Raster" and "Vector" being used. Most CAD programs, including AllenCad produce a Vector graphics file, where the drawing is comprised of drawing "entities", being the individual lines, arcs, circles, etc. that make up the drawing and that are saved in a drawing "database".

Raster graphics files, on the other hand, are simply collections of individual dot information that can also represent drawing objects. As you might imagine, storing all the raster dot information takes considerably more space than saving simple vector information, such as a line's starting and end-point, etc. A great number of different raster data compression schemes have been developed to try and reduce the amount of storage required for raster graphic files. These different compression techniques each create a different graphic file format (IE. Files with extensions such as .gif, .tif .jpg, etc.). Fax and scanned images are examples of some of these raster formats and software can be obtained, that permits the conversion of one format to another.

When you are working with raster images, you will find all commands available to you on the Raster Palette.





This palette can be called by clicking on the Raster

Palette icon that is located on the horizontal toolbar.



### 1. Importing Raster Images

A few CAD programs, including AllenCad, have the ability to import certain raster image format files into the drawing editor. These image files come in as a single "picture" that cannot be edited (apart from scaling, stretching or moving about in the drawing editor). Typically, these images are used as "picture-in-fill" or as a temporary picture to be traced over, before being discarded. Tracing over scanned images in this way has, in recent years, replaced tracing images on a digitizing tablet as the preferred method of obtaining drafting information from paper images.

AllenCad will import raster graphic images in Windows Metafile (.wmf), Windows Bitmap (.bmp) as well as (.rlc) formats.

### 2. Exporting Raster Images

Besides being able to import images in these formats, AllenCad is also able to export drawing information in .BMP and .WMF formats as well. With this capability, you can export selected drawing entities, a section of the drawing, or the entire drawing directly to a new raster graphics file. Alternatively, you can send this drawing information to the Clipboard, where it can be imported by other Windows applications and modified or saved in other formats, as is available in your other Windows program.

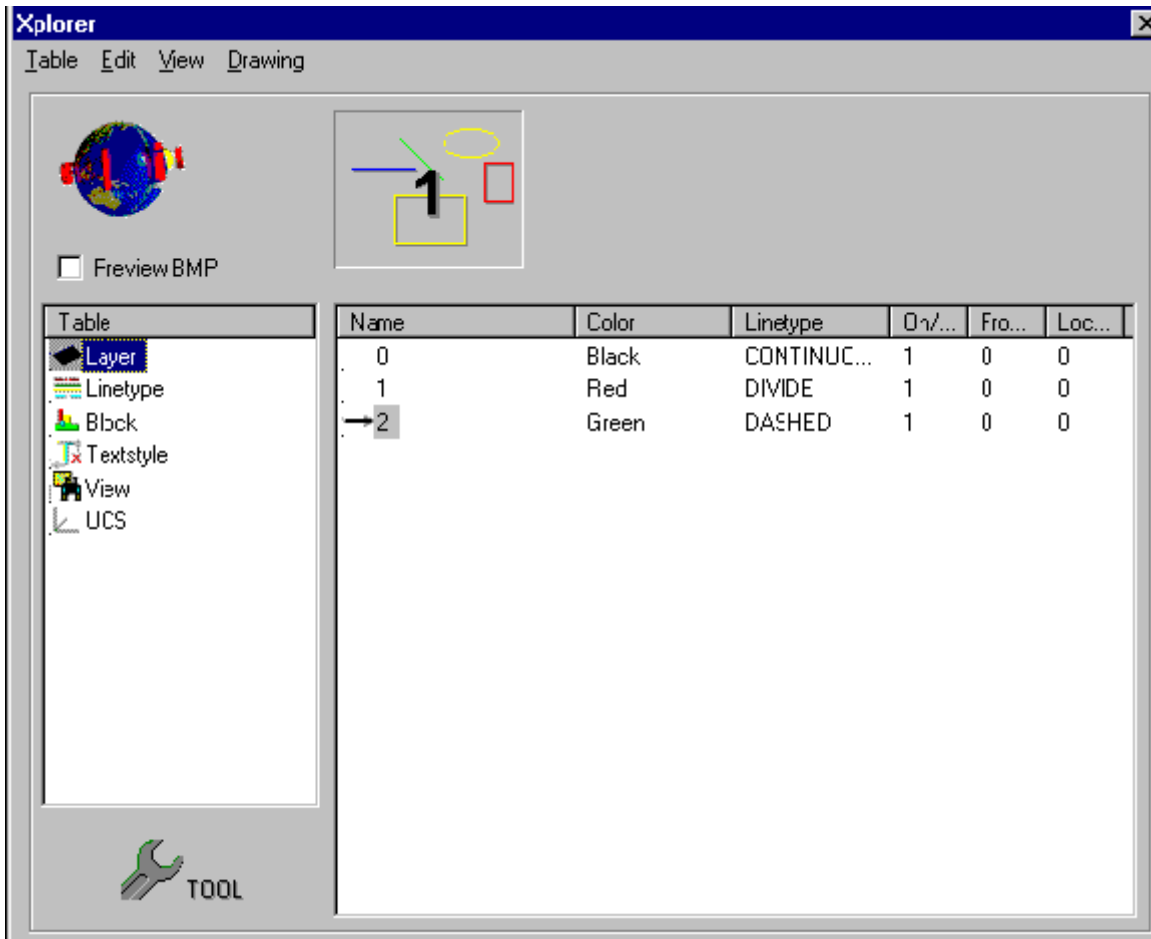
## The Drawing Navigator

### DRAWING XPLOER

The Drawing Xplorer allows you fast and uncomplicated access to important information about the currently open drawing(s). The Xplorer can be used to alter properties of the current drawings.

More precisely, the **Drawing Navigator** is a tool to manage the named objects of the drawing database tables. You can handle the tables and table entries of layers, linetypes, text styles, views, user coordinate systems, and blocks.

To activate this tool, type the command **Xplorer** or select **Drawing Xplorer** from the **File** pull-down menu. The Xplorer window displays a preview bitmap of the currently open drawings in the area under the menu bar. To switch to another drawing, click the appropriate bitmap. A tooltip over the bitmap displays the drawing name, including drive and path.



The Table list box allows you to select a specific table. The named objects for the selected table are shown in the Name list.

The Drawing Xplorer permits the following:

1. Delete unused, unreferenced table entries (allowed only for the active drawing).
2. Rename table entries (allowed only for the active drawing).
3. Set the layer, linetype, text style, user coordinate system, or named view (allowed only for the active drawing) by double clicking to one of the table entries.
4. Insert a block by double clicking to its name.

Click the Tool button to display the corresponding dialog box associated with the table name. For example, if Layer is the table selected in the Drawing Navigator, the Tool button displays the Layer Manager dialog box. This lets you create new layers.

## Grouping Entities

While working on your drawing, it is often convenient to select objects that have been given a group identity. Drawing objects can be members of more than one group and they also retain their individual identities. These objects can remain on different layers, for instance, and still be members of a named group.

To create a group, you first ensure that all member objects are drawn and visible in the drawing editor. You will then type the command **GROUP** at the command line, after which you will be presented with the option dialog box as below. As this will be a new group, you will pick **CREATE**.



You will then be asked to supply a name for this group as well as a description that may be used for later identification. At this point, you will be presented with the familiar object selection dialog:

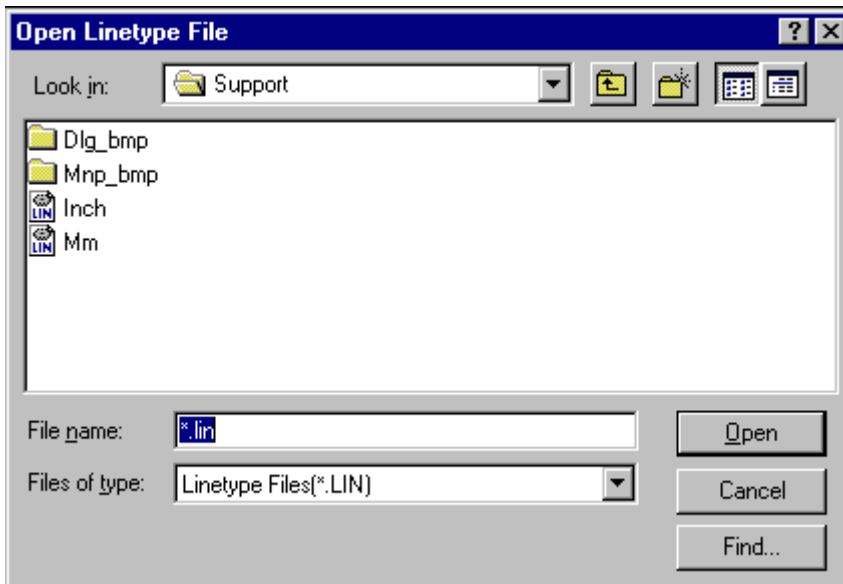


You may choose an option from the dialog or simply pick the objects to be made members of the group. Once the selection has been made, the group is formed and the group name can be used to select all members.

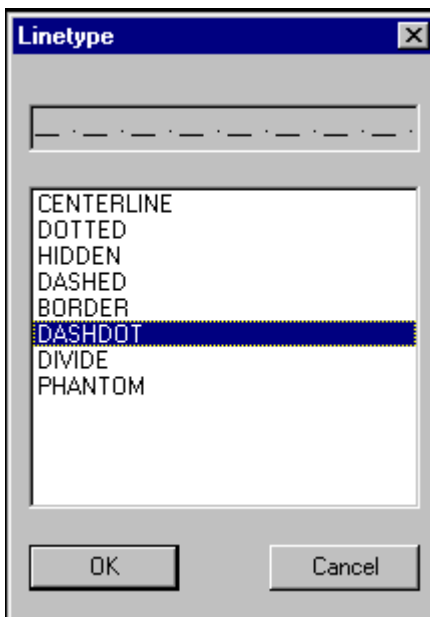
## Loading Linetypes

Linetypes can be assigned to linear entities through layer default (bylayer), object modification (PROPEdit command) or the currently set linetype when the object is created (SETLAYER command or Linetype icon on the Symbol Panel). When you start a drawing, AllenCad loads only a continuous linetype and a bylayer option for you to choose from. Other linetypes must therefore be loaded before they are available for use. To do this, select the "Load Linetype" option on the

Options pull-down menu (**LINETYPE** command) and you will be presented with the following dialog:



Here, you specify the linetype file that you want to open. A linetype file contains linetype definitions for a selection of individual linetypes. Here we see files containing Metric and English unit definition files and we will chose metric (Mm). We next see a dialog containing the linetype names in this file. There is also a window at the top of the dialog, that gives us a representation of each linetype as it is selected.



We simply select those that we wish to use and press OK to complete the loading.

## Controlling Linetype Display

The Linetype Scale Factor (**LTSCALE** variable) determines the scale factor for lines drawn with a linetype. The default is 1.0. A larger number results in extended line patterns (dash lines are longer); a smaller number contracts the line pattern (dash lines are shorter). To change the scale factor, click on the Draw Modes tab in the Settings dialog. This will be displayed after clicking on Settings in the Options pull-down menu.

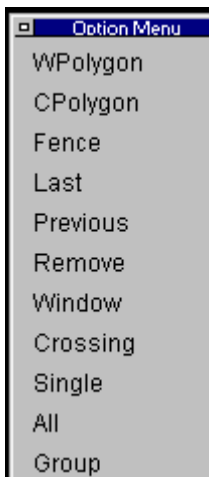
### Notes:

1. If the linetype scale factor is set to a very low number, dashed or centerlines may appear to be continuous type. In such cases, re-set the ltscale to a larger value and perform a regen afterwards to see the gaps in these types of lines.
2. The effects of a change in the LTscale factor will not be observed until the drawing has been regenerated, using the REGEN command.

## Hatching

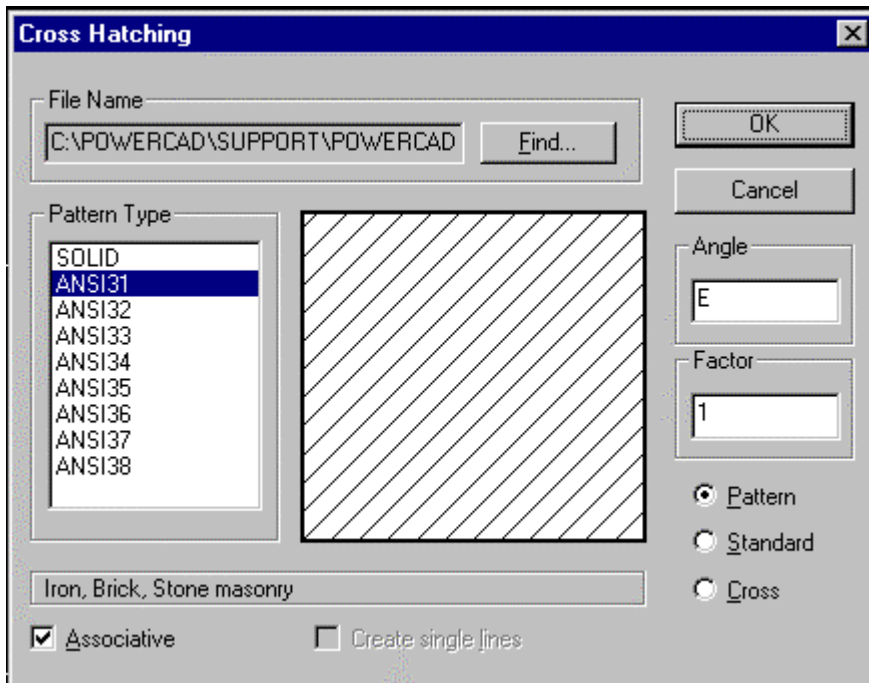
### ADDING HATCH MARKS TO AN OBJECT

Select **Hatch** from the **Detail** menu, or type the Hatch command at the prompt.



This command prompts you to select an object. The Option Bar contains a list of object selection functions. Select the object(s) then terminate object selection process by pressing Enter.

The **Hatch** command opens the Cross Hatching dialog box, which allows you to select a pre-defined hatch pattern, change its parameters, or to create your own pattern. In the center of the dialog box, the preview function lets you see the effect of your choices.



Select the pattern and confirm your selection with OK.

If you do not see the hatch pattern you need to use, click the Find button to select another hatch .pat file. To change the parameters of an existing hatch pattern, alter the Angle and scale Factor values, as required. Factor affects the spacing of the hatch lines, measured in drawing units.

To create a simple, user-defined hatch pattern, click the Standard or Cross buttons. This option displays a preview of a standard pattern consisting of continuous horizontal lines spaced at a distance of one drawing unit.

You can change this pattern via the Angle and Factor parameters. The Cross button adds lines at right angles to the existing lines.

The Associative button allows the hatch pattern to update automatically when the boundary of the hatch is changed.

To apply the designed hatch pattern to your drawing, exit the dialog box with OK.

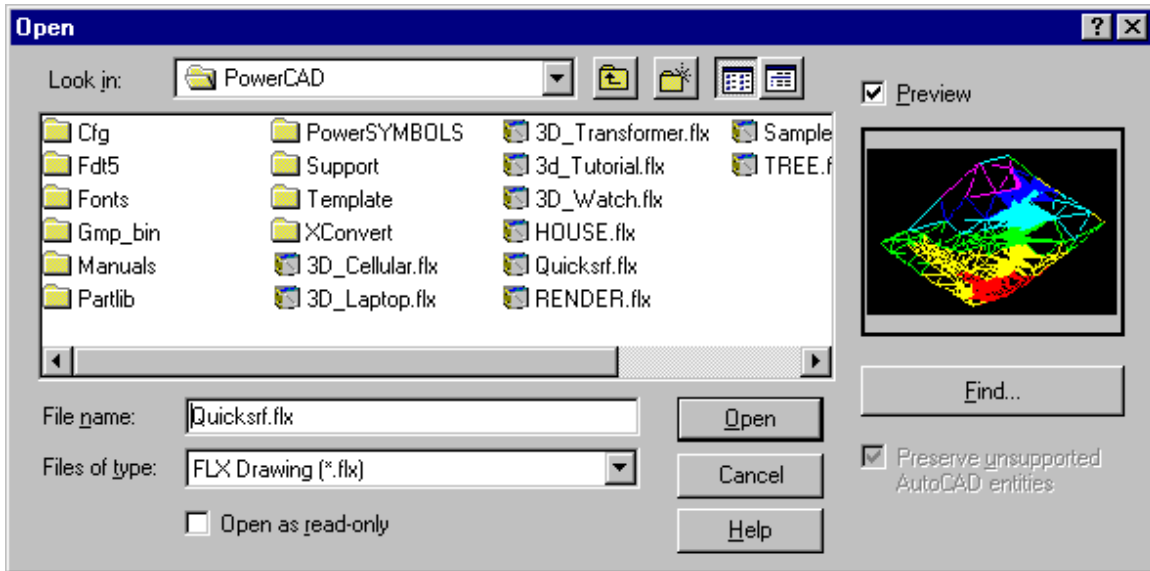
## External References (XRefs)

### XLink Command

The **XLink** command allows you to link one or more external drawings to the current drawing. A linked (or xref) drawing is treated very similar to a part (block) but there is a fundamental difference: an xref does not become a permanent part of the current drawing. The user may appreciate this feature, especially when working with very large files, or when a file is externally referenced to multiple drawings.

**Note:** Externally referenced drawings must be stored in the same location as the drawings that reference them.

The XLink command (Parts > External References) displays the dialog box. Initially, the list box is empty. Click the Attach button to select another drawing file to attach. You may select .flx, .dwg, and .dxf files.

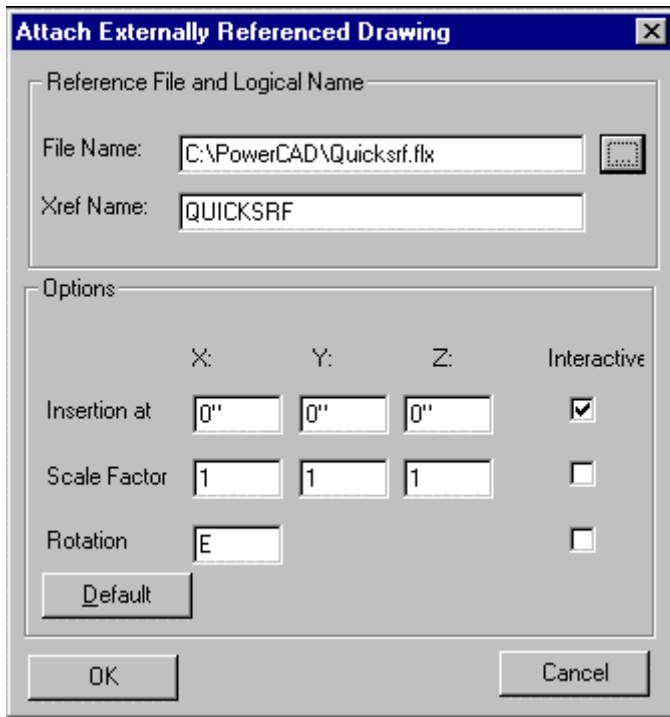


Once one or more drawings are attached, the dialog box displays the following information:

1. The logical name assigned to the xref. The logical name is a way of identifying drawings that have a filename longer than 31 characters (Windows allows filenames up to 255 characters long).
2. The drive, path, filename, and extension of the referenced drawing.

### **Attach**

The Attach option allows you to insert an external drawing as an xref to the current drawing. It displays this dialog box:



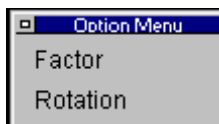
### Reference File and Logical Name

Type the name of the drawing in the File Name field, or click the ? button to select the drawing from a file dialog box.

You must type a logical name in the Xref Name field. This can be the same as the filename, if it is eight or fewer characters. The logical name must be short since it is used as the prefix for the names of dependant symbol names (layer names, linetypes, etc.).

### Options

The other options of the dialog box let you specify preferences for the insertion and are identical to those found in the dialog box of the Insert command: insertion point, scale factor, and rotation angle, as described earlier in this section.



Click OK and the drawing is attached. If any of the Interactive check boxes are on, you will be prompted for the insertion point, scale factor, and/or rotation angle.

## Printing/Plotting

Sending drawing information to an output device (or to a plot file) such as a printer or a plotter is accomplished through use of the **PRINT** command (File – Print). Any printer or plotter may be used that is attached to your computer or that is available for use through a network and that has been configured in your Windows operating system.



Additional capabilities such as the ability to assign printed colors and line widths to drawing file colors can be assigned through use of AllenCad's CONFIG command.

The **PRINT** command also allows you to specify an area to be printed or the scale to be produced on the printed drawing

See Also : Creating Plot Files and Plotting Plot (.plt or .prn) Files

## Creating Plot Files

This is strictly a Windows Printer setup procedure. First, you will need to know the particulars of the remote printer (make and model number) and you may also require a driver from the manufacturer, if Windows doesn't provide drivers for the printer/plotter.

With this information you will "Add a Printer in Windows' Printer folder (Start – Settings – Printer). During configuration, you will specify the make, model and print specifications, and when you are asked to designate the printer's port, you will specify that you are sending output to a file.

To send plots to file, you will either make that printer /plotter your Windows default, or else specify that particular printer for use. Each time you "plot you will be asked to supply the filename and location for your new plot file.

See Also: Plotting Plot Files

## Plotting Plot Files

To plot either a .prn or .plt file outside AllenCad, use Windows Explorer to drag and drop the file icon onto the appropriate printer icon in the Start – Settings – Printers group.

See Also: Creating Plot Files

## Using Layout Mode

One of the primary advantages of working with CAD software is that any drawing plan can be created in real world units. This means that six units measured in the editor can represent six feet, meters, etc, depending upon the units selected in Drawing Setup. Most drawings, however, eventually need to be printed on paper. Usually this means that the drawing needs to be plotted in a (usually) smaller scale.

For some drawings, you may need to print a drawing that incorporates different scales for different parts of the drawing. Examples include:

1. Drawing title blocks.
2. Display different views and regions.
3. Create details of the model.
4. Lay out part lists, legends, annotations, and instructions on the drawing sheet.

### Model Mode and Layout Mode

To serve to solve these tasks in a flexible manner, AllenCad provides two modes to handle a drawing: model mode and layout mode. These modes are sometimes known as **Model Space** and **Paper Space**, or **Print Space**.

The objects are first drawn in model space. In paper space, you create viewports to your model with different views and scales, and annotate the drawing with text.

How you use layout mode is determined by user and company CAD standards. Generally there are two methods applied to their use:

1. Keep the drawing borders at 1:1 scale and scale down the views to fit or
2. Make all views in layout mode at 1:1 scale and scale up the drawing boarder, text and dimensions to suit.

It is recommended you choose one method only.

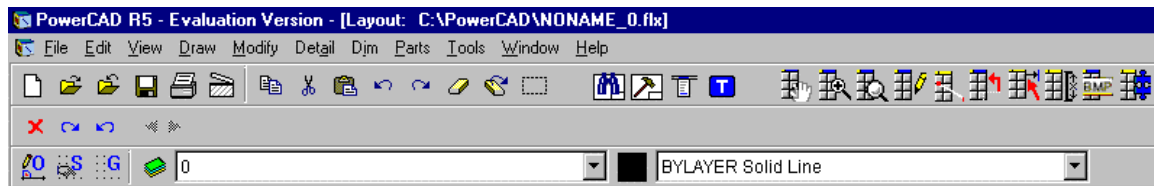
#### Notes:

1. Layout mode is not always the easiest way of finishing a drawing. If you create a pure 2D drawing, often it is easier to insert (on separate layers) title blocks, part lists, and details in model space, which are scaled to a larger factor to fit to a certain paper size. Some drawings are created easier at true scale.
2. When using layout mode, remember that this feature is provided to arrange and lay out a job for presentation and paper output.

### Toggle between Model Mode and Layout Mode

To switch from model mode to layout mode and vice versa, type the **TlMode** command (**File > Layout Mode / Model Space**).

When you switch from model to paper mode for the first time, the layout window is empty, which represents a blank drawing sheet. A single window replaces the tiled viewports of the current drawing (displaying the model in different views). The title bar at the top of this window shows the caption [**Layout:** and the drawing name].



This print space is not blank if your template drawing already contains paper space objects (in this example, paper space usually contains a company title block).

The layout mode performs two basic functions. It:

1. Determines the sheet size for printing, particularly by insertion of a title block.
2. Allows the creation of viewports that display individual views of the model.

You can also draw in paper space just as you do in model space. All the drawing and editing commands are available. It is also common to place drawing notations in paper space, primarily where orientation of these notations must be aligned to the viewports in which the particular section of the model appears. Since paper space is designated to lay out drawing output, a number

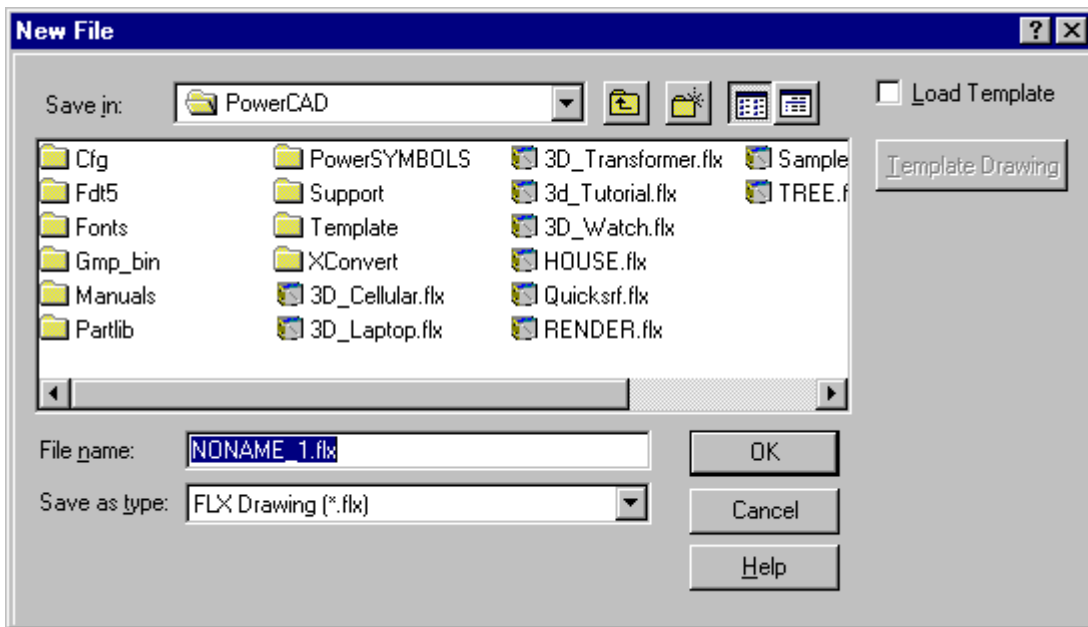
of 3D commands are disabled, including: **3DView**, **PlanView**, **Render**, **Hide**, **FHide**, **Light**, **LightEdit**, **SetViewDir**, **WOpen**, and **QWOpen**.

As described in the following sections, paper space allows you to arrange individual 3D views of the model by creating and modifying viewports.

## Template Drawings

If you have drawing standards or common elements that you wish to apply to more than one drawing, it is a very good idea to save all of these settings and elements in a drawing file containing just those elements. By specifying a **template drawing** when you open a new drawing, you will already be setup correctly with dimension variables, text styles, layer settings, title block, etc.

When you open a new drawing file, you are presented with the following dialog box:



Notice the **Load Template** check box in the top right-hand corner as well as the **Template Drawing** button immediately below. These control if a template is used when opening the drawing, and which template, if so.

You may find that the simplest way of using such a drawing would be to save it in your default drawing directory and to call it up as a "Template" whenever you open a new file needing the template's saved characteristics. You may store and use as many template drawing files as you want, in this way.

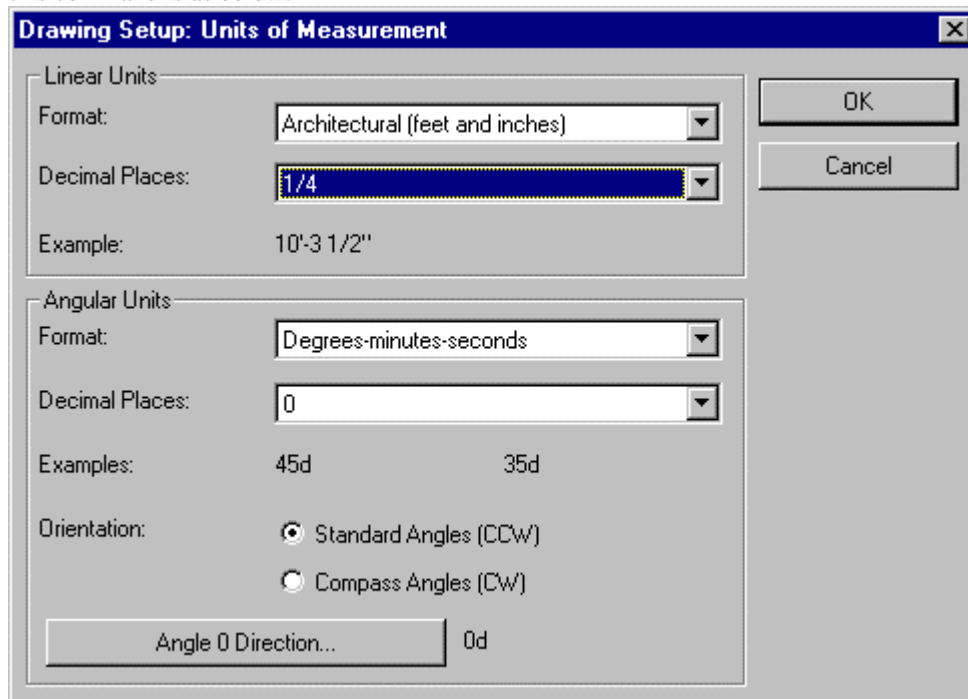
## Using Architectural Units

Whenever you wish to express drawing measurements in feet and inches (or your input measurements are in feet and inches), you should specify "Architectural" units in your drawing unit setup. With Architectural units, one drawing unit = one inch of "field" measurement. Measurements over 12 inches in length will be reported in feet and inches, however, input can be made in either in inches or feet and inches (IE 2'5", 29" or 29 are all considered equal measurements). If, on the other hand, you want all measurements purely in inches, you would specify either decimal or fractional units.

There are many instances in which you may wish to use Architectural units (feet and inches) and although they generally operate in exactly the same way as other unit formats, there are a number of issues to be aware of.

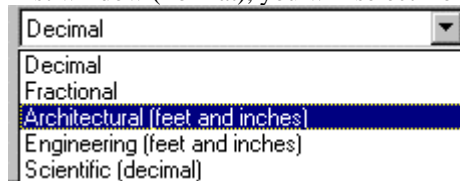
### Setting Up Architectural Units

To begin with, Architectural units are specified when you first begin a drawing, using the SETUP command (pull-down menu - Tools – Drawing Setup Parameters). The dialog that results when you use this command is as below:



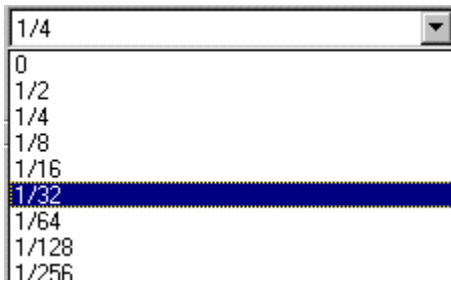
The image shows the 'Drawing Setup: Units of Measurement' dialog box. It has two main sections: 'Linear Units' and 'Angular Units'. In the 'Linear Units' section, the 'Format' is set to 'Architectural (feet and inches)', 'Decimal Places' is set to '1/4', and the 'Example' is '10'-3 1/2"'. In the 'Angular Units' section, the 'Format' is 'Degrees-minutes-seconds', 'Decimal Places' is '0', and 'Examples' are '45d' and '35d'. The 'Orientation' section has two radio buttons: 'Standard Angles (CCW)' (selected) and 'Compass Angles (CW)'. There is a button 'Angle 0 Direction...' and a text field '0d'. On the right side of the dialog, there are 'OK' and 'Cancel' buttons.

Here, you will use the buttons on the right side of the windows to present options for selection, and in the first window (Format), you will select from the list - Architectural (feet and inches).



The image shows a list of unit formats. The list includes: 'Decimal', 'Fractional', 'Architectural (feet and inches)' (highlighted), 'Engineering (feet and inches)', and 'Scientific (decimal)'.

The next window allows you to specify precision (fractions of an inch). Dimensions and distance reports will be to the precision that you specify here. Note that precision of up to 1/256th of an inch can be set.



Angular units are setup in exactly the same manner as other unit formats.

### Entering Architectural Units

There are a number of rules concerning architectural unit entry that you should be aware of:

Hyphens may only be used to between inch and fraction values

Decimal inch values may be used interchangeably with fractional inch entry

Spaces are not permitted. IE five feet six and one half inches may be entered as 5'6-1/2" or 5'6.5" but not 5' 6 1/2". Do not be confused by the fact that the program displays the dimension in the latter format, however.

If you are using Engineering input data (feet and decimals), you can use this input interchangeably with foot and inch entry (IE 10.75 or 10'9")

Points can be specified in relative coordinate fashion, in exactly the same way as other dimension formats.

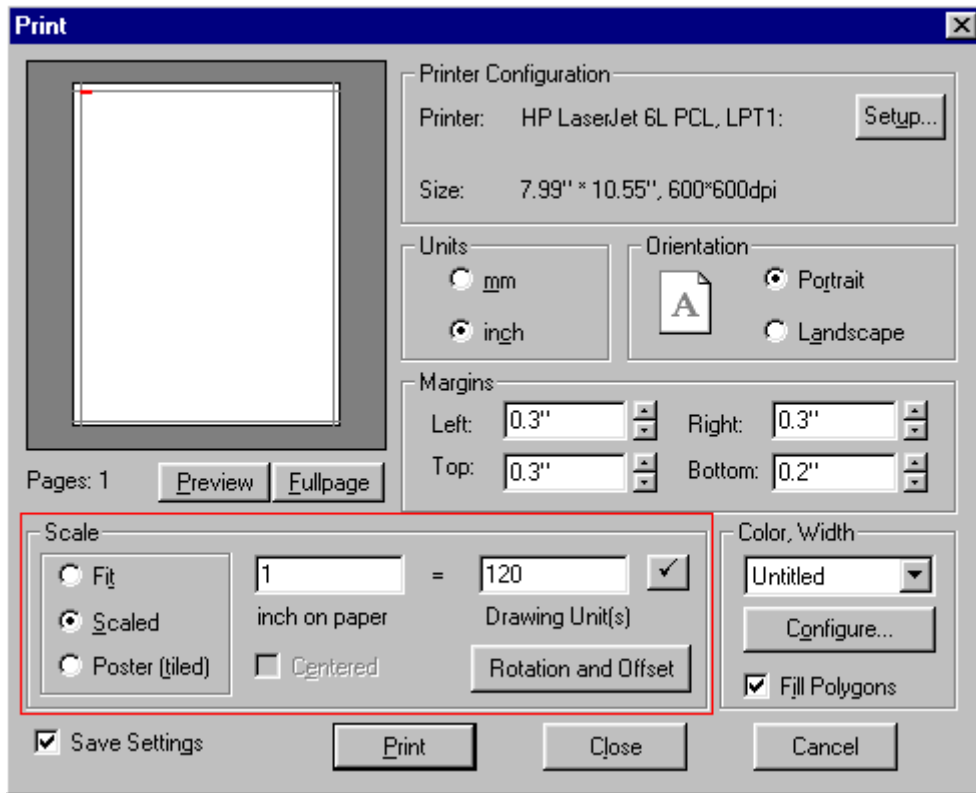
When you are specifying fractions, you can use a denominator of any size, however the precision factor that you have set will determine the value that the program actually uses. It will therefore "round up" the value to the precision setting, if necessary.

### Working With Drawing Scales

Normally, you will wish to create your drawing at a natural 1:1 scale, where one drawing unit equals one inch of actual measurement. This means that an object 6 inches long will be drawn 6 drawing units in length. This means that whenever you insert dimensions or take measurements in the program, the values will be numerically correct.

In most cases, the object that you are drawing is much larger than the size of the paper you will be printing on, so it is usually necessary to plot your drawing at an appropriate scale. If you determine that a scale of 1" = 10' would fit your drawing on the paper you are using, this would be calculated as 1 inch measured on the print = 10x12 = 120 drawing units (10 ft.x12"). This is also referred to as a plot scale of 1:120, meaning that your drawing must be shrunk down to 1/120th it's size to fit on the drawing sheet. Keeping this in mind, you should be sure that all text, dimensions, etc. are actually 120x the size in the drawing, that you want them to be printed at.

When you print, you ensure the units are indicated in inches, then you specify a scale as shown in the scale section of the print dialog as indicated below:



### Title Block Insertion

For those users who may require them, four basic Architectural drawing title blocks have been supplied in the \AllenCad\template directory. These have been scaled at 1:1 and are intended for sheet sizes "A" (8 1/2" x 11"), "B" (11"x17"), "C"(17"x22") and "D"(22"x34"). The drawing files have been named "Arch\_A\_Title", etc.. When you INSERT them into your drawing, you will specify the insertion point as being the lower left corner of your drawing frame and you will specify a scale factor derived from your plot scale. A 1"=40' plot scale would require a scale factor of 480 (which equals 40' x 12" = 480 or 1"=480").

As part of the title block insertion process also, you will be asked to provide text that will be inserted into the title block. To edit these values after the block has been inserted, you will have to explode the title (only once) and use the ATTDEDIT command.

### Selecting Text Sizes

Because you will be scaling your drawing down when you plot, you will need to make your text proportionately larger in the drawing than it will appear on the plot. In other words, if you will be shrinking everything down to one-tenth the size when you plot, text and other notations need to be ten times larger in the drawing than they will appear on your plot.

The following chart illustrates a number of common plot scales and the text scale factors required to obtain specific printed text sizes.

Scale	Plotted Text Size	Drawing Text Size	Text Scale Factor
1/8=1'-0	1/4	24	96X
3/16=1'-0	1/4	16	64X
1/4=1'-0	1/4	12	48X
1/2=1'-0	1/4	6	24X
1=1'-0	1/4	3	12X
1 1/2=1'-0	1/4	2	8X

1:2	0.25	0.5	2X
1:10	0.25	25	100X
2:1	0.25	0.125	0.5X

## Drawing

### Drawing Lines



The LINE command is used to draw lines. This command may be activated from The Draw – Lines pull-down menu or else entered at the command prompt. Lines are defined by the starting and the ending points, that can be entered either through mouse clicks on the drawing, or by entering x and y coordinate values of the points at the command prompt.

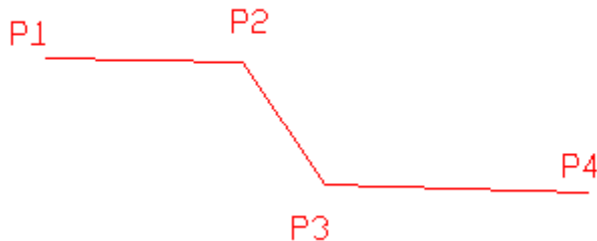
A point ahead may also be specified by entering an angle (bearing) and a distance from the start point or from the origin of the coordinate system. If you were to enter @7.5<90 when prompted for point #2, the resulting point would be at an angle (bearing) of 90 degrees from the first point and 7.5 distance units away. Dropping the @ sign and entering simply 7.5<90 would locate the second point a distance of 7.5 and at an angle (bearing) of 90 degrees from the origin of the coordinate system.

When the line command is given, four choices appear on the option bar:

1. Continuous
2. Segments
3. Append
4. TT

#### 1. Continuous Option

Continuous is the default option when drawing lines. The Continuous option allows you to draw line segments in sequence until the Line command is terminated, by pressing the right mouse button or the Esc key.



The Line command requests a "From point:" (start point) for the first line segment, and continues to request "To point:" until the command is terminated. Each subsequent point is connected to the previous point by a line.

From Point: **P1**

(Points are defined by the coordinate system chosen, with the first coordinate the X coordinate, then the Y coordinate, separated by a commas:)

To Point: 2.5,2.5

(+ 2.5 units in the X direction, and 2.5 units in the Y direction).

To Point: **P2**

To Point: **P3**

To Point: **P4**

When continuous line mode is chosen, the Undo and Close options are added to the option bar. The Undo option can be used repeatedly to erase the previous line segment, until the line is completely gone. The Close option connects the last drawn point with the start point, to create a closed shape.

To terminate the Line command, press Enter or Esc.

## 2. Segments Option

The Segments option creates single line segments, as opposed to a multi-segment (continuous) line.

As with the continuous option, the Line command prompts for the start and endpoints of the first segment. The program then repeatedly asks for start and endpoints. **Note the difference:** the lines drawn have start points independent of the previous endpoint; segments are usually not appended to previous line segments. To end the drawing of line segments, press Enter or Esc.

From Point: **P1**

To Point: **P2**

From Point: **P3**

To Point: **P4**

From Point: <hit Enter key>

To Point: **P4**

From Point: <hit Enter key>

## 3. Append Option

The Append option allows you to connect a new line to the end of an existing line or arc. Users familiar with AutoCAD will find this exactly the same as joining a line segment to another with the PEDIT command and the 'Join' option. When prompted, select the line or arc to connect with:

From Point: **append**

Select line or arc: **[pick]**

To Point: **[pick]**

The new line is connected to the end of the line or arc nearest to the pick point.

When appending the line to an arc, you are prompted the length of the line. The appended line is drawn tangent to the endpoint of the arc. You can either type the length, or pick the length with the cursor; for precision, it is better to type the length.

Select line or arc: **P1**

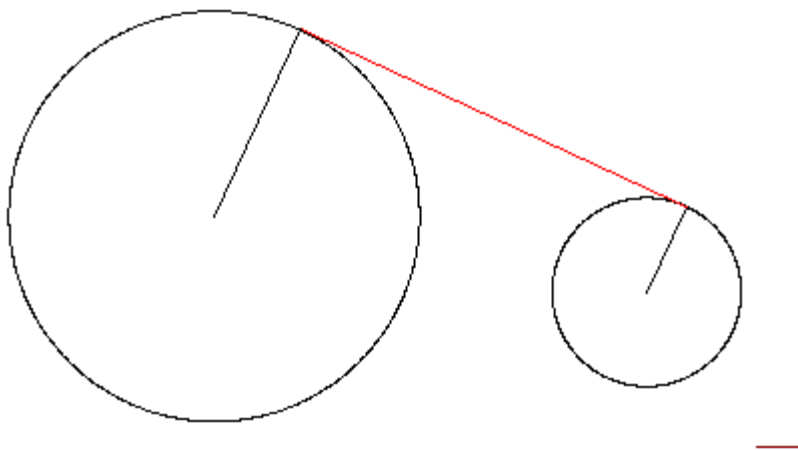
To point: **P2**

To point: <hit Enter key>

## 4. TT Option

The TT option is a special application that allows you to draw a line from the point of tangency on one curved surface to the point of tangency on a second curved surface. An example of this can be seen in the graphic below:





In this example, the red line is drawn using the TT option such that it is tangent to both circles. At these points on the circles, the line is at right angles (900) to the radius line drawn to the point of tangency.

### Close and Undo Options

After you select one of the Line command's option listed above, AllenCad displays the Undo and Close options on the option menu or bar. These options appear on the option bar whenever drawing a line, chain, 2D polyline, or 3D polyline.

#### Undo

The Undo option allows you to undo the last step taken. The command can be repeated to remove segments as long as a segment of the line exists. The last segment is erased with each successive Undo.

The endpoint of the previous segment can then used as a starting point of the next segment.

#### Close

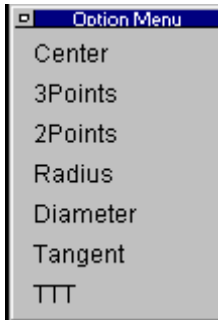
This option allows you to draw a closed polygon. Choose the Close option from the option bar or type C at the keyboard. AllenCad draws a line segment that connects the endpoint of the last line to the start point of the first line. (The Close option also allows you to create closed 3D polylines. )

**See Also:** Using Relative Coordinates for line definition by relative point entry.

## Drawing Circles



**CIRCLE** - Picking the Circle option form the Draw pull-down menu or entering the CIRCLE command will present the user with the following option dialog:



1. **Center** - With this option, the user is prompted to pick the center of the circle and to drag the radius out to a point through which the circle passes.  
IE.  
Pick center point: P1  
Pick point on circle: P2
2. **3-Points** - With this option, the user is prompted to pick 3 points through which the circle passes.  
IE.  
Pick point 1 on circle: P1  
Pick point 2 on circle: P2  
Pick point 3 on circle: P3
3. **2-Points** - With this option, the user picks 2 points that define the diameter and which lie on the circle.  
IE.  
Pick point 1 on circle: P1  
Pick point 2 on circle: P2 (That is co-linear with P1 and center of circle)
4. **Radius** - With this option, the user is prompted to input a radius value (or pick two defining points) and then to specify the circle center.  
IE.  
Either enter Radius: R  
Or pick 2 points defining radius length: P1 and P2  
Pick circle center: P3
5. **Diameter** - With this option, the user is prompted to input a diameter value (or pick two defining points) and then to specify the circle center.  
IE.  
Either enter Diameter: D  
Or pick 2 points defining diameter length: P1 and P2  
Pick circle center: P3
6. **Tangent** - With this option, the user is prompted to pick two entities (line, arc, polyline, etc.) that the circle is tangent to and then pick the center point.  
IE.  
Pick point 1 on a circle: P1  
Pick point 2 on an arc: P2  
Pick center point of new circle: P3

7. **TTT** - With this option, the user is prompted to select three linear or curved entities that the new circle will be tangent to.

IE.

Pick point 1 on circle: P1

Pick point 2 on circle: P2

Pick point 3 on line segment: P3

## Drawing Arcs

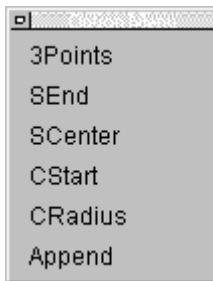


### The ARC Command

Just as there are several ways (geometrically) to describe an arc, AllenCad offers the user several alternate methods to draw the arc. The central command ARC is available from the "Draw" pull-down menu, or may be entered at the command line. The different command options in drawing arcs each use a separate selection of start/endpoint, radius, central angle, etc. that define the arc.

When drawing arcs it is important to understand that all **arcs will be measured to the left** (counter-clockwise direction), starting from the positive x-axis. To "pick" points for the construction of arcs, the user should always use the option OSNAP to set appropriate snaps before executing the command. If, for instance, a tangent to a line is needed to define the arc, the user might want to preset the "Tangent" osnap, or else specify "T" (for tangent) at the command line prior to picking the line tangential to the arc.

When the ARC command is issued, the following option dialog will be presented:



During input for each of these options, the user can effectively switch from one option to another by clicking on input options that are presented in secondary dialogs.

1. **3Points Option** - With this option, the user is prompted to enter 3 points - the start point, a point on the arc and the endpoint.

IE.

Pick start point: P1

Pick point on arc: P2 (moving anti-clockwise from P1)

Pick endpoint: P3 (moving anti-clockwise from P2)

2. **SEnd (Starting point - Endpoint - Radius/Angle) Option**

With this option, the user is prompted to pick the start and endpoints of the arc and then to input or define a radius by picking a point for the center of the arc onscreen.

IE.

Pick starting point: P1

Pick endpoint: P2

Enter Radius: Value (or pick center point).

### 3. SCenter (Starting point - Endpoint - Center point) Option

With this option, the user picks the start point for the arc, then the center point (thereby establishing radius) and finally the endpoint on the arc. Movement, as usual is in an anti-clockwise direction from start to endpoint.

### 4. Cstart Option

This option is a variation on the previous one, in that the user first picks the center point, then the start and finally the endpoint on the arc.

IE.

Center point: P1

Starting point: P2

Endpoint: P3 (Which defines central angle)

### 5. CRadius Option

CRadius also starts an arc by defining the center point. After that, the user is prompted to enter radius (or pick a point that defines radius), after which the user must pick the start and endpoints of the arc.

IE.

Center point: P1

Radius: r

Starting angle: P2

Ending angle: P3

### 6. Append Option

It is possible to append, or connect, arcs to lines or other arcs with the option Append. The arc is determined by two points, the starting point being the connection point, and the endpoint of the new arc.

IE.

Choose the existing line or arc: P1

Endpoint: P2

## Drawing Polygons



The NGON command, entered either at the command line or using the Ngon option on the Draw pull-down menu, will draw a polygon in the edit window. A polygon is defined as a regular multi-sided entity with sides of equal length.



IE. A 5-sided polygon with base at 0 degree bearing.

When the command is issued, the user is prompted to input the number of sides, with an option at this point to specify a line width. Next, the program prompts for either the side length (at an assumed 0 degree bearing) or else the picked location of the next corner. With this information,

the polygon is constructed.

IE.

Enter # of sides: # (optionally specify line width)

Pick first corner: P1

Either enter distance to next corner (assumed 0 degrees brg): Distance

Or Pick next (counter clockwise) corner: P2

## Drawing Rectangles



To draw a rectangle, enter the RECTANGLE command or use the Rectangle option on the Draw pull-down menu. When the command is issued, the user will be prompted to pick the first corner, followed by the diagonally opposite corner.

IE.

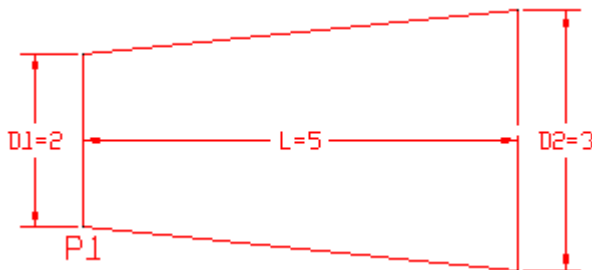
Pick corner 1: P1

Pick opposite corner: P2

## Drawing Trapezoids



TRAPEZOID - Trapezoids (as illustrated below) can be drawn by the program through the use of the TRAPEZOID command or selection of the Trapezoid option on the Draw pull-down menu.



After the command is given, the operator is asked to specify the insertion point (the first corner) and then the angle of the diameter (with default zero degrees). The diameter of the first side (D1) is then prompted for then length (L), and finally the diameter (D2) of the second side.

IE.

Pick Insertion point: P1

Enter insertion angle (0 default accepted above): A degrees

Enter Diameter 1: D1

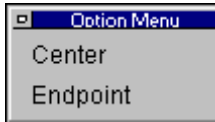
Enter length: L

Enter diameter of side 2: D2

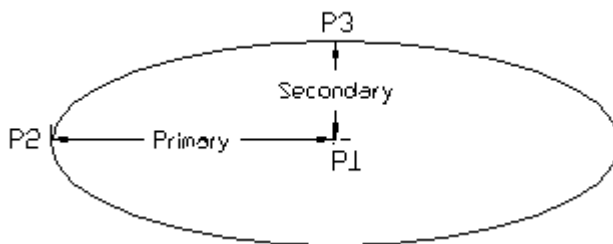
## Drawing an Ellipse



**ELLIPSE** - Using the command ELLIPSE or the Ellipse option on the Draw pull-down menu, enables the user to simply draw an ellipse. On entering the command, the program first prompts for the center point of the primary radius. At the same time, an option dialog is presented:



This dialog makes it possible to input the ellipse endpoint as an option to primary radius center. Next, the program prompts for a picked point that establishes the primary radius. Finally, you are prompted to pick a point that establishes the secondary radius and which also completes ellipse construction.



IE.

Pick Center Point/Endpoint: P1

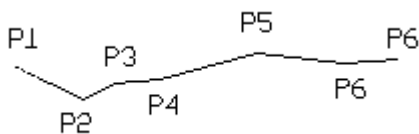
Pick primary radius: P2

Pick secondary radius: P3

## Drawing a Chain



**CHAIN** - The CHAIN command is a combination of the commands Line and Polyline.



The Chain command makes it possible to draw continuous lines in a manner similar to the command LINE using the option Continuous. The procedure is the same as drawing continuous lines. IE Pick Point 1, Pick Point 2, Pick Point 3.... etc.

There are, however, some important differences in drawing lines, polylines, and chains. A chain is like a polyline as the program regards the entire line as a single entity. Like a polyline too, it can be assigned width, but unlike a polyline, only a single width can be specified throughout the chain's length.

Command Input:

First point: P1

Next point: P2

Next point: P3

- /=F1. Width** - The option Width allows the user to define the width of the entire line. The width must be specified before entering the first point. Picking or entering the first point is understood as a verification of the predetermined value.
2. **Undo** - The option Undo allows the user to undo the last step taken. In other words, the last-drawn segment will be erased. The endpoint of the previous segment is then used as a starting point of the next segment. Undos can be repeated as long as a single segment exists.
  3. **Close** - The option Close allows the user to draw a closed polygon. The two endpoints of the chain will be connected to each other and the command will be terminated

## Drawing a Polyline



**POLYLINE** - Polylines are drawing elements that consist of connected lines or connected arc segments. Different elements can be of different widths. The width of a polyline segment can be defined differently for the starting- and end-point.

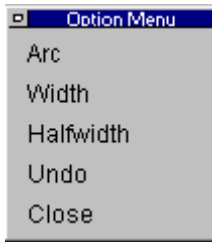


A polyline is defined as one element and treated as such for further operations (like Edit, or Undo), in contrast to continuous lines drawn with the command LINE.

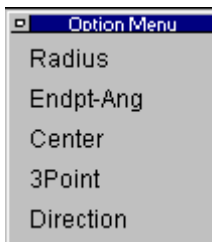
The POLYLINE command can be entered at the command line, or selected from the Draw pull-down menu or Draw palette. When the command is executed, you will be asked to pick the first (start) point and presented with the following dialog:



Here, you can specify the line width (at beginning and end of segment), append to an existing entity, or specify a tangent-to-tangent polyline segment.



After responding to a prompt for a second point on the polyline, you will be offered this dialog that enables you to add an arc segment, change widths, backup a step (Undo) or to join up with the beginning point on the polyline. Choosing the arc option will result in the following option dialog being displayed.



These are options for creating the arc. Should you wish, you can simply pick the arc endpoint while the program shows you a running image of the resulting arc, as you move to set the endpoint.

See Also: Drawing 3-D Polylines

## Drawing a Polyline Contour



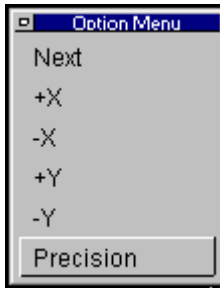
**PCONTOUR** – The PCONTOUR command creates a closed polyline on top of (overlying) objects making a closed boundary. Such a polyline contour can serve for hatching regions of the drawing with the HATCH command or to determine the area or circumference of a region with the AREA command (applying the option Object).

Make sure that you are on a unique layer before using this command, as you may have difficulties selecting your polyline contour, otherwise.

From the menu Draw, choose Polyline Contour or at the command prompt, enter PCONTOUR to issue the command. You will then be asked to select objects that define the boundary to be created. Carefully selecting objects at this point will ensure that you get the boundary that you want.

After the selection set to be examined has been picked, you will be offered the following option dialog and asked to pick a point within the required boundary:





The program establishes the starting object from your selection in this dialog. Generally the start object is not of any significance and you can simply accept the default by pressing Enter or right-clicking with your mouse.

- **Next** sends radial rays from the pick point you specify (see below) to the nearest object.
- **+X** (the default) sends a linear ray in the positive X direction from the pick point you specify to find an entity.
- **-X** sends a linear ray in the negative X direction from the pick point you specify to find an entity.
- **+Y** sends a linear ray in the positive Y direction from the pick point you specify to find an entity.
- **-Y** sends a linear ray in the negative Y direction from the pick point you specify to find an entity.

Next, you are prompted to pick the point to start examination of the given selection set.

Once a boundary object has been found in the direction of the specified pick point, the command traces the boundary in a counterclockwise direction.

After analyzing the specified selection set successfully, a closed polyline is produced covering the boundaries of overlapping drawing entities.

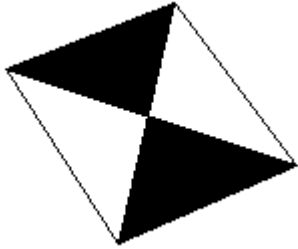
## Drawing Filled Faces



**2DFACE** - The 2DFACE command is used to fill in triangular or polygon-shaped areas in your drawing by picking corner points. You can think of it as coloring these areas with an entity that is on the current layer and with the currently set color.

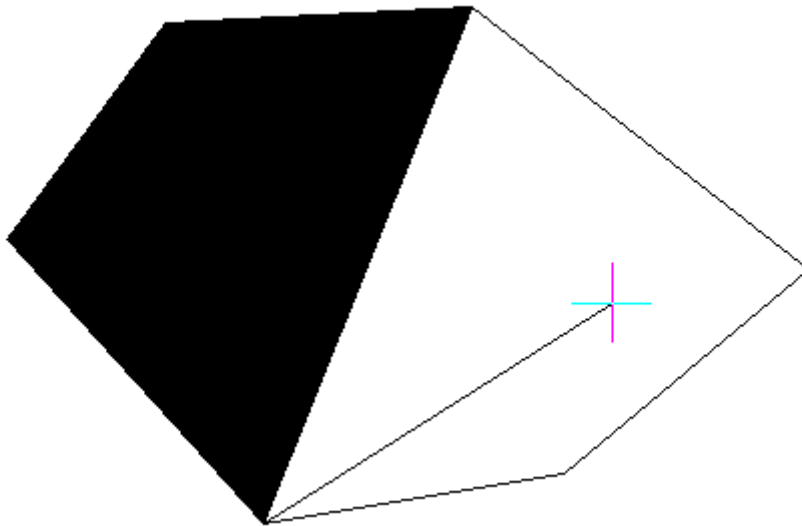
As with most commands that use picked points, you can work more quickly and accurately with existing entities when you use Object Snaps either set manually with each pick or in advance, with set snaps.

To fill in triangular areas, simply issue the 2dface command, then pick the three corners before choosing the close option to complete the fill. For Polygon areas, the pick order is very important. You must pick one side (IE top left - top right) and then the opposite side (IE. bottom left - bottom right) in the same order (in this case left then right). If you were to pick in a clockwise or anti-clockwise direction, you would produce a butterfly shaped fill as shown below, not the polygon shape that you are after.



#### Options:

1. **Continuous** - Often, you will want to fill in an irregular area that is bounded by many straight sides. One option when using the 2DFACE command is the Continuous option. With this option, the last 2 points picked for the first polygon fill are automatically used for the first two points of the next polygon fill. By starting on one side and working your way across the area, you can more easily accomplish this task.



2. **Append** - Another option, Append, allows you to pick one side of a drawn 2Dface to be points 1 and 2 in an adjacent 2dface. After picking points 3 and 4, you are (by default) put back into continuous mode. Pressing Esc terminates the command, as will picking another command to execute.
3. **Segments** - The Segments option simply allows you to draw one 2dface (triangular or polygon-shaped) before the command terminates.

See Also: Drawing 3D Faces

Drawing Rings (Donuts)



**RING** - The RING command is used to draw donut-shaped entities with a fill applied between inner and outer circles. When an inner diameter of zero units is specified, the entity becomes a filled circle.

When the command is issued, a dialog box appears with the Diameter option.



Unless this option is selected, the default values (the last entered values) for inner and outer diameter are assumed. Picking the Diameter option allows you to input new values. Once diameters have been established, the final variable (the location of the ring) is determined when the user picks the center point of the circles. The command is terminated at this point.

## Drawing Construction Points



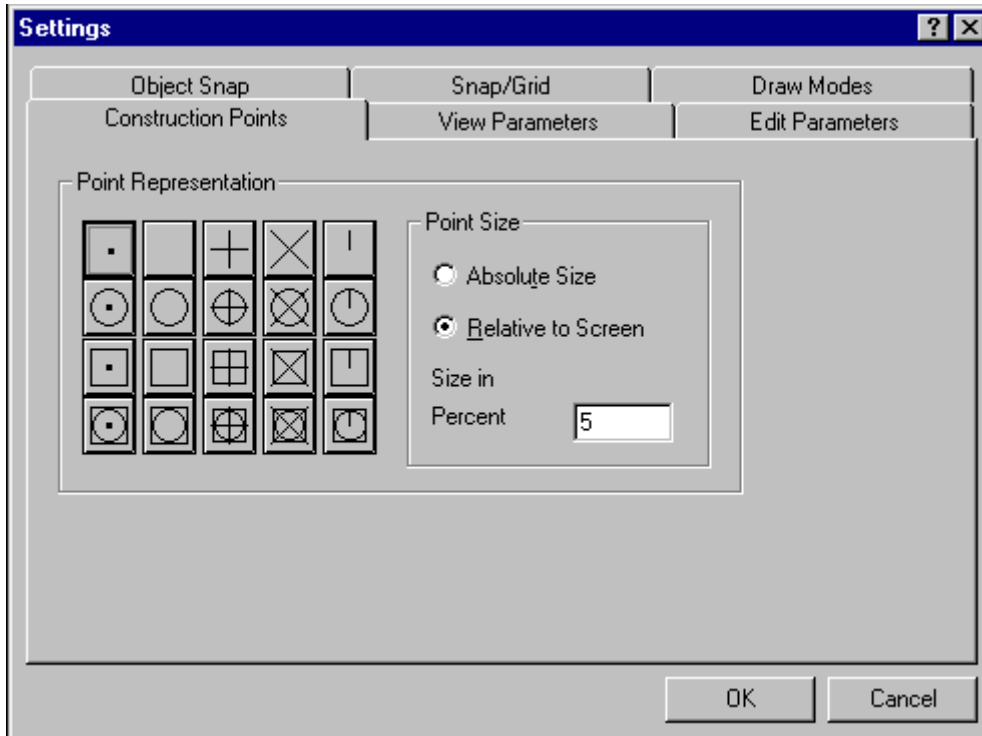
**POINT** - Individual points can be entered into a drawing using the POINT command. Usually these points are used as a reference points for a later drawing exercise, such as drawing 3 dimensional polylines, etc. The POINT command issued from the Construction Points option on the Draw pull-down menu will result in the following option dialog.



When the **point** option is selected, the program waits for input to determine the location of the point, either by a pick with the mouse, or else by coordinates entered in the format X,Y,Z that define the point location.

Both the **Measure** and **Segments** options prompt the user to pick a linear object, such as a line, polyline or arc. Measure will ask for the dimension between points for insertion along the object. Segments will ask the user how many segments the length of the object should be divided into in determining point spacing along its length. Note that these commands do not affect the original linear entity in any way. The linear entity is only used to help define the location at which points are inserted into the drawing.

How these points can be represented on your drawing can also be selected from the Construction Points tab in the Settings selection on the Options pull-down menu. When you make this selection, you will see the following dialog:

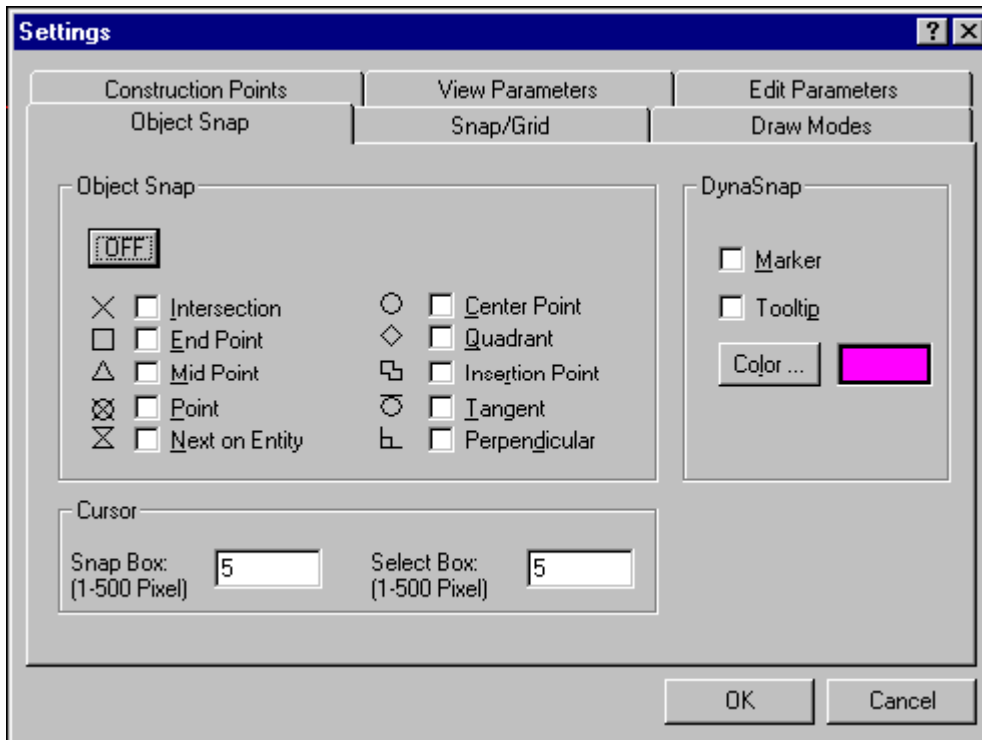


Here, we can see that the standard decimal symbol is currently selected. This can be changed to another, simply by selecting it from these choices.

**See Also:** Using the Divide Command  
 Using the Measure Command  
 Object Snaps

Drawing accuracy depends upon how closely we can select the proper location for our drawing objects. When drawing these objects, we can often specify location by entering co-ordinate values for location, however, these coordinates are often unknown and it would take some investigative work to determine what the values might be. Very often too, objects need to be drawn in a location that is relative to other objects in the drawing. A line needs to be drawn from the center of a circle or to the midpoint of another line, for instance. To make drawing related objects simpler, AllenCad provides the user with precision tools known as Object Snaps (Osnaps for short).

Osnaps can be made active for use in repetitive command use or they can be called as required for a single application. To set them for repetitive use, you would select the Object Snaps tab on the Settings dialog after issuing the command SETTINGS. Alternatively, you can select "Settings" on the Options pull-down menu. The dialog that appears will be as follows:



Here, you can activate the osnaps that you will need by selecting the adjacent box or typing in the underlined letter in each snap name. The Off button will deselect all active settings. When osnaps are active, a pick box will be present on your cursor. This will typically appear after you have issued a command for which points must be input. Box size can be adjusted on this dialog as well. If you have the Dynasnap Marker selected, you will see the symbol for the snap point closest to your cursor position, when you have a drawing command active. This will indicate the snap that is closest to the present cursor position, and that is currently selectable.

There are also short-form option commands that will activate an osnap for a single use. You could type in "Cen" before picking a circle, if you wanted to specify the center of that circle, for example.

Object Snap Input Abbreviations:

1. Endpoint - **end**
2. Mid point - **mid**
3. Center point - **cen**
4. Quadrant - **Qua**
5. Intersection - **int**
6. Implied Intersection - **imp**
7. Perpendicular to - **per**
8. Tangent to - **tan**
9. Next point on entity - **nxt**
10. Insertion point - **ins**
11. Point entity - **pnt**

Command input for drawing a line from the center of a circle to the midpoint of another line would look like:

> LINE

Cen (To activate the center osnap)  
(Pick a Circle)  
Mid (To activate the midpoint osnap)  
(Pick a line segment)

That is all there is to it. You may wish to note these option commands for reference, until they have been memorized. Another point to observe is that **an option command will over-ride any osnap settings made on the settings dialog**. Thus, if the mid-point of an object and a perpendicular to that object appear to be very close together and both osnaps are active, you can simply use the option "Mid" to ensure that it is the midpoint that is picked.

## Using Relative Coordinates

To this point, we have been defining points using absolute (X,Y,Z) coordinates. At times, however, it is faster and more convenient to define a point location relative to another point in the drawing.

Take for example, a situation where you are drawing line segments and you know the length and angle (bearing) of each vertex relative to the preceding vertex. You could calculate all of the vertex coordinates in advance and enter them in absolute coordinates as you draw your line, or you could more simply use relative coordinate input, to define each point relative the previous point.

Two symbols are used when using relative coordinates. The @ symbol indicates that the following coordinate entry is **relative** to the last picked or entered point and < indicates that the number immediately following it is an angle. **All values that follow @ in these expressions are relative to the previous point.**

**Note:** it is acceptable to mix relative and absolute expressions with subsequent points. In other words, you could use a relative expression to establish point 2, an absolute (x,y,z) expression to fix point 3 and another relative expression to locate point 4.

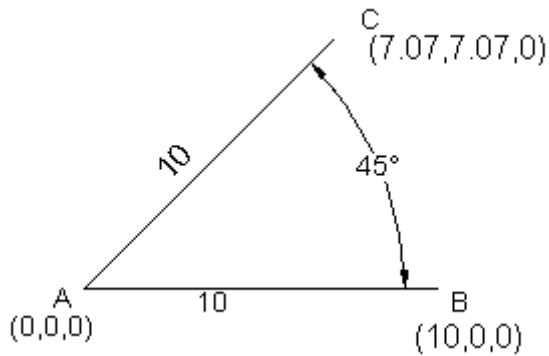
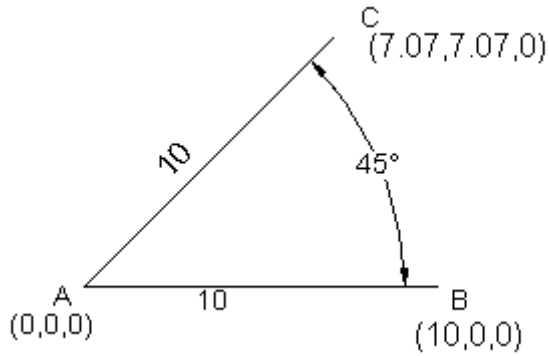
For relative coordinate input, there are five different entry formats available:

### **Directed** – Distance<<

Directed relative coordinates uses an input distance and the current location of the cursor relative to the last entered point to establish direction or angle. The two << symbols that follow distance are the programs indication that cursor location is to be used to establish the angle. Directed relative coordinate input varies from the other forms in that it does not require the @ prefix.

Note: With Ortho mode set to On, this input method provides a very quick and powerful entry method for drawing lines parallel to the axes. IE when used in drawing floor plans

### **Polar** - @ Distance < Angle



In the above illustration, we wish to establish Point "C" relative to Point "A". "A-B" lies along the 0o angle or bearing and "A-C" lies on the 45o angle or bearing, a distance of 10 units apart. To draw a line from A to B in relative polar coordinates, we would give the LINE command, then enter (or pick) point A and then specify point C as @10<45. The line will then extend to C, after which we can simply continue on to points D, E,F..., by entering distance and angle to each in the same fashion. You can alternate between absolute and relative coordinate entry as required.

#### **Cartesian** - @deltaX,deltaY,deltaZ

In this format, we simply indicate the relative (plus or minus) X,Y and Z distances from the reference point to establish the new point. For example, if we wish to set point A from a relative point B in the above illustration, we would pick point B and then specify A as @-10,0,0, as A is -10 x units from B and at the same Y and Z values.

#### **Spherical (3D)** - @distance<angle1<angle2

In this format, you specify the location of the second point relative to the first as being the distance **in** the X-Y plane, the angle (**in** the X-Y plane) and the angle **from** the x-y plane. In the example @10<90,5 distance in the X-Y plane is 10 units, the angle (measured from the positive X axis in the X-Y plane is 90o and angle measured from the X-Y axis to the second point is 5)

#### **Cylindrical (3D)** - @distance,<angle,Z distance

In this format, you specify the location of the second point relative to the first as being the distance (in the X-Y plane), the angle (in the X-Y plane) and at the relative Z distance value from the first point (IE @10<90,5 where distance is 10 units, angle is 90o and Z distance for point 2 is 5)

## Using The Measure Command

The MEASURE command has exactly the same effect as issuing the POINT command with the **Measure** option. Construction points are inserted along the length of a selected linear feature at the interval distance that you specify. Any remaining length that is less than the interval will be located at the far end of the object from the end that you pick when selecting the object. The line, polyline or arc used in this exercise remains unchanged by the command.

The linear object can then be deleted, as necessary, leaving the points arranged along the former objects alignment.

**See also:** the command DIVIDE.

## Using The Divide Command

The DIVIDE command has exactly the same effect as issuing the POINT command with the **Segments** option. Construction points are inserted along the length of a selected linear object to effectively divide the object into a specified number of equal length segments. The line, polyline or arc used in this exercise remains unchanged by the command.

The linear object can then be deleted, as necessary, leaving the points arranged along the former objects alignment.

**See also** the command MEASURE.

## Drawing in 3D

3-D drawing topics are included in the section below entitled "**3-D**" located in "**Advanced Concepts**"

## Modifying

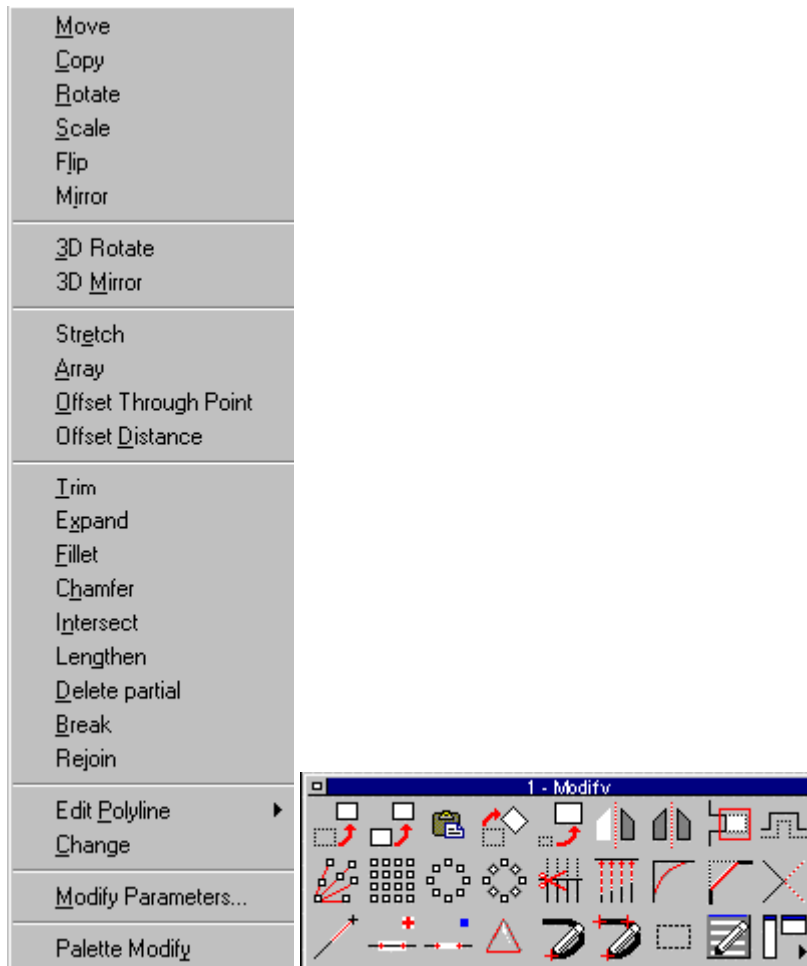
### Modifying

No matter how careful you might be when drawing, you will certainly need to modify drawing objects from time to time. This section will briefly introduce you to the basic commands that you will be using to do this. These commands are all available on the Modify pull-down menu pictured below, but can also be typed in at the command prompt, or selected from the Modify Palette that can be activated from the Palette Modify selection at the bottom of the pull-down menu or from



the Modify Palette icon located on the horizontal toolbar.

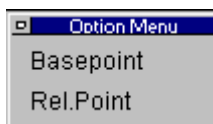




## Moving Objects



Moving objects is a fairly simple concept. After issuing the MOVE command or selecting "Move" from the Modify pull-down menu, you will be prompted to select objects to be moved. After picking the individual objects to be moved or building a selection set, you will be presented with the following dialog:

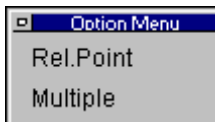


With these two options, you specify the *From* reference point for the move. Choose the Rel.Point option if the move is relative to the origin of the coordinate system, or Basepoint if it is relative to another point in the drawing. For accuracy when picking points, be sure to input (X,Y,Z) coordinate values or use appropriate Osnaps. The command terminates after the user picks a second *To* point on the drawing. All selected objects will be moved in the direction and by the same distance as the bearing and distance between the two picked reference points.

## Copying Objects



Copying objects is very similar in procedure to moving objects described previously. After issuing the COPY command or selecting "Copy" from the Modify pull-down menu, you will be prompted to select objects to be copied. After picking the individual objects to be copied or building a selection set, you will see the following dialog:



Select **Multiple** if you want to make multiple copies of the selected objects and then pick the *first* and *second* reference point on the drawing that define the direction and magnitude of the first copy. Note that the relationship between the original object and the first reference point is maintained as the relationship between the object copies and subsequent *second* reference points. Also, all selected objects will be copied in the direction and by the same distance as the bearing and distance between first and second reference points.

Choose the **Rel.Point** option whenever copying is relative to the origin (0,0,0) of the coordinate system. This sets the *first* reference point at the origin.

To copy objects only once, you can ignore the above dialog and simply pick first and second reference points.

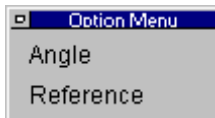
**Note:** To make duplicates of objects without moving them from the original object locations, you may respond to the two point pick prompts by pressing the @ key each time. This is usually more reliable and convenient than trying to make a second point pick in the same location as the first.

## Rotating Objects



Individual or multiple objects can be rotated around a fixed point using the ROTATE command or selecting Rotate on the Modify pull-down menu.

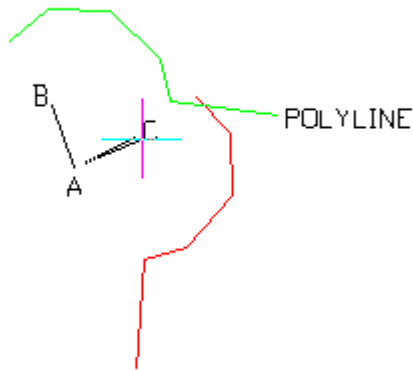
After issuing the command, you will be asked to select objects to be rotated. You can pick individual objects or else use the options available on the option dialog to collect a selection set of objects for the command to operate on. Once the objects have been selected, you will be asked to pick a point of rotation, and then be offered the following dialog.



At this point, you can either rotate the objects manually by moving the mouse and fixing the rotation by picking when the ghosted objects appear to have been rotated to the correct orientation, or, you can pick one of the options on the dialog. The Angle option (default) permits you to specify the rotation in the angular measurement format currently configured for use in your

drawing. Positive angle measurements will rotate objects counterclockwise and negative angles in a clockwise direction.

Often however, you will not know what angle to specify, exactly. You may know that you want things rotated from "this" orientation to "that" orientation in you drawing, and this is where the Reference option can be of assistance. Having established the rotation point (Point "A" below), we wish to rotate the green polyline from an orientation A-B to an orientation A-C, so we will select the Relative option. We are again prompted for rotation angle (in case you have found it written down somewhere in the meantime), however we will specify the angle by picking the vertex "A" of our angle. We will then pick point "B" to establish the "From" bearing, then pick point "C" to establish the "To" bearing. This procedure defines the angle of rotation calculated by the program. The diagram below shows the screen image just before point "C" is selected to establish the angle, and it shows how the program presents a ghost image of the objects (shown in red) indicating the alignment if the current cursor position is selected.

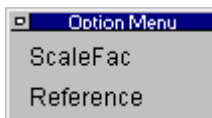


## Enlarging or Shrinking Objects



An object or group of objects can be enlarged or shrunk around a reference point using the SCALE command, or selecting Scale from the Modify pull-down menu.

Once the command has been issued, you will be asked to select objects to scale and a base reference point, followed by the following dialog:



The **ScaleFac** option will prompt you to enter a scale factor to apply to the selected objects. An entry of 2 will double the size at twice the distance from the first reference point, etc.

The **Reference** option allows you to change of the size of an entity without needing to first calculate the scale factor. For example, if you need to enlarge an object with a length of 3.45 drawing units up to 6.9 drawing units, you would first calculate the scaling factor ( $6.9 / 3.45 = 2$ ).

The Reference option makes this calculation for you. Select the Reference option from the option bar. Enter the reference length, and then enter the new length: IE.

```
>SCALE  
Select objects: P1  
Select objects: [Enter]  
Basepoint: P2  
Scale factor: Reference  
Reference length <1>: 3.45  
New length: 6.9
```

See Also: STRETCH Command

## Flipping Objects 180 Degrees



The FLIP command (Modify pull-down - Flip) flips selected entities 180 degrees about a specified axis. (This is unlike the Mirror command, where the original entities remain in place.) The flip axis can be at an arbitrary angle, or be parallel to the X- or Y-axis.

```
> FLIP  
Select objects: P1  
Select objects: [Enter]  
First point of mirror axis: P2  
Second point: P3
```

## Copying Objects at a 180 Degree Orientation



The Mirror command (Modify pull-down – Mirror selection) makes a mirrored copy of selected entities, and keeps the original entities in place. The user will be asked to pick two points on the axis about which the mirroring takes place. This is different from the Flip command, which erases the original entities.

```
> MIRROR  
Select objects: P1  
Select objects: [Enter]  
First point of mirror axis: P2  
Second point: P3
```

### Notes:

1. If there is text among the selected objects, it is likely that you do not want that text mirrored. For this reason, AllenCad does not by default, mirror text entities.
2. If you need the text mirrored, use the EditPar command (Modify > Modify Parameters) to display the Edit Parameters dialog box. Click the check box next to Mirror Text and click OK.

## Stretching Objects



The Stretch command (Modify pull-down - Stretch) stretches entities that lie on the border of the selection rectangle, but moves entities that lie within the selection rectangle. (This command performs like a combination of the Move and Lengthen commands). **Objects may be selected using Crossing and CPolygon options, only.** Stretching is useful for enlarging or reducing entities in one direction. This is in contrast to the Scale command, which scales in both the X and Y directions).

After issuing the command, and selecting objects using Crossing or CPolygon (crossing polygon) you will be asked to select the basepoint for the stretch. This can be done by typing the coordinates, or picking the basepoint. You will then be similarly asked to pick the second point of displacement. The angle and distance between the two points define the direction and the magnitude of the stretch or displacement.

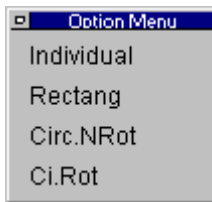
Command input Sample:

```
>STRETCH  
Select objects: P1  
Select objects: [Enter]  
Basepoint: P2  
Second point of displacement: P3
```

## Copying Objects in a Pattern

The ARRAY command (Modify pull-down - Array) creates multiple copies of selected entities at random or in a pattern. The pattern may be in horizontal rows and vertical columns, in a circle, or in an arc. In addition, the entities can be rotated when arrayed in the circular or arc patterns.

After entering the command and selecting the entities, select an option:



### Individual



The Individual option copies the selected entities repeatedly, just as if you were to use the Copy command repeatedly. This option is the default for the Array command.

```
IE >ARRAY  
Individual> [Enter]  
Select objects: P1  
Select objects: [Enter]  
Basepoint: P2  
Target point: P3  
Target point: P4  
Target point: [Enter]
```

## Rectangular



The Rectangular option arranges the array in rows and/or columns, called a rectangular array.

```
> ARRAY
Individual> Rectang
Select objects: [pick]
Select objects: [Enter]
RECTANG
No. of columns: 2
No. of rows: 3
Distance between columns: 4
Distance between rows: 5
```

**Note:** The rows and columns are normally array up and to the left. To array the entities down and to the right, use a negative number for the distance.



The **Circ.NRot** and **Ci.Rot** options copy the selected entities in a circular path, called a circular array. The copying can be around a full 360-degree circle, or a partial circle (an arc.).

**Note:** Circular arrays are constructed counterclockwise from the starting point. For a clockwise rotation, enter negative values for the angle.

The **Circ.NRot** option creates a circular arc without rotating the selected entity.

The **Ci.Rot** option rotates the selected entities so that they always "face the center."

Both options prompt you for the following parameters:

```
>ARRAY
Individual> Circ.NRot or Ci.Rot
Select objects: P1
Select objects: [Enter]
Center: P2
Number of entities: 4
Angle to fulfill <360>: [Enter]
```

- **Center:** the center point of the circular array.
- **Number of entities:** the number of times the Array is to reproduce the selected entities.
- **Angle to fulfill:** the distance the array should go around the circle; 360, if the array is to be a full circle.

**Note:** When creating a circular array, the command uses a reference point on which to position the entities on the circumference of the circle, as shown in the following

Entity	Reference point
Arc	Center of arc.
Circle	Center of circle
Line	Start or end point.
Part	Insertion point.
Point	Insertion point.
Rectang	Start point.
Text	Insertion point.

table:

If you wish to change this reference point, the **Basepoint** option lets you define an alternate reference point on the entity to be array. Remember to use an object snap mode with this option.



The Ci.Rot option (short for "circular, rotated") rotates the arrayed entities when placing them along the circular path. The entities are rotated in the same direction as the circle describing the path. The rotation angle corresponds to the angle between the copies (measured between the reference points of the arrayed entities).

## Copying Objects in Parallel Orientation



The Offset command (Modify pull-down - Offset Through Point) creates parallel copies of lines, circles, arcs, and 2D polylines. The command copies the selected entity and places it a specific distance from the original. The original entity stays in place.

### Notes:

1. The Offset command can create unexpected results, depending on the entity selection:
2. Line and straight 2D polyline: an identical copy is created, to make parallel lines.
3. Circle and arcs: a copy is created, but is scaled by a factor to make concentric curves. The center of the original circle or arc is used as a basepoint for scaling.
4. 2D polylines made of line and arc segments: due to the arc portions being scaled, the offset polyline may look quite different from the original.

There are two ways to specify the offset distance:

1. One method is to pick a point, using object snaps if necessary, rather than selecting the Distance option available in the dialog box. AllenCad offsets the copy at the pick location.
2. The second method is to select the **Distance** option (the **DOffset** command). Enter the offset distance. You are then prompt "Offset to Side?". Pick any point on either side of the original entity to indicate offset direction.

### Both Sides

You can offset an entity to both sides by selecting the **Both Sides** option when you are asked to specify the side to offset. The both sides option is only available when the distance option is selected:

>OFFSET

Selection of an entity  
(Line, circle, arc, 2D-Polyline): P1  
Offset by point: Distance  
Distance <5> 10  
Offset to side? P2  
Selection of an entity  
(Line, circle, arc, 2D-Polyline): [Enter]

In both cases, the Offset command continues, until you press Enter or Esc, and an established distance is offered as a default value for subsequent offset commands.

See also "Using Offset to Advantage"

## Trimming Objects



The Trim command (Modify pull-down - Trim) cuts off entities that cross a selected "cutting edge (or edges)." The cutting edge is the intersection with the entities to be trimmed. The entities are trimmed on the side that you pick. The command keeps repeating until you end it with the Esc key.

>TRIM  
Select entities for intersecting edges  
(Line, circle, arc, 2D-Polyline): P1, P2  
Select entities for intersecting edges  
(Line, circle, arc, 2D-Polyline): [Enter]  
Select entity for trimming  
(Line, circle, arc, 2D-Polyline): P3, P4  
Select entity for trimming  
(Line, circle, arc, 2D-Polyline): [Esc]

Start the Trim command and select one or more entities that will become the intersection edge.

The next prompt asks you to select the entity that you want to trim. The selection is sensitive to location: the point you pick identifies the portion that will be trimmed away. As entities are selected, they are trimmed and the trimmed portion disappears from the drawing.

## Extending To An Intersection



The Expand command (Modify pull-down - Expand) lengthens entities to a boundary edge. The boundary edge is another entity, such as a line, arc, circle, or 3D polyline. Note: AutoCAD users know this command as EXTEND.

**Note:** the Expand command does not work with "closed" objects, such as circles. Entities to be expanded are lengthened. They do not change direction. A line remains a line; an arc remains an



arc.

Start the Expand command then select the entities that will act as the boundary edge. You may select one or more lines, circles, arcs, and 2D polylines. Press Enter to end the boundary edge selection.

IE. > EXPAND

Select entities with boundary edges (line, circle, arc, 2D-polyline): P1, P2

Select entities with boundary edges (line, circle, arc, 2D-polyline): [Enter]

Select entity for expanding (line, circle, arc, 2D-polyline): P3

Select entity for expanding (line, circle, arc, 2D-polyline): [Enter]

Then select the entities to be expanded. This is sensitive to your pick location. The end of the object closest to your pick point will be the end to be extended. When more than one boundary edge has been selected, the entity is expanded to the first boundary edge it encounters. Picking a second time results in the expansion continuing to the second boundary edge, etc.

**Note:** Some objects cannot be used to expand to other entities, most notably Parts or Blocks.

## Trimming at Intersection



The Fillet command (Modify pull-down - Fillet) rounds the intersection of two lines. These can also be lines of an object such as a rectangle. The Fillet command treats 2D polylines differently from lines: every vertex of the polyline is filleted. The command cannot fillet curved objects, such as arcs. See Also INTERSECT and CHAMFER.

When the fillet radius is 0.9 (the default), this command precisely cleans up the intersection of two lines.

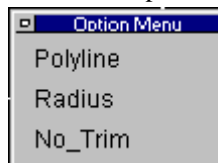
The Fillet command prompts you to select the two lines to be filleted.

> FILLET

Select first line: P1

Select second line: P2

Then these options are shown in the options bar:



1. **Polyline Option** - The Polyline option fillets a single polyline. Every vertex of the polyline is given the fillet radius.

> FILLET

Select first entity: Polyline

Select 2D-polyline: P1

2. **Radius Option** - The Radius option allows you to define the radius for the fillet. By

default, the radius is 0.0. When you change the radius, AllenCad remembers it for the next command operation.

> FILLET

Select first line: Radius

Filleting radius <0.0000>: .75

Select first line: P1

Second line: P2

3. **No-Trim** - The Trim option functions as a switch: it decides if the lines beyond the fillet are to be erased. When the options bar displays Trim, you switch off the trimming by clicking the word; it then changes to No-Trim.

## Chamfering At Intersection



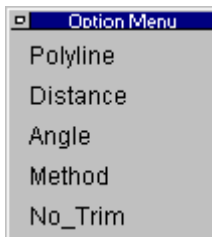
The Chamfer command (Modify pull-down - Chamfer) creates a beveled corner between two intersecting lines.

> CHAMFER

Select first line: P1

Second line: P2

Select the two entities to be chamfered by picking them with the cursor. You will then be offered the following options:



1. **Polyline Option** - The Polyline option chamfers a single polyline. Every vertex of the polyline is given the input chamfer dimensions.
2. **Distance Option** - The Distance option allows you to specify the two chamfering distances (back from the intersection):  
>CHAMFER  
Select first line: Distance  
First chamfering distance <1.0000>: [Enter]  
Second chamfering distance <0.5000>: 1.000  
Select first line: P1  
Second line: P2
3. **No-Trim Option** - The No-Trim option functions in exactly the same manner as described with the Fillet command. It controls whether the entity lengths beyond the intersection are trimmed off, or not.

4. **Angle Option** - This option allows you to specify the chamfer distance on the first entity, as well as the angle of the chamfer.

## Intersecting Linear Objects



The Intersect command (Modify pull-down - Intersect) is a special case of the Fillet command, with the fillet radius always set to 0. This makes the command useful for trimming intersecting lines to be trimmed.

```
> INTERSECT
Select first line: P1
Second line: P2
```

Select the two lines to be trimmed. Be sure to make the pick on the segment of each line that is to remain.

## Lengthening Linear Objects



The Lengthen command (Modify pull-down - Lengthen) changes the length of open entities, such as lines and arcs. Unlike the Expand command, Lengthen works without a boundary edge; it extends an entity by a specific value or ratio (percentage value) of the original size.

Although the command is called "Lengthen," it may be used to lengthen or shorten entities. Entities are changed in length, not scaled. Entities are always lengthened at the end closest to the cursor pick point.

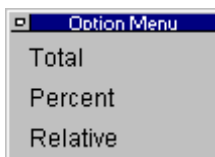
```
> lengthen
Select entity (line, arc): [pick]
Current Length <7.11>: Dynamic:
```

After selecting the line or arc to lengthen, the command reports the entity's current length. The **Dynamic** option is the default. You drag the line or arc to its new length. When you click with the cursor, the new length is fixed.

```
> LENGTHEN
Select entity (line, arc): P1
Current length <100.00>: Dynamic P2
Select entity (Line, Arc): [Enter]
```

The Lengthen command continues until you press Enter or Esc.

The option bar displays other options:



1. **Total Option** - The Total option changes the length to an absolute distance. The change is applied to the length of a line or to the circumference of an arc. To assist you, the current length of the selected entity is displayed in parentheses.  
 > LENGTHEN  
 Select entity (line, arc): P1  
 Current length <100.00>: Total  
 Total: 200.00  
 Select entity (Line, Arc): [Enter]
  
2. **Percent Option** - The Percent option allows you to specify the new length by a percentage value. This percentage value relates to either line length or the circumference of an arc. For example, specifying 200 doubles the length of the line or the arc, while the value of 50 shortens the entity to half its original length.  
 > LENGTHEN  
 Select entity (line, arc): P1  
 Current length <100.00>: Percent  
 Percent: 60  
 Select entity (Line, Arc): [Enter]
  
3. **Relative Option** - The Relative option lengthens a line or arc by a value defined by the distance between two points. This option is useful when the entity must be lengthened by reference to another object. (If the Angle option has been selected, you determine the angle by which the arc is lengthened by indicating two points in the XY-plane.)  
 > LENGTHEN  
 Select entity (Line, Arc): P1  
 Current length <10.00>: Relative  
 Relative: P2  
 Second point: P3  
 Select entity (Line, Arc): [Enter]
  
4. **Angle Option** - To lengthen arcs, use the option Angle. This option functions like a switch. Selecting it from the option bar causes all of the following options to display a prompt for the included angle of the arc.  
 > LENGTHEN  
 Select entity (line, arc): [pick]  
 Current Length <9.32>: Dynamic: Angle  
 Included angle of arc <205.90>: Dynamic: [pick]  
 Select entity (line, arc): [Enter]

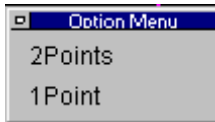
### Deleting a Portion of an Object



The DelPartial command (Modify pull-down - Delete partial) erases a portion of a line, 2D polyline, circle, or arc. This command is useful for dividing an entity: the result is two independent entities, which can be edited or operated on separately.

Whereas the Trim command cuts off the end of open entities, the DelPartial command cuts out their middles.

**Options:**



**2Points Option** - The 2Points option is the default. It requires that you pick two points:

```
> DELPARTIAL
Select object (line, circle, arc, 2D-polyline): P1
First point: P2
Second point: P3
```

The section is erased between points P2 and P3 and two separate objects are created.

When the entity is a circle, it becomes an arc. When a 2D polyline has a width greater than zero, it is cut at right angles to the polyline.

**1Point** - To separate or break an entity, choose the **1Point** option.

```
> DELPARTIAL
Select entity (line, circle, arc, 2D-polyline): P1
First point: 1P
Break point: int
of object: P2
```

This option is useful for breaking an entity into two pieces, for example, at its intersection with another object. To do this:

1. Select the **1Point** option (or type 1P for short) at the "First point:" prompt.
2. Select the break point. It may be wise to use the object snap options to increase accuracy.

You achieve the same result with the 2-point option when you type @ at the "Second point:" prompt. The @ symbol signifies the last point entered.

```
> DELPARTIAL
Select entity (line, circle, arc, 2D-polyline): [pick]
First point: int
of object: P1
Second point: @
```

In both cases, the entity will look the same as before however, the object is now divided into two separate entities.

**Notes:**

1. You cannot break a circle with the 1Point option, since that would create an arc of 360 degrees.
2. You can also use the DelPartial command to shorten an entity. Here's how: Specify the first point on the entity where you want the new endpoint to be, then pick the other point past its endpoint. The entity is cut off at the first point you picked on the object.

**See Also:** "Deleting a Portion of an Entity – Break Command"

## Deleting a Portion of an Entity - BREAK Command

The BREAK command (Modify pull-down - Break) breaks objects at a point or at points you select, just as the DelPartial command does.

```
> BREAK
Select entity (line, circle, arc, 2D-polyline): [pick]
Second point (or F for first point): [pick]
```

The BREAK command has been added for convenience of those familiar with other CAD programs. It is similar but not identical to the DelPartial command. The primary difference is that the Break command assumes that you might want to use the object pick point when selecting the entity as the first point of the break.

You can however, respond to the "*Second point:*" prompt with the First option (or type F for short). This explicitly specifies that the following point will be the first pick point of the entity selection, followed by a second point:

```
> BREAK
Select entity (line, circle, arc, 2D-polyline): [pick]
Second point (or F for first point): F
First point: [pick]
Second point: [pick]
```

You have the choice to break the entity into two pieces without erasing any part of it. To do this: specify the same point for both the first and the second points. Type @ at the request for the second point, which indicates the previous point entered. This is useful when you want to break an entity at the intersection with another entity:

```
> BREAK
Select entity (line, circle, arc, 2D-polyline): [pick]
Second point (or F for first point): F
First point: int
of object: [pick]
Second point: @
```

## Combining Co-linear Objects



The Rejoin command (Modify pull-down - Rejoin) unites separate line segments, which are aligned in the same direction. It also joins arc segments that share the same center and the radius. This function is useful for reuniting line and arc segments that have been divided by the DelPartial command or BREAK commands.

Select the command, then pick the entities to be rejoined:

```
> REJOIN
Select first entity (line, arc): P1
Select second entity (line, arc): P2
```

## Modifying Object Parameters - CHANGE Command

The Change command (Modify pull-down - Change) edits the geometric character and the properties of many entities, including lines, circles, arcs, text, attributes, and part. The properties or geometry that can be changed depend on the entity.

**Note:** To change the layer, color, or linetype properties, you may find it more useful to use the PropEdit Command (Edit > Modify Properties).

> CHANGE

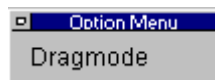
Select entity (line, circle, arc, text, attribute definition, block insertion): P1

Select entity (line, circle, arc, text, attribute definition, block insertion): [Enter]

Modification point: P2

Select the entity to be changed.

You may preview the change by selecting the **Dragmode** option from the options bar.



Since the properties changed depend on the entity selected, the following sections describe the changed by entity.

**Lines** - The Change command changes one endpoint of the line. You can move the endpoint to a new position, while the other end of the line remains the same. If the endpoint of a line is moved without using Dragmode, the point to be moved will be the point closest to the selection point. If Dragmode is activated, the point that lies closest to the cursor position will be moved. By changing the cursor position, the endpoint of the line to be moved can be changed. If needed, the MOVE command can be used to move the entire line to a new location in the drawing.

**Circles** - The Change command alters the diameter (or radius) of a circle. Select the circle, then determine the new diameter by picking a point or typing an X, Y, Z coordinate. If needed, the MOVE command can be used to change the radius point of the circle

> CHANGE

Select entity (line, circle, arc, text, attribute definition, block insertion): [pick a circle]

Select entity (line, circle, arc, text, attribute definition, block insertion): [Enter]

Modification point: P2

**Arcs** - The Change command alters the diameter (or radius) of an arc. Select the arc, then determine the new diameter by picking a point or typing an X,Y coordinate. If needed, the MOVE command can be used to move the arc to a new location.

> CHANGE

Select entity (line, circle, arc, text, attribute definition, block insertion): : [pick an arc]

Select entity (line, circle, arc, text, attribute definition, block insertion): [Enter]

Modification point: P2

**Text** - The **Change** command changes the following properties of a text entity. These text attributes, or parameters, may be changed. Existing values are displayed as parameters and can be taken over by pressing Enter.

- **Modification point:** pick (or enter the coordinates) a new insertion point for the text.
- **Text style:** enter the name of the new text style.
- **Text height:** define the new height by typing a numeric value or by picking a new height. Dragmode is automatically activated during this option.
- **Rotation angle:** change the rotation angle by picking a point or type a number. Dragmode is automatically activated during the change of this attribute.
- **Text:** type different text, which replaced the entire selected text.

>CHANGE

Select entity (line, circle, arc, text, attribute definition, block insertion): [pick text]

Select entity (line, circle, arc, text, attribute definition, block insertion): [Enter]

Modification point: P2

Text style <STANDARD> Normal

Height <2>: 1

Rotation angle <0>: 45

Text <first floor>: 1st floor

**Attribute Definition** - The Change command changes the following properties of attributes (as for text):

- Insertion point.
- Text style.
- Text height.
- Rotation angle.

As well as the these specific properties of attributes:

- **Name:** determines the name of the attribute; enter a new name.
- **Request:** enter a new line of text for the attribute prompt.
- **Parameter value:** the constant or variable value of the attribute; enter a new value (a sequence of characters or numbers).

You will find more information about attributes in "**Parts and Attributes, XRefs, & Groups.**" in your **User Manual**.

> CHANGE

Select entity (line, circle, arc, text, attribute definition, block insertion): P1

Select entity (line, circle, arc, text, attribute definition, block insertion): [Enter]

Modification point: P2

Text style <STANDARD> Normal

Height <2>: 1

Rotation angle <0>: 45

Text <first floor>: 1st floor

Name <Att003>: [Enter]

Request <Index No.>: current number

Parameter value <1.01>: 2.01

**Part** - The Change command allows you to change the insertion point and the rotation angle of a part (block). Dragmode is automatically activated during the change of this attribute.

> CHANGE

Select entity (Line, circle, arc, text, attribute definition, part): P1

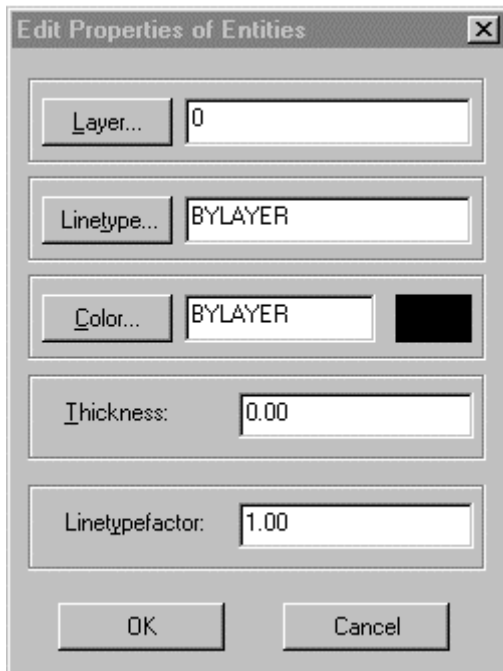
Select entity (Line, circle, arc, text, attribute definition, part): [Enter]



Modification point: P2  
Rotation angle <0>: 90

## Changing an Object's Layer, Color, Linetype, etc.

You can easily change the layer for a selected group of objects using the PROPLAYER command. Alternatively, you can change the layer, color, linetype, thickness or linetype scale factor of selected objects, using the PROPEDIT command. When you issue this command, you will be prompted to select the objects to be edited and then presented with the dialog below. You can change as many of the object properties as you wish, while issuing this single command.



## Text

### Text Entry Options

With AllenCad, you can enter single line text using three commands, TEXT, DTEXT and RTEXT. For clarification, single line text refers to the fact that each line of text produced is a separate entity. Both TEXT and DTEXT can produce multiple lines of text that can be addressed individually.

With TEXT text, the characters are entered into a dialog and when finished, the whole text entry is placed and positioned on the drawing as a block. This can be useful whenever you are unsure of where you wish to fit text onto the screen, before it's dimensions are known.

With DTEXT ("typewriter text") you specify the text characteristics and then proceed to type in the text, meanwhile the text appears on the drawing, character by character as you type. You may prefer to use this option when the left boundary of the text is set, but the right boundary may need to be "fitted".

RTEXT ("Reference Text") simply allows you to select an existing text string that has the text parameters

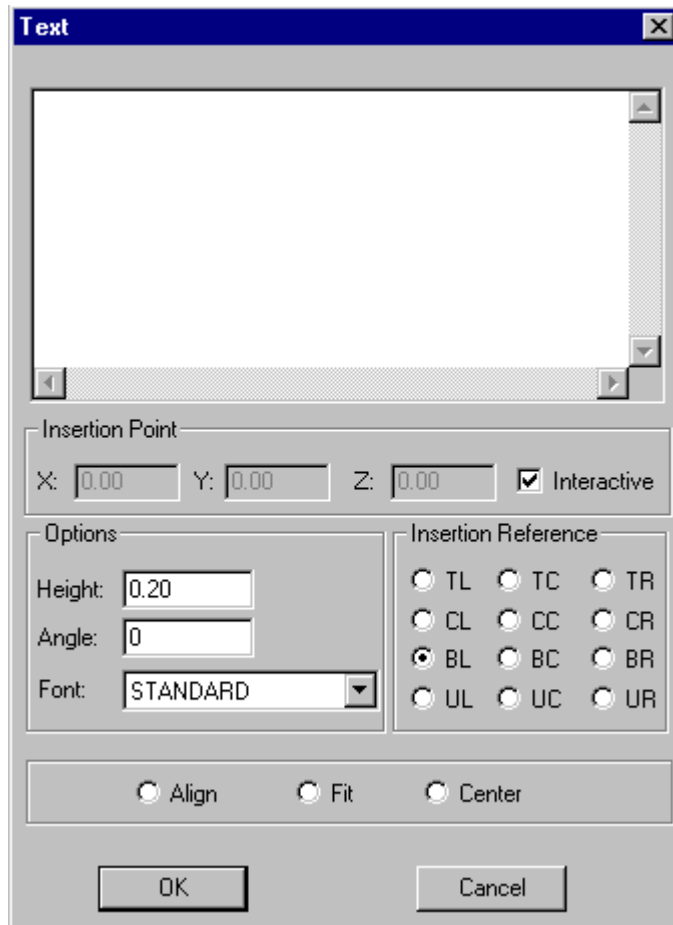
you want (layer color, height, etc.) - to edit the text – and then place the new text on the drawing.

MTEXT (multi-line text) on the other hand, allows you to enter multiple lines of text that are not treated as separate entities. When you begin the entry, you are asked to provide a frame (width) that the text is to fit within, and the text automatically wraps to the next line whenever the right boundary is reached during text entry.

Both TEXT, DTEXT and RTEXT entries can be modified using the TCORRECT command, however MTEXT entities must be modified using the MTEXTEDIT command.

## Entering Text

To enter text, select the Text option on the Detail pull-down menu. The following dialog will then appear on your editor.



Text is written into the upper window and parameters can be modified below, as needed. By default, this text will be placed in the current layer and in the currently set color. Also note that "interactive" has been selected for insertion point. With this option, a box representing your text attaches to the cursor for placement in the drawing. To specify coordinates, simple deselect interactive to enable coordinate input.

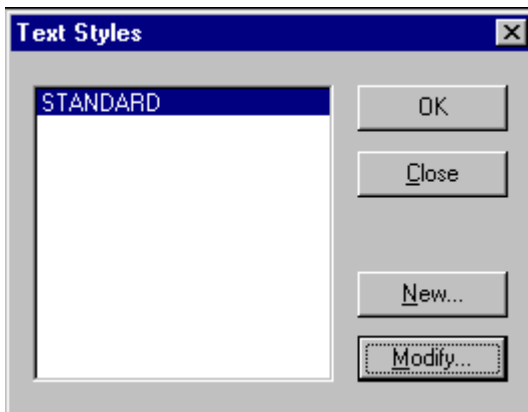
**See Also:** Command DTEXT – dynamic (typewriter) text

Command MTEXT - multi-line text

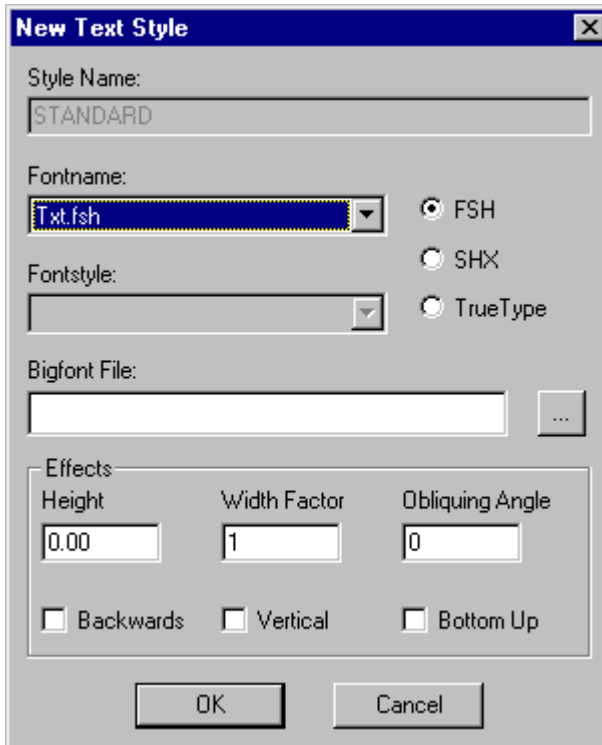
Command RTEXT – Reference text

## Creating and Editing Text Styles

When you begin a new drawing without a template, you are provided with a single text style that uses the "Text" font. This font has default values for text height, obliquing angle, etc. Before inserting text, you will probably want to set up the various text styles that will meet your requirement. To do this, you will select **F**onts from the **D**etail pull-down menu. You will then be presented with the following dialog:



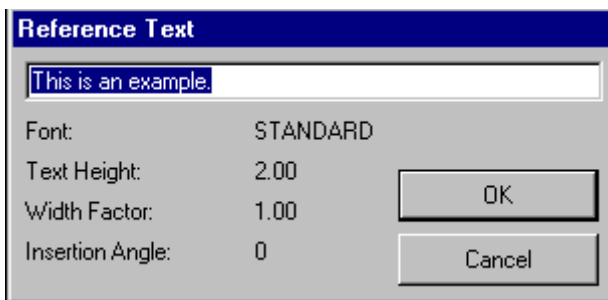
In it, you will see the single text style that is currently available. You can choose **N**ew to create a new style or **M**odify to change the set style parameters. When you select New or Modify, you will be presented with the dialog below:



This indicates that you have elected to modify the "Standard" text style that was the original default style. By changing the entries and parameters here, you will modify default settings for the "Standard" text style. All text that is subsequently inserted using the Standard style will be formatted accordingly.

### Text by Reference

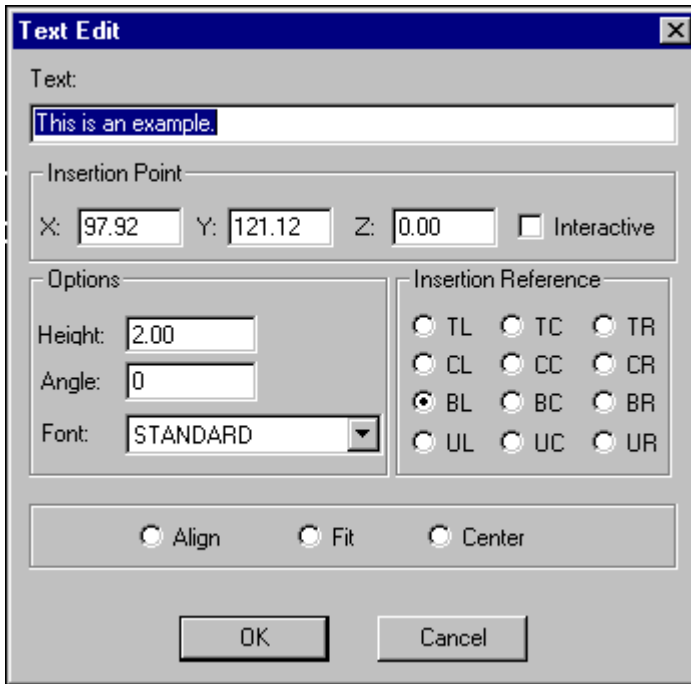
If you want to enter new text and to have it use the same parameters (style, height, layer, etc. ) as some existing text simply select Reference Text... from the **Detail** pull-down menu. You will be asked to pick the text that you want to use as reference and afterward be presented with the following dialog box.



You will then simply change the text entry to what you want and press **OK**. At this point, you will be asked to place the new text into position on the drawing and will be given a box representation of that text attached to the cursor to guide you in placement.

## Modifying Text

To modify the style of an existing text string in the drawing, you will go to the **Detail** pull-down menu and select **Modify Text**. You will then be asked to select the text to be modified and presented with the following Dialog box.



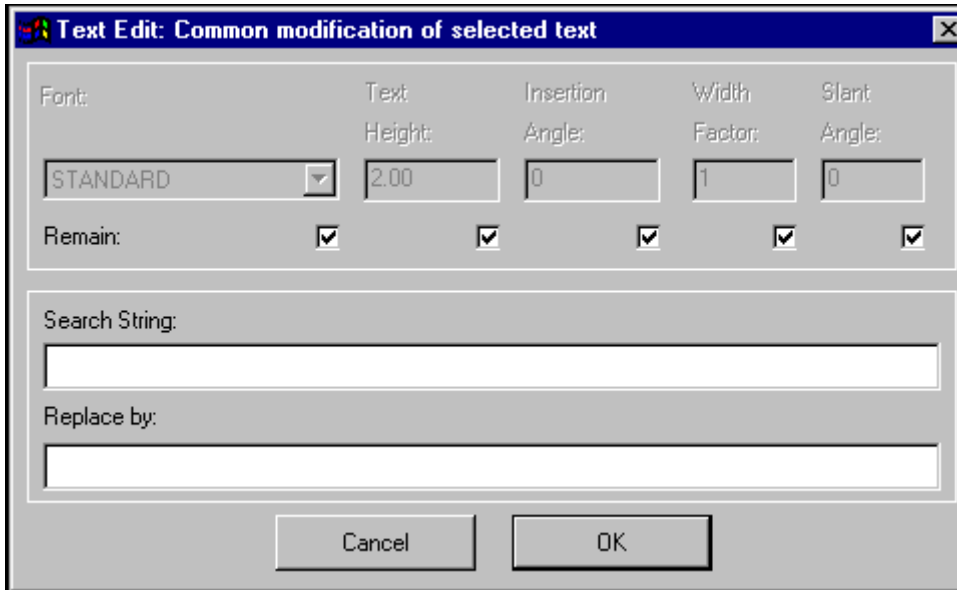
The **Text Edit** dialog box is used to modify text properties. It features a title bar with a close button. The main area includes a **Text** input field containing "This is an example.". Below this is the **Insertion Point** section with fields for X (97.92), Y (121.12), and Z (0.00), along with an **Interactive** checkbox. The **Options** section contains fields for **Height** (2.00), **Angle** (0), and a **Font** dropdown menu set to **STANDARD**. The **Insertion Reference** section displays a 3x3 grid of radio buttons for corner and center alignment: TL, TC, TR, CL, CC, CR, BL, BC, BR, UL, UC, and UR. At the bottom, there are three radio buttons for **Align**, **Fit**, and **Center**, and **OK** and **Cancel** buttons.

After changing the parameters and clicking **OK**, the changes to the text will be made. These changes affect only the selected text... not all text using this same style.

**See Also:** Command MTEXTEDIT – multi-line text editing

## Correcting Text

A capability that permits a "search and replace" text correction to be made on selected text. This is initiated by choosing **Correct Text...** from the **Detail** pull-down menu. You will be asked to select text to be considered for correction then be presented with the following dialog box:



As you will notice, along with the ability to search and replace text strings within the selected text, you are also given the opportunity to modify text style parameters for the selected text.

### **Font... (Text Style Definition)**



**Command: FONT**

**Alias FT**

#### **Located:**

- 1. Pull-down Menu: Detail – Fonts**
- 2. Palette: Detail**
- 3. Or Enter Command or Alias Directly at the Command Line**

The command FONT loads fonts from the font files and customizes them to the requirements of the drawing.

Fonts define the shapes of the text characters that make up each character set. In AllenCad, you can use:

1. AllenCad's own compiled shape (FSH) fonts,
2. AutoCAD's own compiled shape (SHX) fonts
3. True Type fonts.

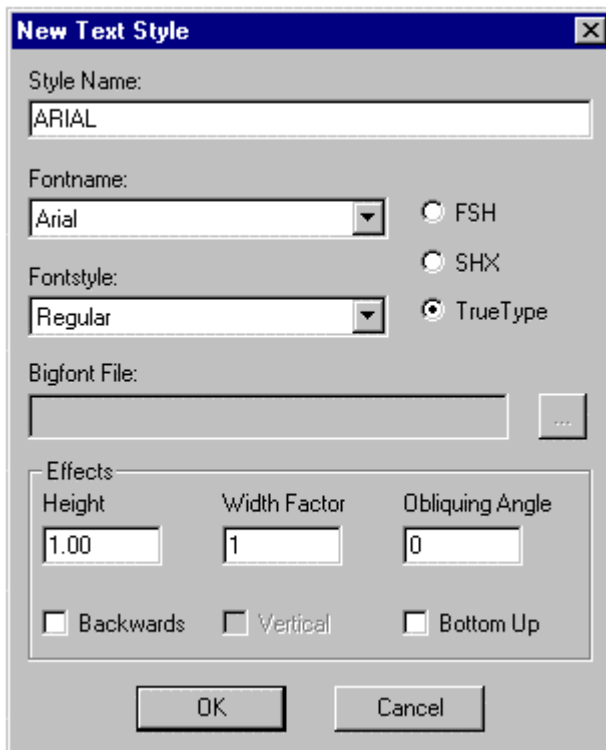
All installed True -Type fonts, provided with your Windows system, are listed by font family name.

SHX fonts and FSH fonts are listed by the name of the file in which the fonts are stored.

All fonts to be used in the program must first be assigned to a text style. When you give the FONT command, you are therefore presented with the following dialog:



Here, you create a new text style or choose an existing one to modify, by assigning a different font. With the required style name highlighted, press OK and you will be presented with the font dialog:



In this case, we have chosen to create a new style called Arial and to apply the Arial true type font to this style. To do this, we click on the True Type radio button and select the down arrow on the Fontname box to display the available choices. We scroll to Arial and select it.

Next, we may need to select a fontstyle. Again we click on the down arrow in the fontstyle window and choose regular text, as we do now wish it to be bold, italic or otherwise.

If we wish to use a Bigfont file, we press the browse button to the right of the window and select our bigfont file from the available choices.

Other text effects are available at the bottom of the dialog. We can assign a text height, width factor or Obliquing (Tilting) Angle. If we want the text to run right to left in a mirror image, we can select the Backwards option. The Bottom Up option will give us text that is upside down. .

**Note:** Some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

Confirm the settings of the properties by selecting OK. Fonts added or changed in this manner can now be used for creating or editing text objects when the modified text style is current.

**Note:** Defined text styles are saved with the drawing file only with a reference to the corresponding font file. They are available any time that the drawing is opened, if the corresponding font file is found in the programs directories.

---

**See also:**

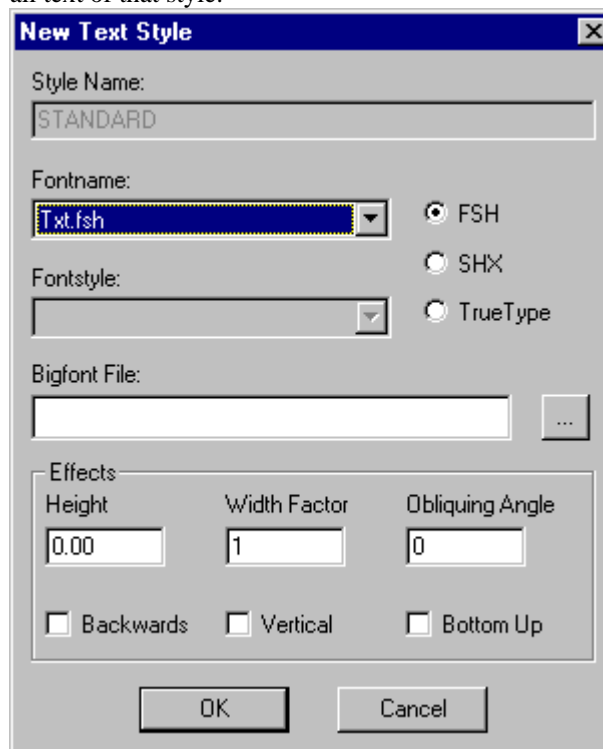
Adding Additional Fonts

Command QFONT

Command TEXT

## Compiling and Assigning New FSX Font Files

AllenCad uses binary font files in both native .FSH format and AutoCAD's compiled .SHX file format. In addition to these, it can also use Windows' TrueType Font files. These binary files are vector format fonts, which are much smaller and faster to process, than regular un-compiled font files would be. Fontfile creation is a two-step process. First, new un-compiled .shp or .fsh files are defined in plain text in .shp or .fsh shape files, and second, these are compiled into .FSH format, using the FCOMPILE command. At this point, these FSH fontfiles are available for assignment when establishing or modifying a text style. This assignment is made, using the FONT command. In addition to assigning fontfiles to text styles, this command also specifies additional text parameters, such as height, format, etc. to be applied to all text of that style.



In the FONT command dialog pictured above, you will see the three radio buttons for font type – FSH, SHX and True-Type. You select which file type that you want to load, and then select the Fontname from

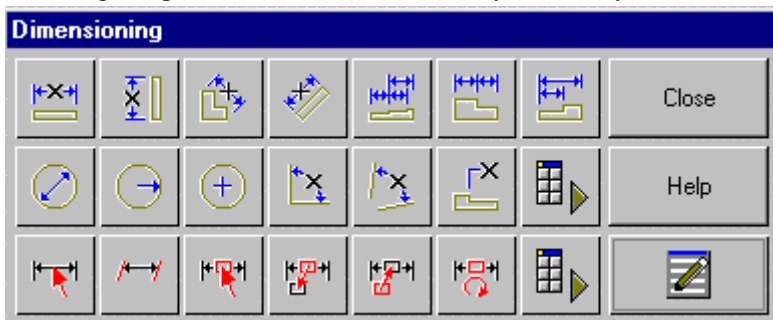


the dropdown list in the window to the left of these buttons. Below that, you can select fontstyle (regular, bold, etc. – as may be available to that particular font), and below that, you can specify the other text characteristics as listed.

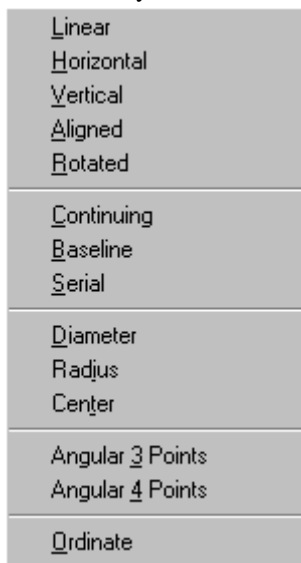
## Dimensioning

### Dimensioning Objects

Dimensions may be placed on the drawing as text entities, however better tools have been provided for this purpose. These can be found on the Dim pull-down menu and dimension palette or entered as commands at the command line. Alternatively, entering DIM at the command line will bring an option box with label and modify icons for you to choose from.



These utilities measure the distances in the drawing and place the measurement label, as well as reference arrows in the drawing for you. As with other toolbar and palette icons, you can hold the mouse pointer over the icon for a second or two to produce a status line message that will identify the icon for you.



Please refer to the Dimensioning Variable Summary for links to information on each variable, or refer to the topics below.

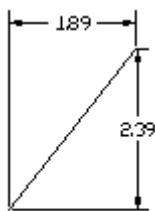
### Setting Dimension Parameters

Before you start dimensioning your drawing, you will want to ensure that your dimensioning settings are appropriate to the scale at which your drawing will be plotted. While dimension text, arrow sizes, etc.

might all be quite clear and readable onscreen, they may be too small to be seen at the plotted scale, for instance.

The DIMTYPE command calls the dialogs that control all dimension settings. If you work with standard scales and dimension parameters, you may wish to ensure that these have been setup correctly in a template drawing. This way, you will not have to configure your dimension parameters each time you start a new drawing.

## Linear Dimensioning



Linear

Linear Dimensioning creates either a horizontal or vertical dimensioning depending on the position of the dimension line. This command is a combination of dimensioning with orthogonal modes horizontal and vertical.

To use this command, define two points of measurement. Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line. The distance between the two points is measured along the X-axis for horizontal and Y-axis for vertical dimensioning and displayed. Pressing the RETURN key will verify the calculated value and insert it into the drawing.

IE.

1. Pick point: P1

2. Pick point: P2

Select dimension line positioning: P3

Verify or modify dimension text <1.89>: Return

## Horizontal Dimensioning



Horizontal

Horizontal Dimensioning is linear dimensioning that both measures and labels the distance between two points parallel to the X -axis.

To use this command, define two points of measurement. Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line. The distance between the two points will be measured along the x-axis and displayed. Pressing the RETURN key will verify the calculated value and insert it into the drawing.

IE.

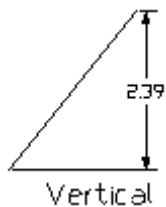
1. Pick point: P1

2. Pick point: P2

Select dimension line positioning: P3

Verify or modify dimension text <1.89>: Return

## Vertical Dimensioning



Vertical Dimensioning is linear dimensioning that measures and labels the distance between two points or else the linear extension of an object parallel to the Y-axis.

To use this command, define two points of measurement. Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line. The distance between the two points will be measured along the Y-axis and displayed. Pressing the RETURN key will verify the calculated value and insert it into the drawing.

IE.

1. Pick point: P1

2. Pick point: P2

Select dimension line positioning: P3

Verify or modify dimension text <2.50>: Return

## Command DIMALI



**Command DIMALI**

**Alias DAL**

### **Located:**

- 1. Pull-down Menu: Dim – Aligned**
- 2. Palettes: Create Dimensions and Dim**

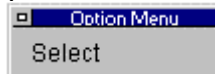
### 3. Or Enter Command or Alias at the Command Line



Aligned dimensioning is a type of linear dimensioning that measures and labels the absolute distance between two points, regardless of the relative position of the axes. This makes it possible to dimension lines, edges and spaces etc. that are not located on the principal axes of the coordinate system (IE. not parallel to x, y or z-axes).



The user will be asked to define two points of measurement (DEFPOINTS). As an alternative the program offers the user the chance to select a specific dimensioning object function. Activate this function by



choosing Select from the options bar. This allows lines, polylines, circles and arcs to be selected. Point to the desired object with the cursor. In the case of lines, polylines and arcs the start and end points of the object will be interpreted automatically and used for the measurement calculations.

The first point of a circle will be defined as a point situated along the perimeter of the circle. The second point will be obtained by drawing a straight line from the first point, through the center point of the circle, to the opposite side of the perimeter of the circle.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line. This can be done either by entering the relevant coordinates, or with the cursor.

The absolute measurement between the two points will be displayed. Pressing the RETURN key will accept the calculated value. The dimension will be inserted into the drawing and shown on a horizontal plain.

> DIMALI

1st Point: P1

2nd Point: P2

Dimension line positioning: P3

Dimension text <2.5000>: RETURN

Or

> DIMALI

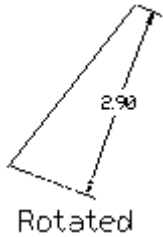
1st Point: SELECT <CR>

select line, polyline, circle, arc: P1

Dimension line positioning: P2

Dimension text <2.5000>: RETURN

### Rotated Dimensioning



Rotated Dimensioning is also a type of linear dimensioning that measures the distance between two points, at an angle specified by the user.

The user will be asked to define the angle at that the dimensioning is to appear in relation to the axial alignment of the coordinate system. This can be done by entering the coordinates at the keyboard, or by using the cursor to click on two points. Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line. The distance between the two points will be displayed at the designated angle orientation. Pressing the RETURN key will accept the calculated value and cause it to be inserted horizontally into the drawing.

IE.

Specify angle of rotation <0>: 70 (in this example)

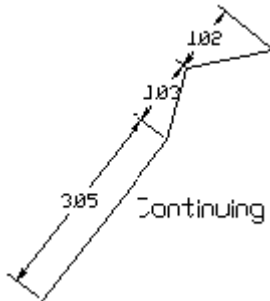
Pick first point: P1

Pick second point: P2

Select dimension line positioning: P3

Verify or modify dimension text <2.90>: RETURN

## Continuing Dimensioning



Continuing Dimensioning is a variation of linear dimensioning in that existing dimensioning is continued or extended. Here a second dimension is linked to an existing dimension to create a dimensioning chain. Continuing dimensions can only be created, when a drawing already contains an existing linear dimension that can be used as a basis for the chain.

A continuing dimension always inherits the same properties as the existing dimensioning. For this reason it is only possible to connect dimensioning of the same type.

After issuing the command, a message will appear in the text window requesting the selection of an existing dimension. Select the desired dimension, taking care to place the selection cursor as close as possible to the point at that the continuing dimension is to be linked. Choose the second point by entering the relevant coordinates, or by selecting with the cursor. Whenever possible,

employ the object snap function for greater accuracy. The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as this will automatically be placed in line with the existing dimensioning.

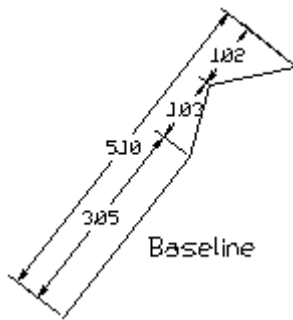
IE.

Select a dimension: P1

Pick 2nd point: P2

Verify or modify dimension text <1.02>: RETURN

## Baseline Dimensioning



Baseline Dimensioning adds a second measurement line to the drawing, parallel to the already existing dimensioning lines. This allows the user to read the cumulative dimension between two endpoints, extending over a number of individual dimensions within a chain.

This type of dimensioning assumes that a form of linear dimensioning already exists. The cumulative dimension will inherit the same properties as the existing measurements.

With the cursor, mark the starting point of the dimensioning chain, taking care to use the same direction of measurement as laid down during the creation of the individual dimensions within the chain.

Select the second point to be used for the baseline dimension by entering the coordinates at the keyboard or with the cursor. Whenever possible, enlist the help of the appropriate OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as this second line will automatically be drawn at a given distance from the existing dimensioning lines.

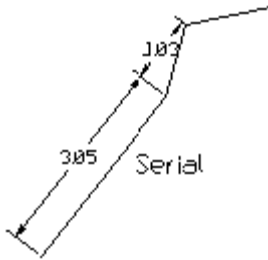
IE.

Select a dimensioning line (near start point): P1

Pick or enter second point: P2

Verify or modify dimensioning text <5.10>: Return

## Serial Dimensioning



Serial Dimensioning is carried out in the same way as described above for Continuing Dimensioning. However, where only one additional dimension can be added to an existing dimension with Continuing dimensioning, Serial dimensioning enables the user to link as many dimensions as required. This assumes that at least one dimension already exists that can be used as a basis for the rest.

It is only possible to add dimensioning of the same type since the additional dimensioning will inherit the properties of the existing dimensioning.

Select the dimensioning that is to act as the basis for the series, taking care to place the selection cursor as close as possible to the point at that the Continuing dimension is to be linked.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as it will automatically be placed in line with the existing dimensioning.

IE.

Select a dimension: P1

Series <continuous> RETURN

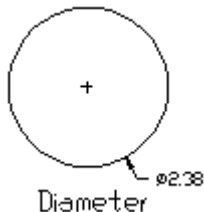
Pick 2nd Point: P2

Dimension text <1.03>: RETURN

Series <continuous>: RETURN

Pick 3rd Point: P3 ...Etc.

## Diameter Dimensioning



Diameter Dimensioning is used to display dimensions of circles and arcs. Unlike the horizontal or vertical dimensioning functions, no measurement lines are shown. In this case, a single extension line is used to show the relationship between the value and the object itself. The diameter symbol signifies that this is a special dimensioning feature used only in circle and arc dimensioning.

The first prompt will ask for the specific circle or arc. Select with the cursor. Please note that the point at that the user selected the object will be defined as the point at that the dimensioning extension line will connect to the element. The second prompt will ask for the position of the extension line. Select a coordinate at the required distance from the object. The alignment of the extension line will have been fixed by the position of the two selected points and cannot be altered. The extension line will be drawn along an imaginary straight alignment that extends from the middle point of the object to the measurement point, as defined during object selection (DEFPOINT). The extension line and measurement text will be displayed and a cross (+) will automatically appear at the center of the circle or arc.

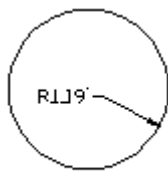
IE.

Select circle or arc: P1

Verify or modify dimensioning text <2.38>: RETURN

Extension length - Pick: P2

### Radius Dimensioning



Radius

Radius Dimensioning is carried out in the same way as diameter dimensioning, the only difference being that a radius dimension is calculated instead of a diameter measurement.

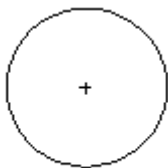
IE>

Select circle or arc: P1

Verify or modify dimensioning text <1.19>: RETURN

Extension length - pick: P2

### Center Dimensioning



Center

Center Dimensioning Center marking can only be used in conjunction with circles and arcs. It is used to position a mid point mark (+) at the center point of an arc or circle. The function is carried out in a single step. The user will be asked to select the desired circle or arc. Use the cursor to select the required element.

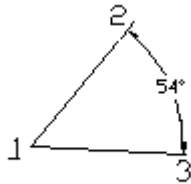


If a circle or arc is correctly selected the mid point mark will automatically appear. If it is not correctly selected the user will be urged to re-select the object.

IE.

Select circle or arc: The plus (+) sign is placed at the center point.

### Angular 3-Point Dimensioning



Angular- 3-Point

Angular 3-Points Dimensioning uses three measurement points that are placed at the apex and legs of an angle. This function not only allows the user to calculate the interior angle between the two legs, but also the exterior angle.

The user will be asked to select the vertex position. Either type in the coordinates, or select with the cursor. It is suggested that the user use the Object Snap options whenever applicable. Define a point on each of the legs of the angle. Define the location of the dimensioning arc by selection with the cursor. The dynamic preview function will assist the user in gaining a visual impression of the result. The position chosen for the dimensioning arc will also decide which angle (interior or exterior) is to be dimensioned. After confirmation, the program will display the measured value. This can be confirmed with RETURN, or amended as required.

IE.

Identify vertex: end of object: P1

First point: P2

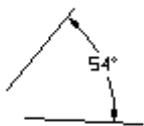
Second point: P3

Position dimension arc: P4

Verify or modify dimensioning text <54>: RETURN

Select dimension placement: RETURN

### Angular 4-Point Dimensioning



Angular 4-Point

Angular 4-Point Dimensioning differs from 3-point dimensioning in that the intersection of the lines (the center of the arc) does not need to be identified and picked. Either two lines or two points on each that define the line can be entered as input for this function. Either the interior or exterior angles can be measured and dimensioned.

The first prompt will ask for the appropriate objects. (lines, polylines, arcs or circles.)

I.E.

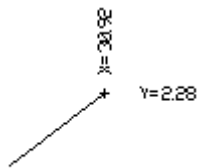
Select line1 (or 2 points that define it): P1

Select line2 or 2 points that define it): P2

Verify or modify dimensioning text <60°>: RETURN

Select dimension placement: RETURN

## Ordinate Dimensioning



Ordinate

Ordinate Dimensioning can be used to determine either the X or Y coordinate of a point and enter it into the drawing. These coordinates must either represent a point object within the drawing, or be that of a geometrically significant point of a drawing entity.

The X and Y options will be displayed within the options bar. By selecting one of the options, the user can determine whether the X or Y coordinate value is to be defined. Click the corresponding button, bearing in mind that the X value is set as the default and will therefore automatically be used if no other option is selected. The user will then be prompted to select the point to be measured. Type in the coordinate values, or select with the cursor. If the location represents a specific point on an entity, use the appropriate OSNAP. The program will ask for leader length. a dimension may be input or a position picked. The extension line originates at the point to be measured and connects to the measurement value text.

I.E.

Select Coordinate point: X

Pick point to be labeled: P1

Specify extension length: P2 (or specify location)

Verify or modify dimensioning text <2.28>: RETURN

## Advanced Concepts

### User Coordinate System (UCS) Setup

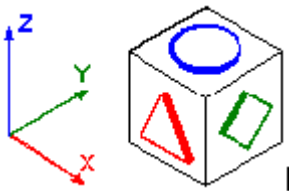
A **"User Coordinate System"** is a coordinate system that you define. It is primarily used for 3D drafting. Creating your own UCS allows you to drawn in "plan view" at any angle in space.

Until you create a UCS, you are drafting in the world coordinate system. With a UCS, you can:

- Move the origin (0,0,0) to any point in the world coordinate system.
- Turn one or more axes relative to the world coordinate system.
- Move and angle the user coordinate system.

The user coordinate system, for example, makes it possible to change the location of the drawing plane so that it always lies parallel to the objects to be drawn. This makes it easier to construct 3D objects.

To determine the coordinates of a point, distances are measured from the point to the X- and Y-axis; in a 3-D system, measurements are also taken along the Z-axis. AllenCad differentiates between a world coordinate system (WCS) and the user coordinate system (UCS).



### Creating a UCS

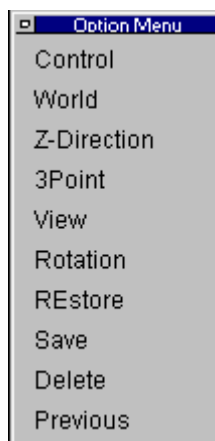
A UCS is created in two steps. First, you specify the origin and direction of the axes for the new UCS. Second, you save the UCS by name, so that you can reuse the UCS without having to re-enter its parameters.

The UCS command creates a new user coordinate system. At the "UCS origin <0,0,0>" prompt, enter the coordinates for the origin of the new user coordinate system. Or, press Enter to use the default values of 0,0,0.

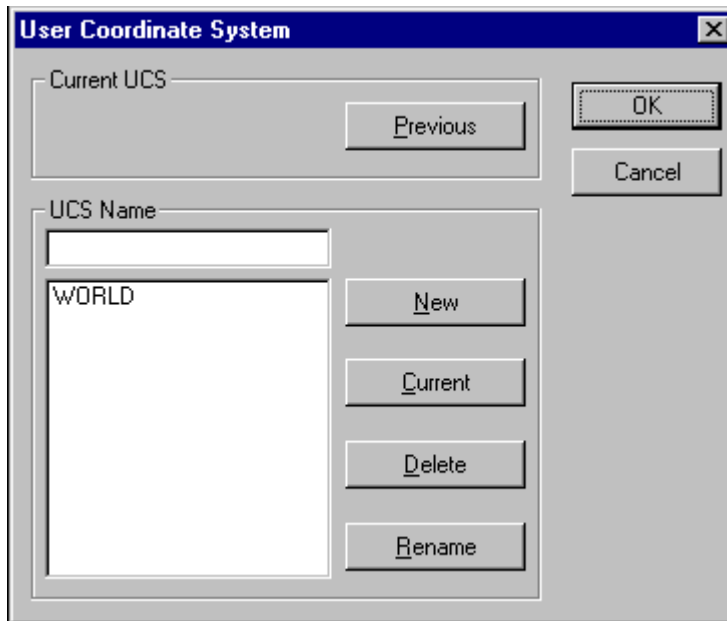
> UCS

UCS origin <0,0,0>:

Alternately, you can use the following options to interactively determine the origin of the user coordinate system and the direction of its axis:



**Control Option** - The Control option displays the User Coordinate System dialog box to select an existing UCS. The dialog box (below) also lets you create a new UCS, rename it, or even delete it.



**World Option** - The World option changes the UCS back to the world coordinate system.

**Z-Direction Option** - The Z-Direction option changes the parameters of the coordinate system in two steps:

1. You determine the origin by entering the new coordinates (in X,Y,Z format) or by picking a point .
2. The direction of the axes of the user coordinate system is determined by fixing the z-axis. To do that, choose a point through which the positive Z-axis should go. The X- and Y-axis remain the same. The XY-plane is orthogonal to the Z-axis.

>UCS

UCS origin <0,0,0>: Z-axis

UCS origin <0,0,0>: [Enter]

Point on the positive z-axis: P1

**3Point Option** - The 3Point option is meant for drawing planes that are neither parallel to the viewing plane nor to the axis of active coordinate system. We recommend you use the cursor to pick the coordinates:

1. You determine the origin by entering the new coordinates (in X,Y,Z format) or by picking a point .
2. Pick a point (or type coordinates) through which the positive X-axis should run. This determines the axis unambiguously.
3. The direction of the other axes is determined by fixing a point in the positive area of the XY-plane. This point does not necessarily have to lie on the Y-axis.

For practical reasons, you should locate the axes of the UCS so they are oriented with the plane of the drawing object. For example, choose the corner points of this plane with object snap. The selected plane becomes the drawing plane. All entities are drawn on this plane.

> UCS

UCS Origin <0,0,0>: 3Point

UCS Origin <0,0,0>: P1

Point on the positive x-axis: P2  
Point on the positive x-y plane: P3

**View Option** - The View option orients the user coordinate system parallel to the active view. The UCS command does not ask you for any coordinates. The X-axis of the coordinate system lies parallel to the horizontal axis of the viewing plane; the Y-axis is oriented vertically; and the z-axis is orthogonal (at right angles) to it

> UCS  
UCS origin <0,0,0>: View

**Rotation Option** - The Rotation option rotates the active coordinate system around one axis. You may choose the angle of rotation and the direction of the rotation.

> UCS  
UCS origin <0,0,0>: Rotate  
Rotate around x-, y-, or z-axis: Z  
Rotation around Z-Axis <0>: 30

To rotate the coordinate system around two or three axes, repeat these steps for each axis:

1. Determine the axis around which to rotate the coordinate system. Either type its character or click its option (X, Y, or Z) from the option bar.
2. Determine the direction and angle of the rotation. Entering or pointing an angle does this.

You specify the angle of the rotation by typing the angle or by picking two points in the drawing. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system. Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example, -30) results in a rotation to the right.

### Controlling the UCS

Of course, it would not be very effective if you had to make the above adjustments every time you needed to change the coordinate system. Therefore, AllenCad allows you to save pre-defined user coordinate systems by name with the drawing. Once a UCS is saved, you can always reuse it; the UCS names can also be renamed or deleted.

All this is possible via the UCS command's **Control** option. Or, from the menu bar, select Options > User Coordinate System > UCS Manager.

### Creating a New UCS

You create a new user coordinate system in these steps:

1. Create the UCS with an option as described above: Z-Direction, 3Point, View, or Rotate.
2. Open the dialog box and type a name in the UCS Name field. The name can be up to 31 characters long, as well as dash ( - ) and underscore ( \_ ), but no spaces.
3. Click New.
4. Click OK.

### Renaming a UCS

If you need to rename a UCS:

1. Select a UCS name from UCS Name list.
2. Click the name and type a the new name.
3. Click Rename. This verifies the renaming; the new name of the UCS appears in the field.
4. Click OK.

**Note:** the world coordinate system cannot be renamed.

### **Deleting a UCS**

Erasing a UCS is similar to renaming a user coordinate system:

1. Select a UCS name from the UCS Name list.
2. Click Delete. The UCS is then deleted; its name no longer appears in the list.
3. Click OK.

**Note:** the world coordinate system cannot be deleted.

### **Changing the Active UCS**

You can create as many user coordinate systems as you need in a drawing, but only one user coordinate system can be the active coordinate system. To change the active UCS:

1. Select a UCS name from the UCS Name list.
2. Click Active. The user coordinate system becomes the active coordinate system. The field shows the name of the active coordinate system.
3. Click OK.

### **Reverting to the Previous Coordinate System**

To change to a previous coordinate system, click the Previous button. It enables you to quickly change between two coordinate systems. When you click this button, AllenCad changes to the previous coordinate system without verification.

## **Drawing with Hidden Lines**

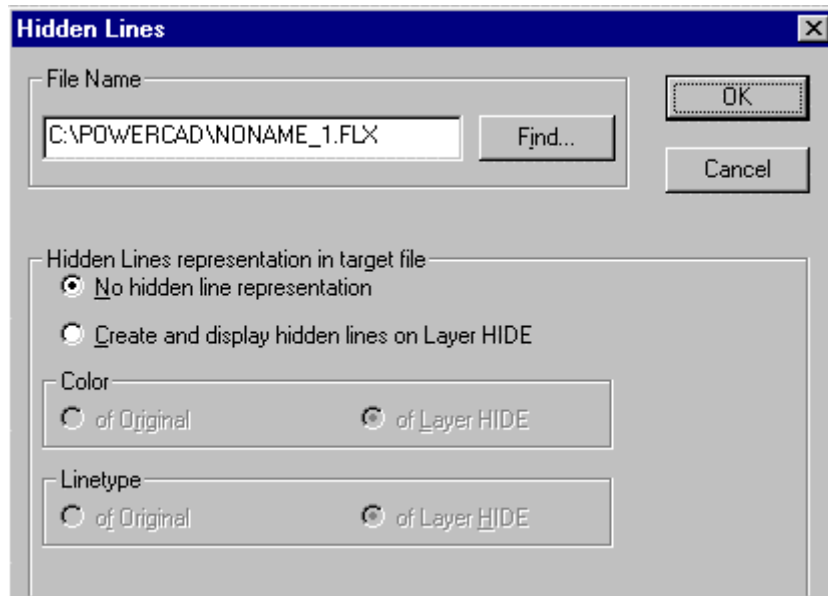
Three-dimensional models are created in AllenCad, initially as a wire frame drawing. With the 3DView command, you can view the 3D geometry in predefined isometric views, as well as from any 3D view direction. When opening additional windows of the current drawing, you can specify independent 3D view directions for the new viewport.

With most view commands, AllenCad displays all lines that make up the wire frame model. To display a more realistic image of your 3D model, you can have AllenCad remove the lines that are hidden from your view (called "hidden line removal"), and to creating full-color renderings the 3D model.

This chapter describes the visualization commands, which are found in the View menu:

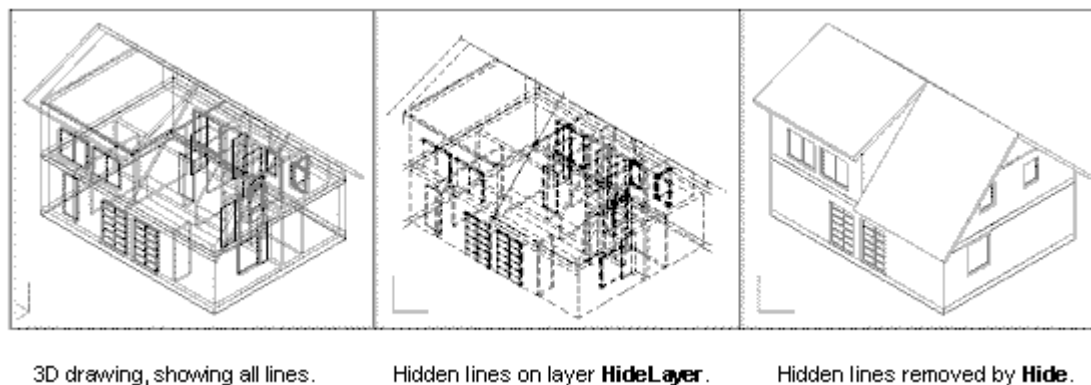
<b>Visualization Command</b>	<b>Meaning</b>
HIDE	Create a new hidden-line drawing in a separate window
FHIDE	Perform a fast hidden line removal
RENDER	Render the 3D drawing
RCONFIG	Specify parameters for the rendering
LIGHT	Insert a light to illuminate the rendering
LIGHTEDIT	Edit the light parameters
ARENDER	Automatically create a rendering

The HIDE command (View > Hidden Lines > Hidden Line Drawing) creates a separate .flx drawing file containing the current view with the hidden lines removed.



As an option, lines and edges that are computed as hidden, are placed on a system-generated layer named "HideLayer" (see figure below). You can turn on or off this layer, assign them a dashed linetype or another color. In addition, you can remove lines from that special layer and retain those you want to keep for demonstration purposes.

Note that the "3D" hidden-line view is actually a flattened 2D drawing.



The **FHide** and **Render** commands are based on the OpenGL graphics system from Microsoft, which provides high-performance hidden-line removal and rendering

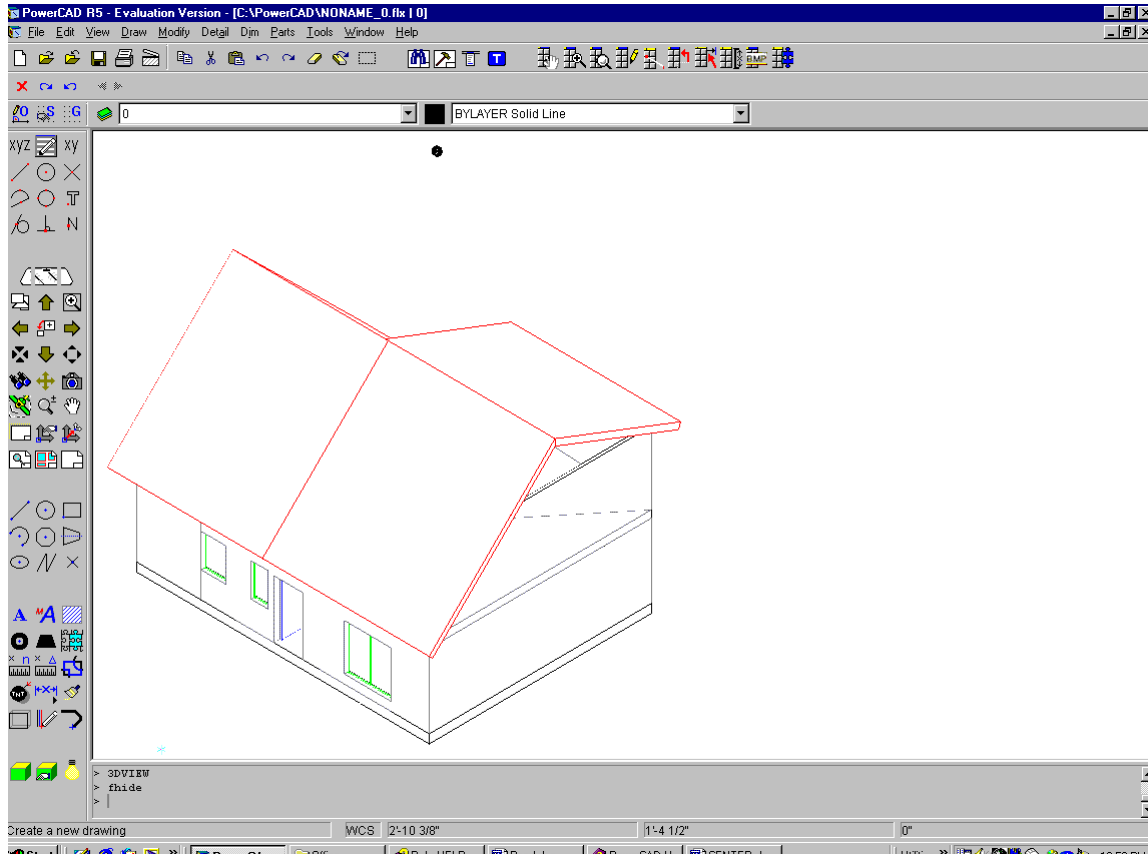
### FAST HIDE

The Fhide command (View > Hidden Lines > Fast Hide) creates a hidden line view in the

current drawing. (This is in contrast to the Hide command, which outputs the hidden lines to a new drawing.) This FHide command provides you a visual impression of the 3D model from the current viewpoint.

You can continue to draw in hidden-line display mode, even using object snap functions. The hidden-line representation is maintained until a Redraw, Regen, Zoom, or Pan command refreshes the drawing window.

The FHide command is based on the OpenGL system from Microsoft, which allows high performance hidden-line removal.



## Using and Editing Raster Images

### The RASTER PALETTE





The View > Raster menu item opens the Raster palette (toolbox) with most of the commands available on it.

All raster editing and manipulation commands are available for RLC (large scanner) format. Only some of the editing commands are available for TIF and BMP files as noted in the following pages.

There are conversion programs on the market that may assist you in changing from another format to one of these supported formats. AllenCad includes a converter for RLC<>BMP (monochrome only)

#### Notes:

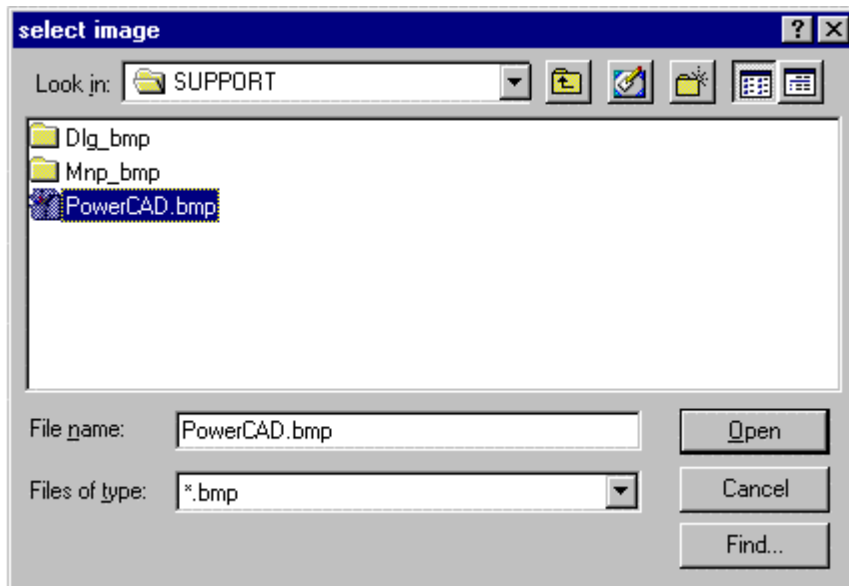
1. It is necessary to ensure your windows system is setup to view ALL file types otherwise you will not be able to view/locate it
2. Because inserted raster images do not become integrated in the drawing file, it is important to not delete or move the original raster image file so the link between the two remains intact.

#### LOAD RASTER FILE (RLOAD)

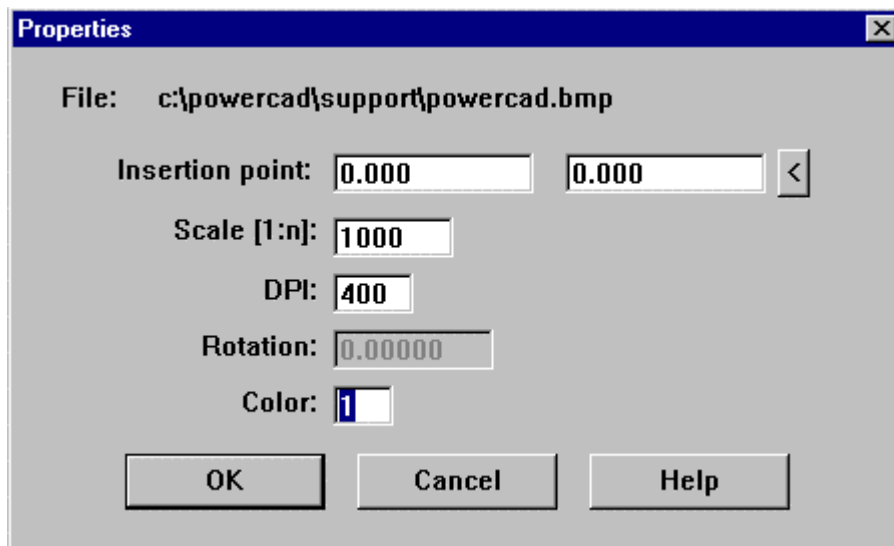


This palette button inserts a scanned or raster image from a file into the current drawing. For version 4.01 and higher, the use of color tif and .bmp file formats are available. These two formats have been added to the existing .rlc format from previous releases.

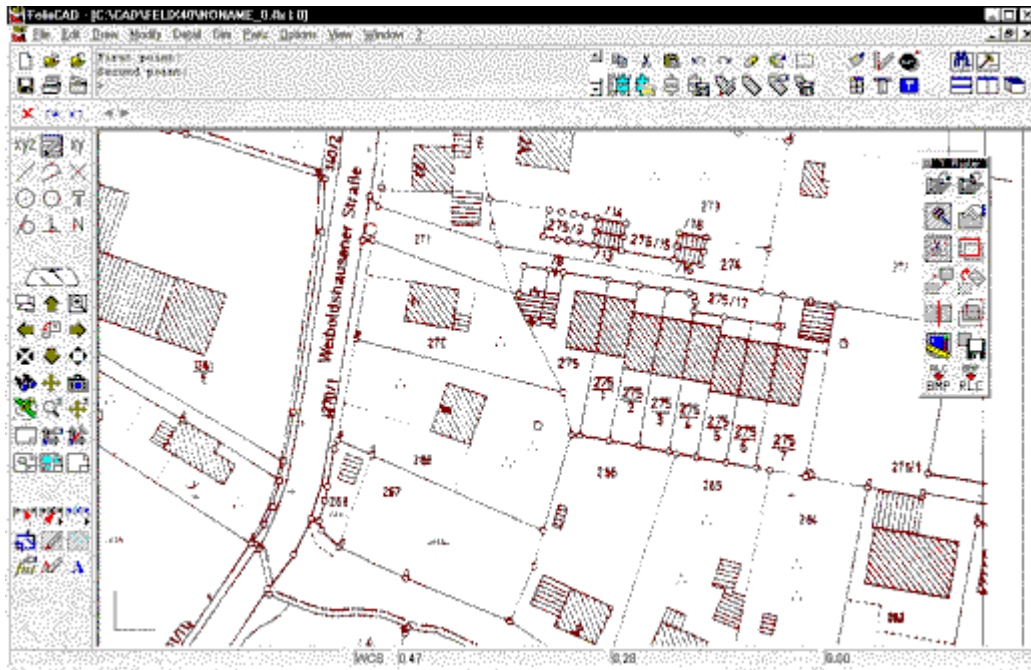
The **Load Raster File** button displays the Select File dialog box to allow you to select the raster image you wish to insert into the drawing. Pick the raster file type (RLC, TIF, or BMP) from the Files of Type pick list. AllenCad is currently limited to two raster images at one time.



Once you select an image, the following dialog box ask you where to insert the image, the scale factor, the color of the image, the DPI (dots per inch), and the rotation factor.



Once you have entered the information, the raster image appears on the screen. You may need to zoom to the extents of the image using the zoom tools on the control panel to see the complete image.



## REMOVE RASTER (RUNLOAD)



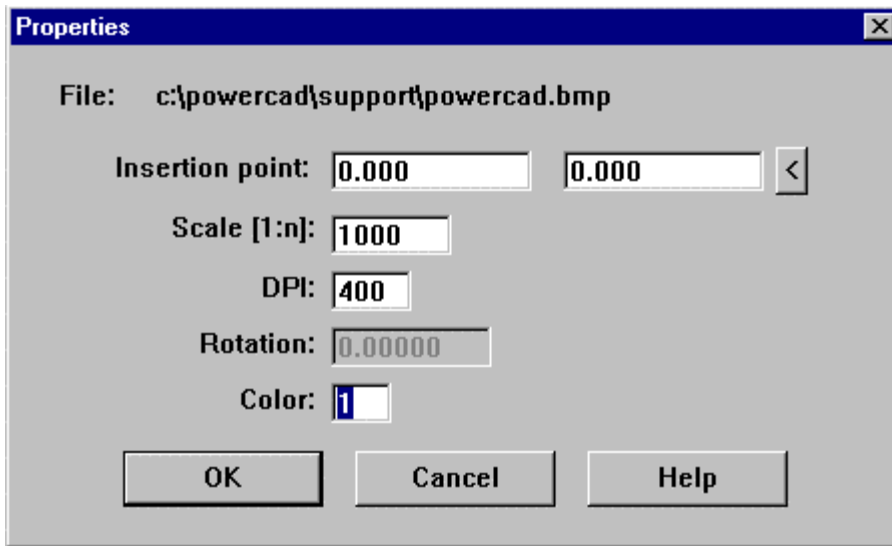
This **Remove Raster** button removes the selected image from the drawing. If two images are currently loaded, the system will prompt for which image to delete.

## PROPERTY SETTINGS OF RASTER FILE (RPROP)



The **Property Settings of Raster File** button brings the following dialog box and allows you to change any of the fields. The changes are only done to the current image and do not affect the original stored on your computer.

- **Insertion point:** location of the image's lower-left corner.
- **Scale [1:n]:** Sets the scale factor to make the raster image larger or smaller.
- **DPI:** dots per inch, affects the size of the image.
- **Rotation:** orients the image in reference to the drawing. Positive rotation is counterclockwise, negative rotation is clockwise.
- **Color:** Sets the color the entire image takes on:



#### SAVE RASTER (RSAVE)



The **Save Raster** button allows you to save all of the changes to the image, done by the Property Settings Raster File button, back to your hard drive.

These changes are permanent, so be careful and make sure you have a back up of the original raster image before saving the changes.

#### TOGGLE RASTER FRAME ON/OFF (RBORDER)



The **Toggle Raster Frame On/Off** button toggles the rectangular frame border around the image. The frame is turned either off, or on. The frames of all raster images are affected at the same time. When on, the frame shows the outer edge of the image and is in the same color as the image.

#### MOVE RASTER (RMOVE)



The **Move Raster** button moves an image in the drawing, as the name suggests. If you have more than one raster image in the drawing, you are asked to select the one to be moved:

select raster /<point>/?/Last/#n [#0]:

The options have the following meaning:

- **<point>**: you select the raster image with the cursor.
- **Last**: select the last image selected.
- **#0**: Type the id number of the image; the first image is #0.
- **?**: displays all numbers to choose from.

You are then asked for a Base point and a Destination point, just like the standard Move command.

#### ROTATE RASTER (RROTATE)



The Rotate Raster button lets you rotate RLC and TIF raster images (does not support BMP

images) to match your vector drawing. The prompt is as follows:  
select raster /<point>/?/Last/#n [#0]:

This allows you to select the correct raster image to rotate, using the same options as for the **Move Raster** command. Next, you are prompted:

Rotation angle:

This is the amount of rotation: clockwise is a negative rotation and counter-clockwise is a positive rotation.

### MIRROR RASTER (RMIRROR)



The **Mirror Raster** button (RLC only) allows you to mirror image RLC files only.

You first select the image on screen:

select raster /<point>/?/Last/#n [#0]:

After selecting the image, you are asked;

mirror axis:

Two options appear on the options bar: **Horizontal** and **Vertical**. Pick the one you wish to use. After a few seconds, the raster image is rotated and AllenCad displays the **Properties** dialog box. This lets you specify the properties of the mirrored copy.

The original image is not removed. The mirrored image is displayed according to the image properties.

### RMONT



The **RMont** command (RLC only) lets you copy a portion of, or the whole, raster image.

You are prompted to place a window around the portion to be copied. This is similar to the **Zoom Window** command. You are prompted for the **First Corner** and then the **Second Corner**.

After picking the area to copy, you are asked to confirm the properties of the new image. The changes are only for this drawing session, unless you save the property changes.

### RCUT



The **RCut** command (RLC only) lets you remove a portion of the raster image. You must first draw a closed polyline around the area to be clipped. The polyline must be a series of straight-line segments; arcs will be treated as if the chord of the arc was the boundary. You must close the polyline.

You will be prompted to pick the raster image to clip, just as in the move image and rotate image commands.

select raster /<point>/?/Last/#n [#0]:

Then you are asked to pick the polyline to use for the clipping. Finally, the command asks:

save properties for 'C:\Image.rlc'?:

If you answer **no**, then the image is simply clipped and that portion falling within the polyline boundary is removed from the screen and the properties remain unchanged. If you answer yes, the following is displayed;

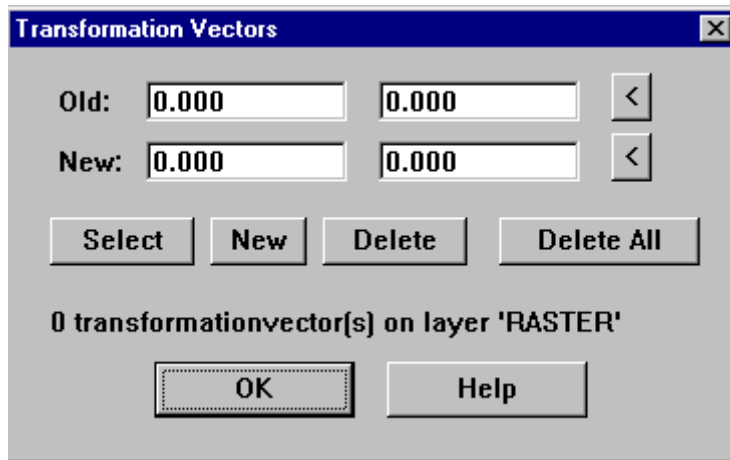
properties saved for 'C:\Image.rlc'.

Then the image is cut or clipped inside of the polyline and the properties are saved for the original raster image.

## RVECTORS



The **RVectors** command (RLC only) edits the transformation vectors required to adjust the raster image to match the vector drawing. The vectors you define will be drawn on the layer "RASTER." You can actually draw the vectors before starting this command and it will detect them for you.



- **Old** is the current starting location of the currently selected transformation vector. You can type a new location if you wish or use the < button to visually pick a location on the screen.
- **New** is the current ending location of the currently selected transformation vector. You can type in a new location if you wish or use the < button to visually pick a location on the screen.
- The **Select** button will let you select which vector you are currently editing. You pick the vector on screen.
- The **New** button create a new vector from the coordinates displayed in the Old and New fields. It duplicates a vector if you pick this button while old values are in the fields above, but you can still manipulate those vectors by using the two < buttons to the right of the New and Old coordinates.
- The **Delete** button deletes the currently selected vector, both from the routine and the screen.
- The **Delete All** button deletes all vectors on the layer "RASTER."

### Raster Layer Display

This location displays the number of vectors detected on the layer "RASTER." Do NOT use this layer for anything else or you will confuse the program.

### 4-POINT TRANSFORMATION (RTRANS)

This command performs a simple transformation that stretches the raster image to match your vector line work. You are required to enter four vector translations in the edit/define transformation vectors menu. Then when you select this option the stretching of the raster will be done. Make sure you enter exactly four transformation vectors, since this routine will not run with more or less than four! Once the transformation is completed, you will be prompted for the properties to confirm the insertion point color, etc.

### PRINT/PLOT SORTING

For printers and plotters that can output raster images (such as inkjet, laser, electrostatic, and dot matrix), you can control whether you want the vectors printed on top of the raster image -- or the other way round. This way you can see the vectors on top of the image, if desired. The prompt of the PrintOpt command is, as follows:

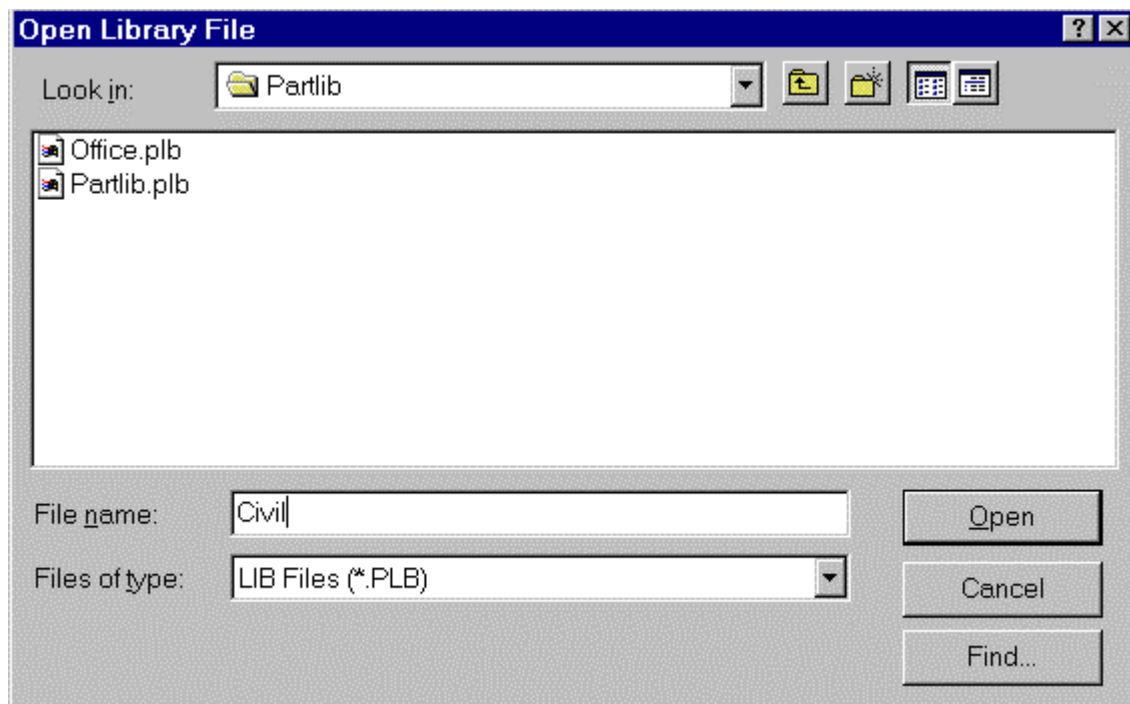
print raster before vector or after vector [Before]:

Valid replies are **B** for before or **A** for after.

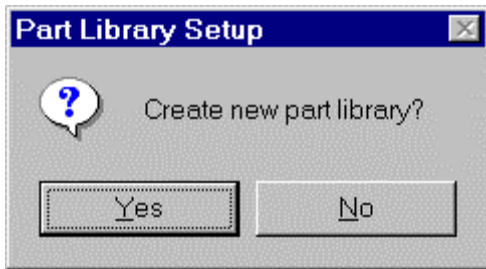
Now you can go ahead and plot your diagram as you normally would. The raster will now print/plot as you have indicated.

## Part Library Setup

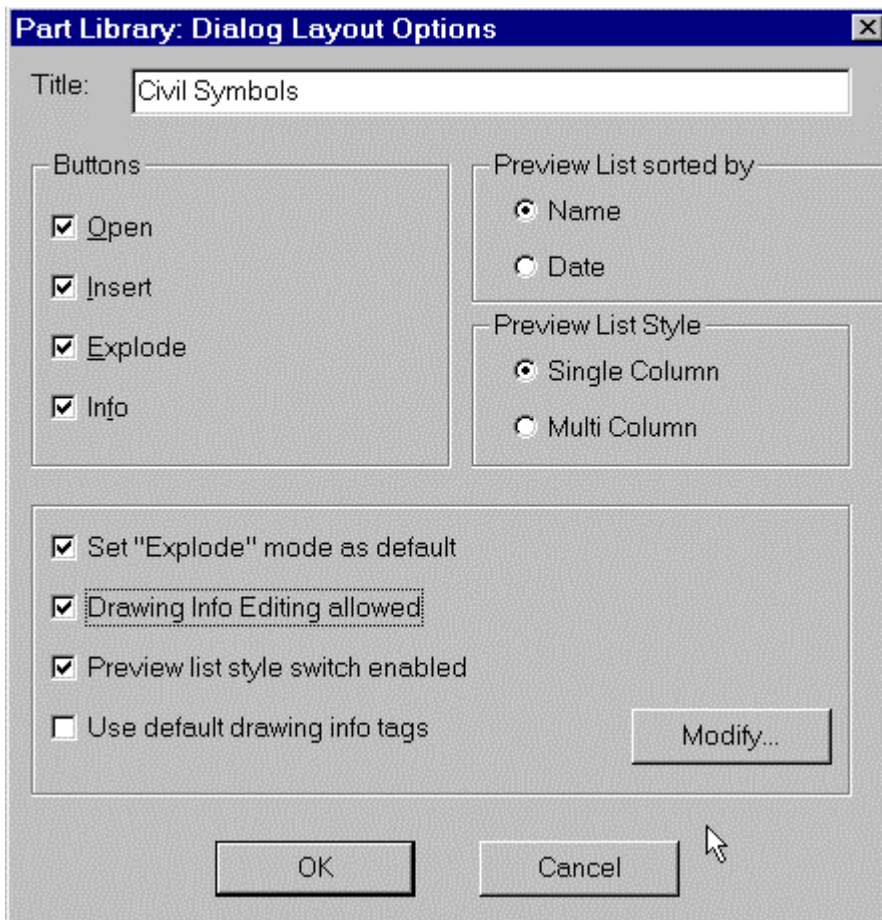
The command PLBSETUP (Parts – "Part Menu setup") allows the user to set up new part libraries and/or modify existing ones. Parts must be .flx format drawing files to be used in the library and by default, the program looks in the /Partlib/flx subdirectory of the program for these files. When the command is issued, the program presents the following dialog:



In this instance, we have a number of civil engineering symbols that we wish to create a library for, so we will enter the name Civil in the file name space and click on the Open button. When we do, the program recognizes that this is a new library name and presents us with a the confirmation dialog below:

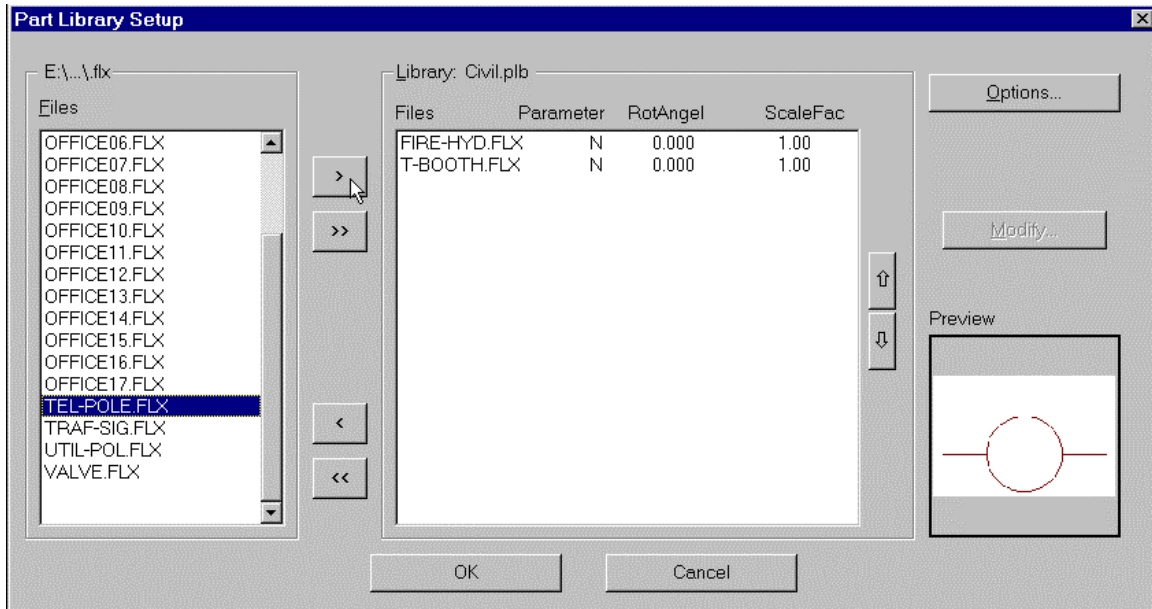


After confirming by clicking on Yes, AllenCad opens the dialog "Layout Options" below:



Here we can give the library a title and set various insertion and viewing options for the symbols in the library. When we click OK, the program presents us with a dialog box that allows us to specify which files to include in the library:

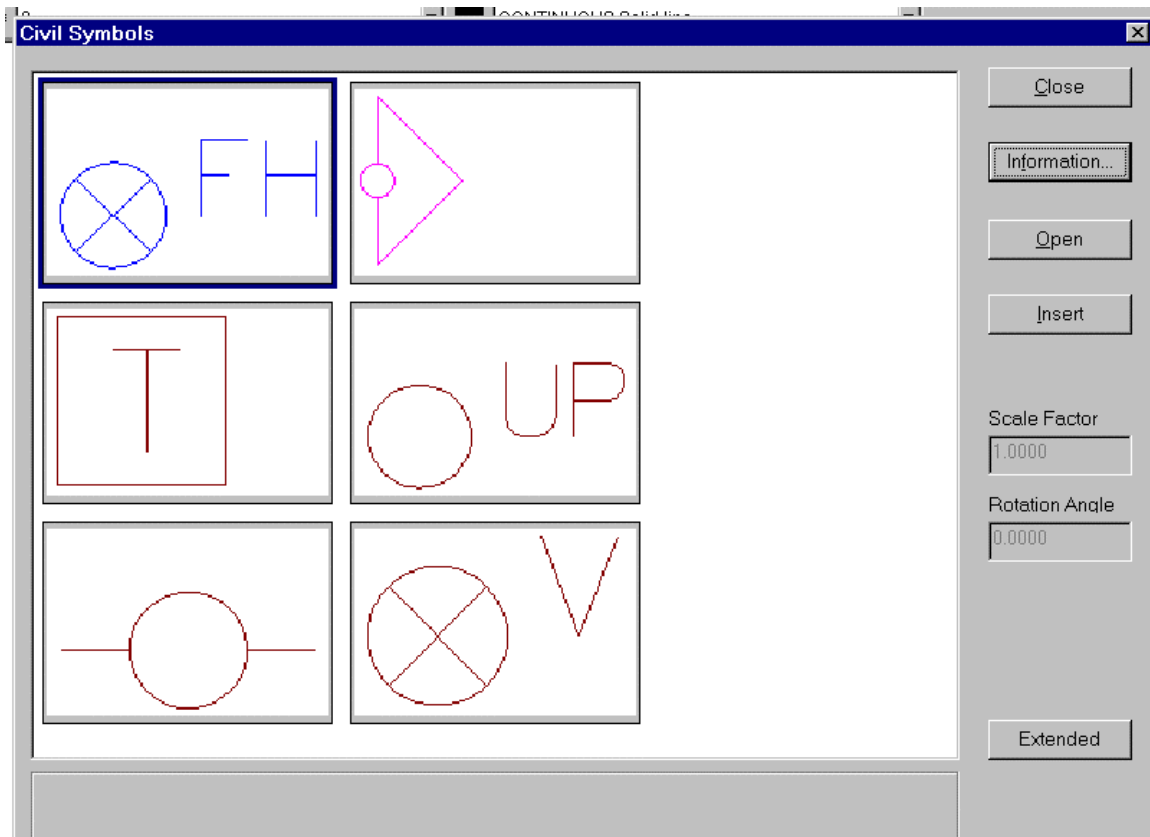




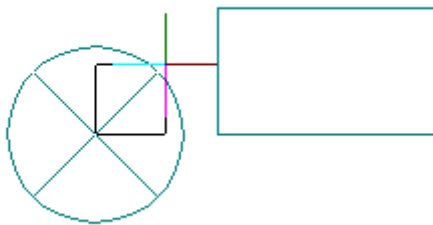
In the left window, all symbol files in the Partlib\flx subdirectory are listed. We can highlight those that we want to include and click on the ">" button to add them. The "<" button will de-select files that we have added but later change our mind on. Simply select the file in the central window with a mouse click and click on "<" to remove it from the list. The buttons ">>" and "<<" simply add and remove all files in the lists. Clicking on OK completes the library setup.

## Using Part Libraries

The PARTLIB command (Parts – "Part / Symbol Library") gives the user access to the files included in the library. One Part Library is default at any one time and this default can be changed, by clicking on "Set Current Library" in the Parts pull-down menu. When the PARTLIB command is issued, the user sees the following dialog:



At this point, the user can scroll through the "thumbnails" of the symbols contained in the current library and can select one for insertion by clicking first on the symbol and then on the Insert button. To confirm identification of a selected symbol, the user can simply click on the information button to obtain file information. Here we have clicked upon the fire hydrant symbol and when we press the Insert button, we see the symbol being lead into position by the cursor:



Having specified the symbol to insert, the program requires three additional pieces of information before the insertion is complete:

- **Insertion point** - At this point, we can either click to set the insertion location with the mouse or enter insertion point coordinates at the command prompt in X,Y,Z format.
- **Horizontal and Vertical scale factor** can be established by dragging with the cursor (see the rectangle) created onscreen in the above image. This is the rectangle described by the center of the circle (the symbol insertion point) and the opposite corner at the cursor

location. This method may be used where precision is not a factor, however it is also possible to enter a horizontal scale factor, followed by a vertical scale factor, at the command prompt. If the vertical scale factor is the same as the entered horizontal factor (assumed default), the user may simply press the enter key a second time without entering a vertical factor value.

- **Rotation Angle** - The last step in the insertion process is to set a rotation angle for the symbol. The default value is zero degrees, however another value may be entered at the command line or else set by with a mouse click when the cursor sets the desired orientation. This completes the symbol insertion, with the symbol appearing in the drawing as below:



## Part Creation and Insertion

Besides inserting parts from designated Part Libraries, it is also possible to work with other drawing files (or parts as they are called when used in other drawing files). Parts (or blocks as they are sometimes referred to) can be either internal or external to the current drawing.

- **Internal Parts** - An internal part or block can be created from selected drawing objects using the **BLOCK** command. Besides selecting the entities that comprise the block, the program also prompts you for a name to assign to the block. After creation, the block can be inserted repeatedly, using the **INSERT (Parts – Insert Part)** command. This saves much time, as the elements do not need to be redrawn for each application. Internal parts are available for insertion only within the drawing file in which they have been created.
- **External Parts** – An external part is simply an external drawing file that may be used (inserted) in the current drawing. These files can be in either .dwg or .flx format and can be created either as a separate drawing exercise, or else from within the current drawing, using the **WBLOCK** command. This command functions almost identically to the **BLOCK** command, with the exception that the file is saved externally and the user can specify where the file is to be stored.

**Inserting Parts** – As mentioned above, internal and external parts can be inserted using the **INSERT** command. When this command is issued, the following dialog appears:

**Insert Part**

Part Name

FIRE-HYD

External Internal

Options

	X:	Y:	Z:	Interactive:
Insertion at	0.0000	0.0000	0.0000	<input checked="" type="checkbox"/>
Scale Factor	1	1	1	<input type="checkbox"/>
Rotation	0			<input type="checkbox"/>

Default ☐ Exploded

OK Cancel

By default, the last part inserted into the drawing is offered for selection. The user can also specify other internal and external parts to insert, as well as specifying insertion point, scale factor rotation and also whether the part will be exploded (broken down into individual component objects) upon insertion. Note particularly the Interactive check boxes. When these are checked, the user can input values, either by command line entry or else establishing location or magnitude through mouse picks on the drawing surface.

## Layout Mode and Model Mode

When you first begin your drawing, you are in what is termed "**Model Mode**". In model mode, you generally develop your drawing model, consisting of lines, arcs, dimensioning and other entities. In some situations, it can be very helpful to use another mode - "**Layout Mode**" to prepare your drawing presentation prior to printing. In Layout mode, you can present views of your drawing model from different angles and group them on a single drawing sheet, for instance. Alternatively, if you have a very large model, you can show different sections of the model on different drawings or at different scales if required.

You use the TLMODE command (View pull-down – Layout Mode/ Model Space) to switch back and forth between these modes.

While you can draw entities in either mode, you are best advised to keep layout mode drawing

elements limited to specific presentation elements, such as title blocks and presentation notations. All other drawing elements should generally be drawn in model mode, where they complete your drawing model and are also available as required for use in other views.

You can think of Layout mode as being similar to covering up your drawing with a large sheet of paper. In layout mode, you cannot see your drawing until you cut windows in the sheet. These rectangular windows are what we call viewports (see Using Drawing Windows) and are created using the VIEWPORT command. Several viewports can be "cut" and the picture of the model within each viewport can be shifted, rotated, scaled or viewed from a different 3 dimensional angle, as needed for the presentation. The picture of the drawing model in each viewport is entirely independent of the picture in another viewport.

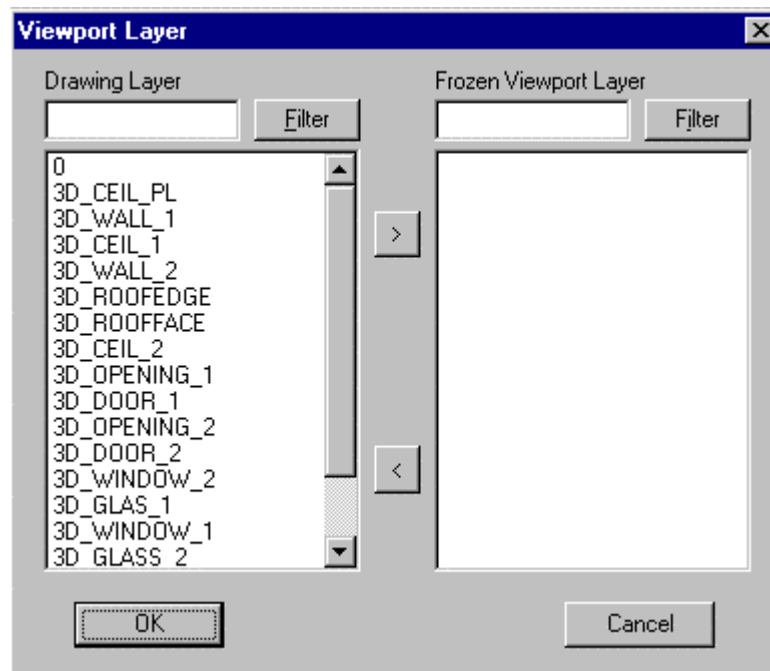
The VIEWPORT command calls up the main option dialog as pictured below:



You will use the **New** option to create a new viewport. This option will prompt you to select two opposite corners that define the location of the rectangular viewport

The **On** or **Off** options control the display of the viewport, and you will be prompted to select the viewport frame after selecting these options.

The **Layer** and **Player** options to control the entities that are displayed in each viewport. These commands prompt for the viewport selection and then present you with the following dialog:



All drawing layers are listed in the left window. Those that you wish not to display in the viewport can be selected and moved to the right window using the > button. Similarly, selected can be de-selected by highlighting them in the right window and moving them to the left window by pressing the < button.

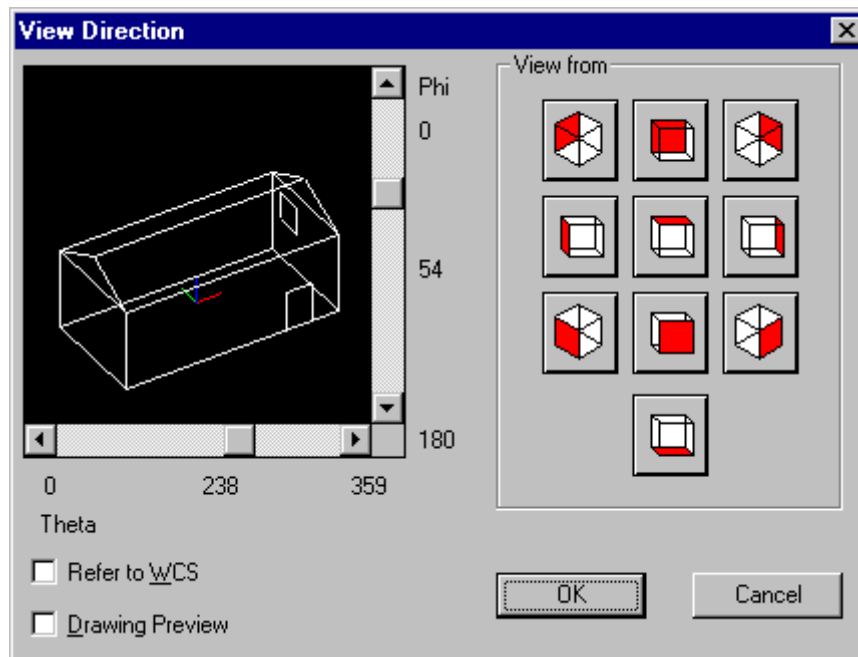
The **View** option is used to manage the image scale, orientation and position within the viewport. You will be asked to select the viewport and then presented with the following dialog:



The **Zoom** and **Ratio** options are used to control the size of the image while **Pan** will move the image within the window.

**Nview** will place a named view within the viewport. These are often created in model mode using the **WOPEN** and **VIEW** commands.

The **3DView** option prompts for viewport selection and then produces the dialog:

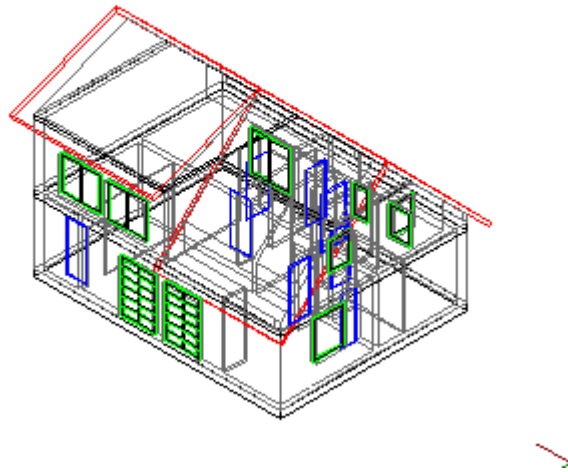


With this, you can select the orientation of the view of the drawing model.

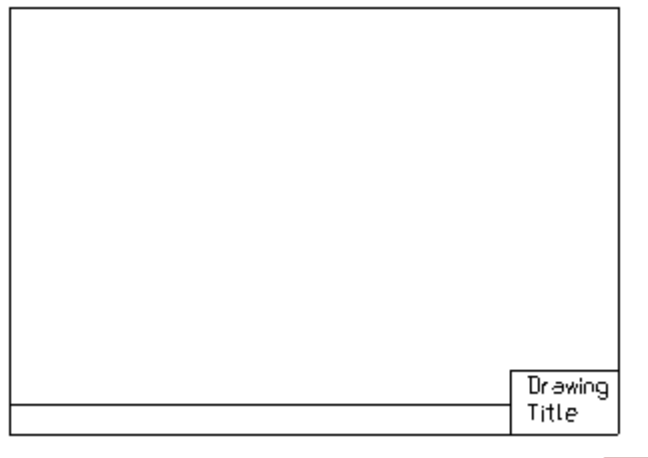
Similarly, **Viewpoint** will allow you to input 3 dimensional coordinates of the point of viewing as well as an option for rotating the image.

With that in mind, we will use our sample drawing "House" to demonstrate the use of viewports and Layout and model modes:

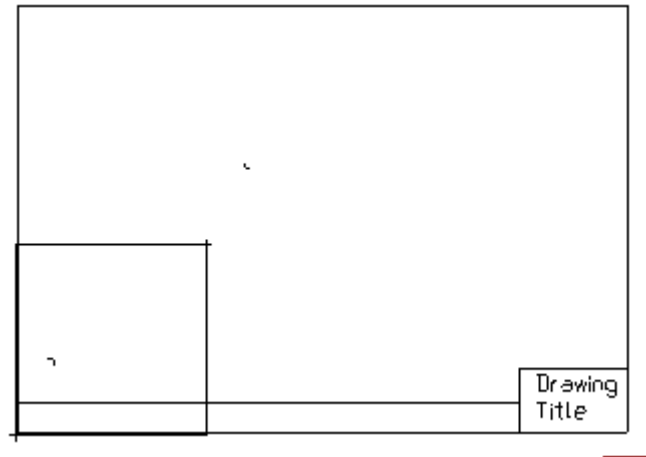
Opening up the drawing, we are in Model mode - the mode in which the house model was created. The view that we have of the model is as below:



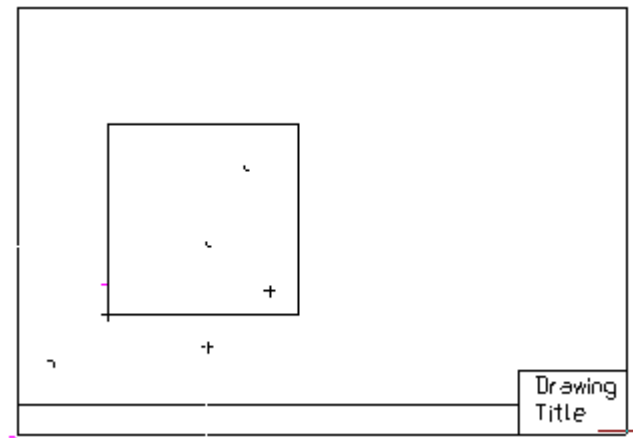
Next, we will toggle into Layout mode by issuing the command `TLMODE`. In this mode we are faced with a blank edit window and we will next insert our sample title block and surround. This we will call up using the `INSERT` command and specify the external part "Title". This is an A1-sized drawing sheet and we will insert it at a 1:1 scale factor. At this point our Layout space view of the drawing appears thus:



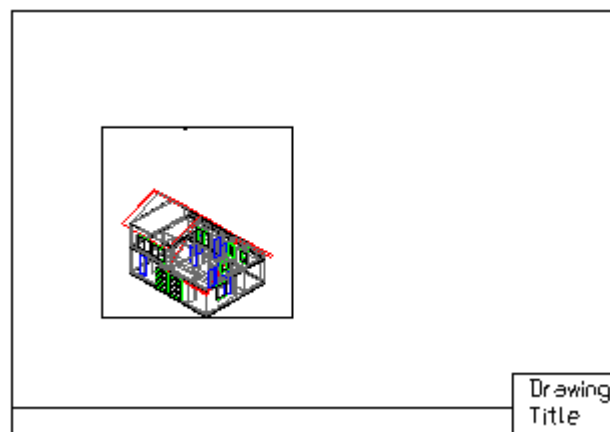
Next, we want to open up two viewports to show the drawing "underneath" from two different vantage points within the drawing surround. We will size the viewports initially to be 250mm x 250mm so we will draw a rectangle using the `RECTANGLE` command and specifying corners as 0,0 and 250,250. Our drawing now looks like:



Next, we want to move the rectangle into position so we will issue the MOVE command and selecting the rectangle, move it from 0,0 to 120,160. At this point we have:



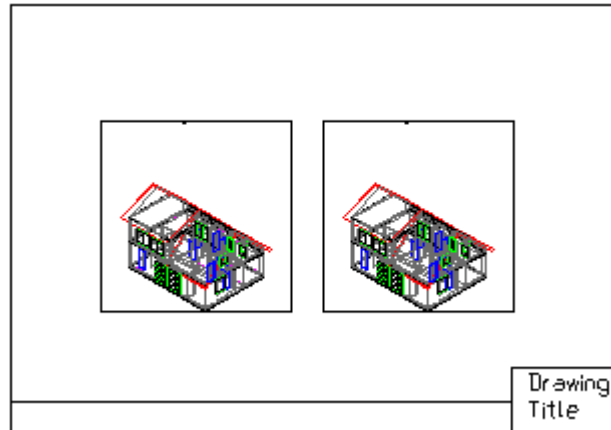
Next, we want to open up a view to the drawing model so we issue the command VIEWPORT and select the New option, then pick opposite corners of our rectangle. We now have a drawing that looks like:



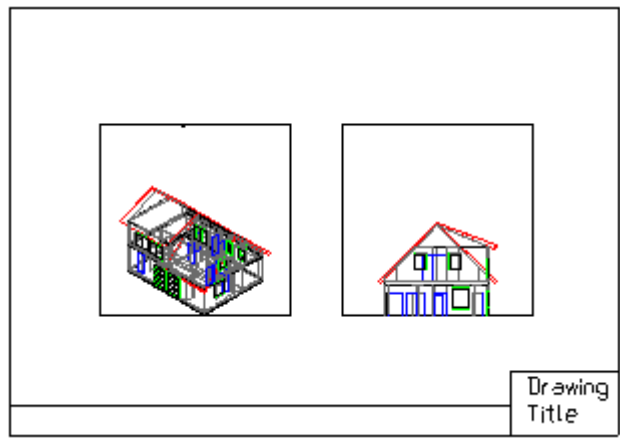
In this drawing, we want to have a second view of the house from a different angle. We could



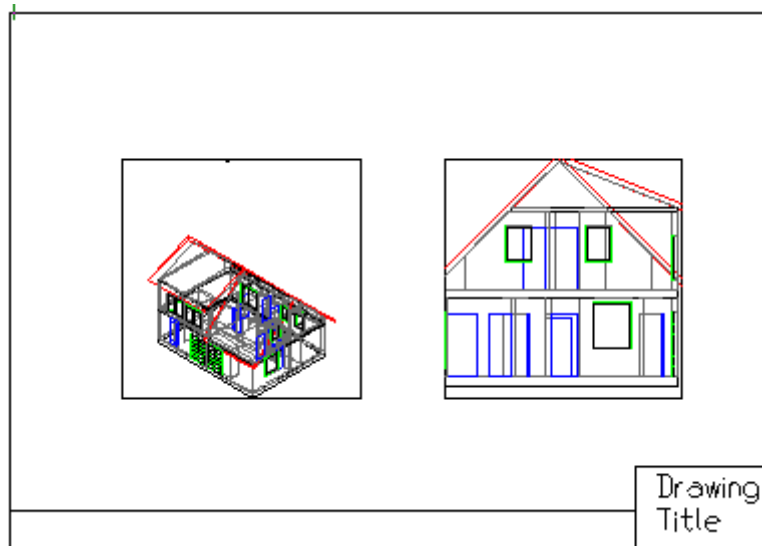
repeat the process for a second viewport, however we can also copy the existing viewport and save time. We will therefore switch on Ortho mode (to ensure the viewports line up with each other), then we will use the COPY command then select the viewport frame to copy the viewport to the open space to the right of the existing one:



This view of the model is in the same orientation as in the first viewport, so we need to issue the viewport command again and select the View option. We are asked to select a viewport to change and we will select the right viewport frame. Because we want an end view, we will select the 3Dview option in the dialog box and pick the view orientation we want from the dialog box. At this point, our drawing looks like:

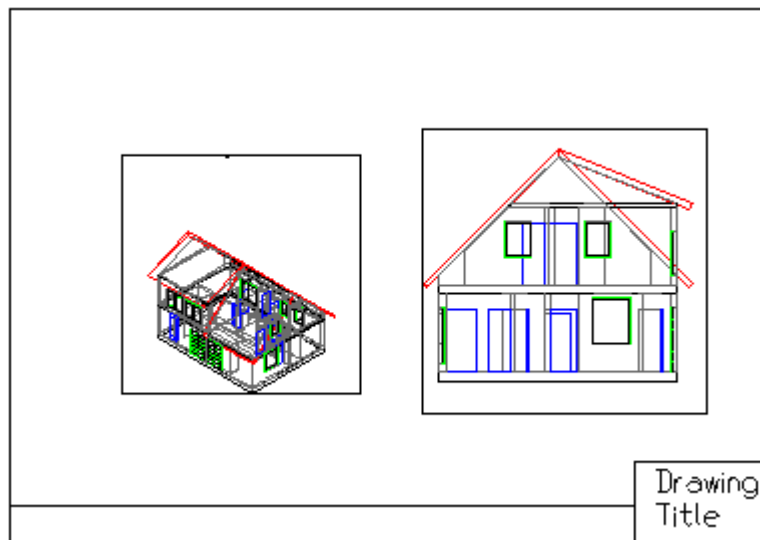


But... we decide that we want to show this at double the scale as the first, so we return to the Viewport command and select View, then select the Viewport, then the option Zoom and then apply a factor of 2. We can see that we need to pan a bit to center the View of the end of the building, so we use the Viewport's Pan option before pressing return to complete the command. At this point, our drawing appears as:



We now have the proportions we need, but see that the viewport itself is a bit small. We will therefore use the Scale command and select a factor of 1.2 to apply to the viewport frame. This gives us the final drawing views that we want on this drawing.

**Note:** Normal modify commands apply to the viewport frame. To change the image within the viewport, you will need to use one of the Viewport command options.



Should we want to create more views, we could copy the title block and viewports that we want and then edit the views to produce a second drawing with the necessary perspectives. Also, we can return to Model mode at any time by toggling back from Layout mode using the TLMODE command. This we may want to do if we wish to edit anything in the drawing model.

### Using Relative Coordinates

To this point, we have been defining points using absolute (X,Y,Z) coordinates. At times, however, it is faster and more convenient to define a point location relative to another point in the drawing.

Take for example, a situation where you are drawing line segments and you know the length and angle

(bearing) of each vertex relative to the preceding vertex. You could calculate all of the vertex coordinates in advance and enter them in absolute coordinates as you draw your line, or you could more simply use relative coordinate input, to define each point relative the previous point.

Two symbols are used when using relative coordinates. The @ symbol indicates that the following coordinate entry is **relative** to the last picked or entered point and < indicates that the number immediately following it is an angle. **All values that follow @ in these expressions are relative to the previous point.**

**Note:** it is acceptable to mix relative and absolute expressions with subsequent points. In other words, you could use a relative expression to establish point 2, an absolute (x,y,z) expression to fix point 3 and another relative expression to locate point 4.

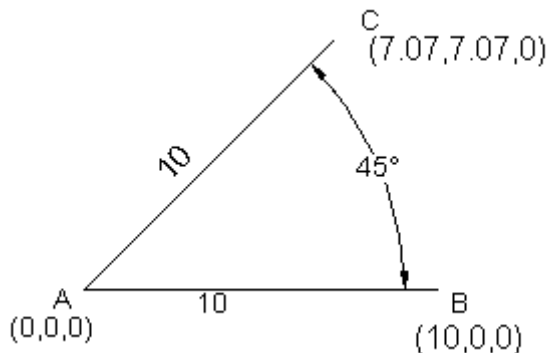
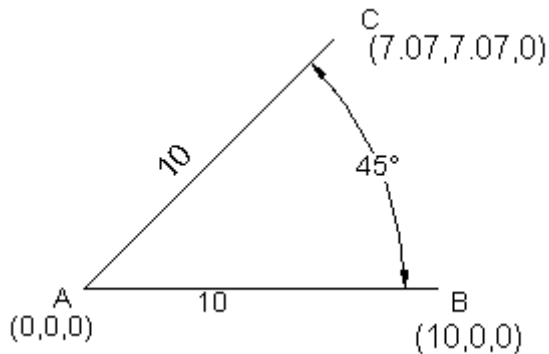
For relative coordinate input, there are five different entry formats available:

**Directed** – Distance<<

Directed relative coordinates uses an input distance and the current location of the cursor relative to the last entered point to establish direction or angle. The two << symbols that follow distance are the programs indication that cursor location is to be used to establish the angle. Directed relative coordinate input varies from the other forms in that it does not require the @ prefix.

Note: With Ortho mode set to On, this input method provides a very quick and powerful entry method for drawing lines parallel to the axes. IE when used in drawing floor plans

**Polar** - @ Distance < Angle



In the above illustration, we wish to establish Point "C" relative to Point "A". "A-B" lies along the 0o angle or bearing and "A-C" lies on the 45o angle or bearing, a distance of 10 units apart. To draw a line from A to B in relative polar coordinates, we would give the LINE command, then enter (or pick) point A and then specify point C as @10<45. The line will then extend to C, after which we can simply continue

on to points D, E,F..., by entering distance and angle to each in the same fashion. You can alternate between absolute and relative coordinate entry as required.

**Cartesian** - @deltaX,deltaY,deltaZ

In this format, we simply indicate the relative (plus or minus) X,Y and Z distances from the reference point to establish the new point. For example, if we wish to set point A from a relative point B in the above illustration, we would pick point B and then specify A as @-10,0,0, as A is -10 x units from B and at the same Y and Z values.

**Spherical (3D)** - @distance<angle1<angle2

In this format, you specify the location of the second point relative to the first as being the distance **in** the X-Y plane, the angle (**in** the X-Y plane) and the angle **from** the x-y plane. In the example @10<90,5 distance in the X-Y plane is 10 units, the angle (measured from the positive X axis in the X-Y plane is 90o and angle measured from the X-Y axis to the second point is 5)

**Cylindrical (3D)** - @distance,<angle,Z distance

In this format, you specify the location of the second point relative to the first as being the distance (in the X-Y plane), the angle (in the X-Y plane) and at the relative Z distance value from the first point (IE @10<90,5 where distance is 10 units, angle is 90o and Z distance for point 2 is 5)

## 3-D

### 3-D Viewing

### 3D View Control...



**Command: 3DVIEW**

**Alias 3V**

#### **Located:**

1. **Pull-down Menu: View – 3D View**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the View menu allows you to define the view parameters. .



The program uses as its standard viewing angle, the top view (also called Plan View). At this angle the x-y-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views.

Along with the ability to view the drawing from any vantage point, it is also possible to rotate the view in the X – Y plane. You may want to do this, for instance, when orienting your drawing for printing.

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**See also:**

SETVIEWDIR command

View Manager: Saving / Retrieving Views

Using Drawing Windows

AllenCad is capable of opening as many as **four drawings** at a time. Each drawing is viewed in its own window. Further, you can open as many as **four viewing windows (viewports) of each drawing**. Windows (See Using Drawing Windows) allow you to see the drawing from different angles, or see different details - at the same time.

A number, ranging from 0 to 3 identifies each window. This number appears immediately behind the filename on the title bar, separated by a vertical line, as in Filename | 2.

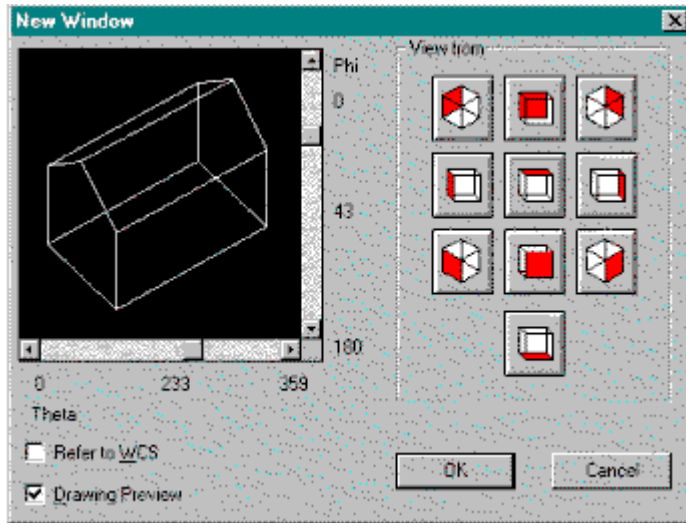
The windows can be arranged in tiles, cascades, or enlarged to fill the entire drawing area.

When more than one window exists, only one window is active. The active window is identified by the color of the title bar. You pick another window to make it the active window; click on any part of the window with the cursor.

AllenCad's standard viewing angle is the top view (also called the **plan view**). At this angle, the X, Y-plane is viewed straight down the Z-axis. The drawing or construction plane is parallel to the screen.

### Creating a New Viewport

Once a drawing is open, you create additional viewports with the **WOpen** command (Window > Open New Viewport). Before the program opens a new window, the New Window dialog box asks for the view direction. *The AutoCAD equivalent to this command is the MVIEW command.*



Select a standard view from the View from buttons, or specify a custom view with the Theta and Phi sliders. (The view you select can be changed at any time.) Click on OK. The new window is created in standard size and arranged in the manner proscribed with the existing windows (tiled, cascaded, etc.).

1. **Theta / Phi** - The preview window, showing a house, lets you interactively change the 3D-view direction. Move the Theta and Phi sliders and watch the house rotate in three dimensions. When satisfied with the viewpoint, click OK.
2. **View From** - Select a standard view from the View From buttons.
3. **Plan View** - To switch quickly to a plan view, use the PlanView command (View > Plan View to Construction Plan).
4. **Changing the View** - The **3dView** command (View > 3D View) displays the View Direction dialog box, which is the same as the New Window dialog box discussed above. It allows you to look at a drawing from any point in space that you wish to define. You must define the view direction, as described above. To change the 3D viewpoint at the command line, such as in macros or FLISP routines, use the **SetViewDir** command.  
     > SETVIEWDIR  
     Viewpoint (X,Y,Z): Rotate
5. The **Rotate** option allows you to specify a new view direction, based on two angles entered as real numbers, as shown by the following prompts:  
     Angle theta (rotation angle in x-y-plane) <0.0>:  
     Angle phi (rotation angle to x-y-plane) <0.0>:  
     The Angle theta is measured from the X-axis in the XY-plane. The Angle phi is specified up and down from the XY-plane.

**Note:** The SetViewDir command cannot be used in paper space.

### Changing the Active Window

Once you have more than one window open, there are two ways you can change the active window:

1. Click any part of a window to make it the active window.
2. Click the Window menu, which lists the names of all open windows. The active window is marked by a checkmark in front of its name. You may choose one of the listed windows as the active window by clicking on that view, or by entering the number assigned the view.

### Copying a Window

The **QWOpen** command (Window > Copy Viewport) opens a new window and allows you to make a quick copy of the active window (Copy option) or of the plan view (Plan option).

> qwopen

New Window: Copy

### Closing a Window

The **WClose** command (Copy > Close Current Viewport) closes the current viewport. Or, you may use the standard Windows methods of closing a window: type Ctrl+F4 or click the x in the upper right corner of the window.

## 3-Dimensional Drawing

### 3D View Control...



**Command: 3DVIEW**

**Alias 3V**

#### Located:

1. **Pull-down Menu: View – 3D View**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the View menu allows you to define the view parameters. .



The program uses as its standard viewing angle, the top view (also called Plan View). At this angle the x-y-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views.

Along with the ability to view the drawing from any vantage point, it is also possible to rotate the view in the X – Y plane. You may want to do this, for instance, when orienting your drawing for printing.

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**See also:**

SETVIEWDIR command

View Manager: Saving / Retrieving Views

## Object Thickness

### THICKNESS

By applying the thickness property to entities, you create a three-dimensional effect called "2½D." The extrusion of entities creates the illusion of depth. Thickness is measured in the Z-direction.

You can give thickness to the following entities:

- Arcs
- Circles
- Lines
- 2D polylines
- 2D faces

If any of these entities are drawn with a non-continuous linetype pattern (such as dash-dot) or if they are a wide polylines, then the pattern and width is extruded.

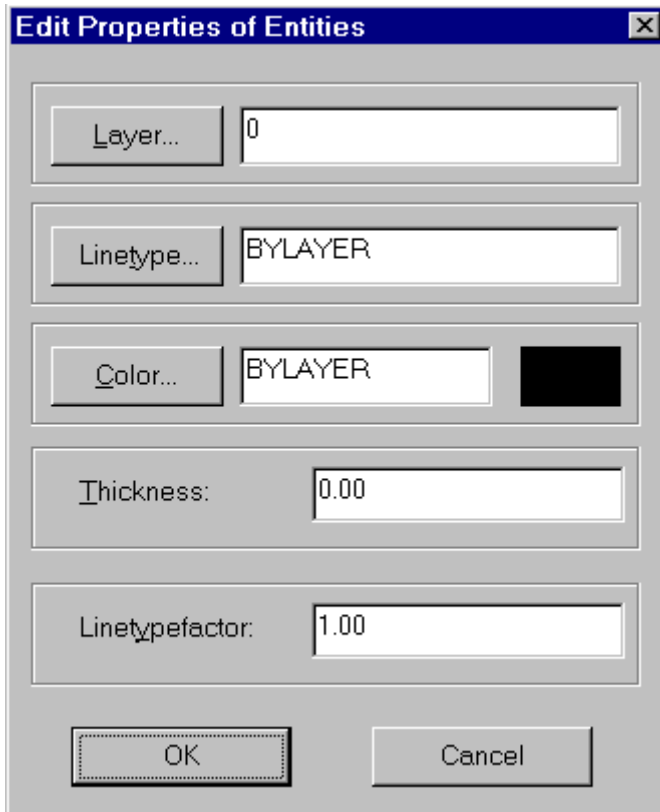
Thickness can be assigned to other drawing objects, such as 3D polylines, 3D faces, text, attributes, attribute definitions, block insertions, dimensions, and hatches, but AllenCad does not display the extrusion. To set a preference for subsequent drawing, set the Thickness system variable. Type the system variable name THICKNESS and enter a numeric value. To turn off thickness for further drawing, set the value to zero. **To alter the thickness** of drawing object, use the **PropEdit** command.

As an example, we will draw a simple rectangle:

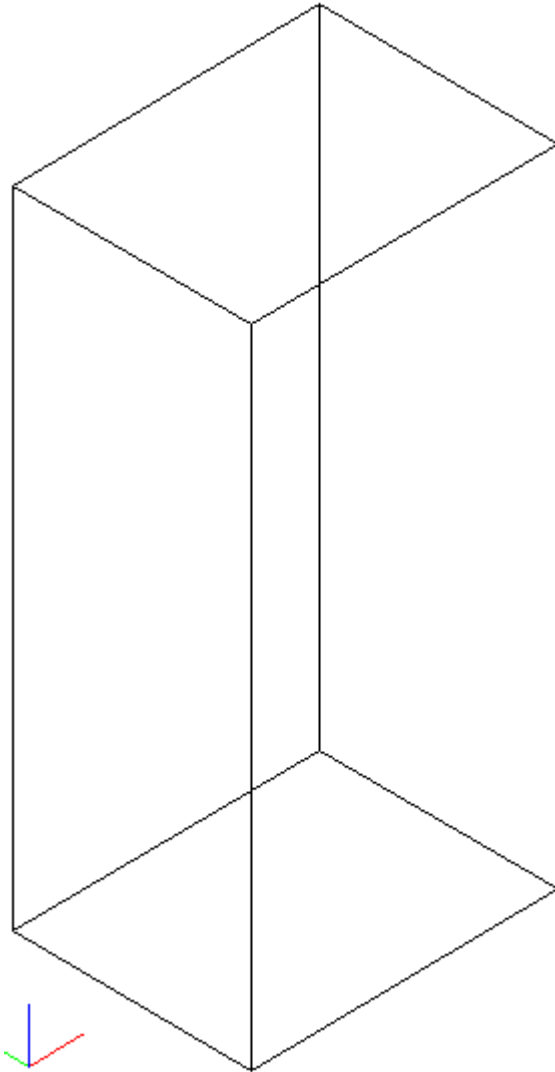


Next, we will use the PROPEDIT command and select the rectangle, at which point we will see the following Dialog:





By entering a thickness value (positive or negative), the rectangle will be "extruded" in either a positive or negative "Z" direction from the elevation at which the rectangle is drawn. In order to see the effects of this extrusion, we will need to select one of the 3dView options offered by selecting 3dView on the **View** pull-down menu. Our drawing will then appear as pictured below.



### **Applying Object Selection and Object Snap to Extruded Entities**

When selecting or snapping to entities with an extrusion thickness, keep these two rules in mind:

1. **Object Selection:** When requested to select objects, AllenCad recognizes extrusion lines.
2. **Object Snap:** When you snap to extrusion line of entities with thickness, AllenCad finds the appropriate point of the entity's base.

### **Treatment of Extruded Objects in Rendering and Hidden Line Removal**

Objects with thickness are recognized by the rendering and hidden-line removal commands: ARENDER, RENDER, HIDE and FHIDE.

### **Elevation**

The System Variable ELEVATION can be set and used to determine the elevation ("Z" value) at which objects will be subsequently drawn at. By default, all objects are drawn at a "Z" value of zero.

ELEVATION simply resets the default value. Using ELEVATION greatly speeds up the drawing exercise where coordinate value input is required, and where a number of elements must be drawn at the same elevation. The operator may simply specify "X" and "Y" coordinate values and have the current default value for "Z" applied automatically.

For Example:

ELEVATION 10 - (Sets default "Z" to 10)

LINE - (Selects Line draw mode)

12,15 - (Begins line at coordinates (12,15,10))

20,13 - (to 20,13,10))

22,15 -(To 22,15,10))

etc.

## Drawing 3D Polylines



**POLY3D** -3D Polylines are drawing elements created from straight-line segments. It is possible to define 3D coordinates for each vertex. This can be done by picking a point that has x, y and z coordinate values, or else by entering coordinates directly at the command line in the format x,y,z.

Unlike 2D Polylines, 3 D polylines cannot include arcs and only support a zero width. Like 2D polylines, however, the 3D polyline is treated as a single object by the program.

To run the 3D polyline command, simply enter **POLY3D** at the command line, or from the Draw pull-down menu, select 3D Entities - 3D Polyline.

The command entry sequence is as per the following example:

POLY3D

First point. P1 (pick point 1 with the mouse)

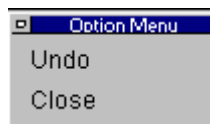
Next point: P2 (here we could enter 2,3,4 (for x=2, y=3, z=4)

Next point: P3

Last point: P4

Exit command: RETURN

While entering points, you will see the following option dialog displayed:



Pressing **Undo** will eliminate the last point that you picked and allow you to re-enter it correctly. Repeating the Undo will eliminate points on the 3D polyline in the reverse order that they were entered in. Pressing **Close** will cause the 3D polyline to close back to the start (first) point from the last point entered.

The **Undo** function can be repeated as many times as necessary until the Undo List is empty. Each Undo step will erase the last drawn 3D polygon element.

**See Also:** Drawing (2D) Polylines

## Drawing 3D Faces



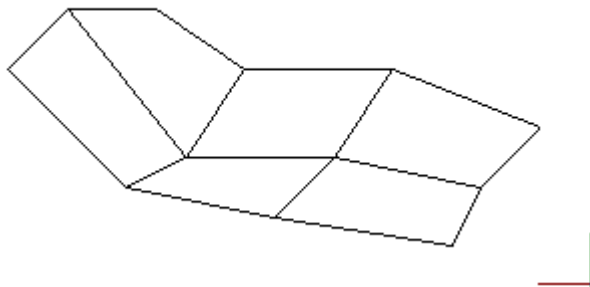
**3DFACE** - This command is executed in a similar fashion to the **2DFACE** command, except that points selected may have different Z- values, and a clockwise or anti-clockwise pick order is required, instead of the left-right, left-right order of the **2DFACE** command. Another difference is that the faces are not filled in with color, but instead are shown as wire-frame models, though each face is in fact a solid surface. This becomes apparent when parts of the surface become hidden from view. These models are often used afterwards for rendering.

## Creating a 3D Mesh

**Command: 3DMESH** - The **3DMESH** command creates polygon meshes. Usually it is used to generate irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

As each vertex must be input one at a time, it is most common to rely on macros or LISP routines to read the vertex coordinates from a file and to construct the mesh automatically. We will describe the input for a small mesh here, just to demonstrate the concepts.

Technically speaking, a 3D mesh is a wire frame rectilinear blanket composed of M column lines by N row lines passing through a matrix of M x N three-dimensional points in space. M and N are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh. In more simple terms, you can think of a 3Dmesh as being like a length of wire fencing that is defined by the intersections of the wires. The mesh can be in any orientation and can be joined to itself to create any 3-dimensional geometric shape.



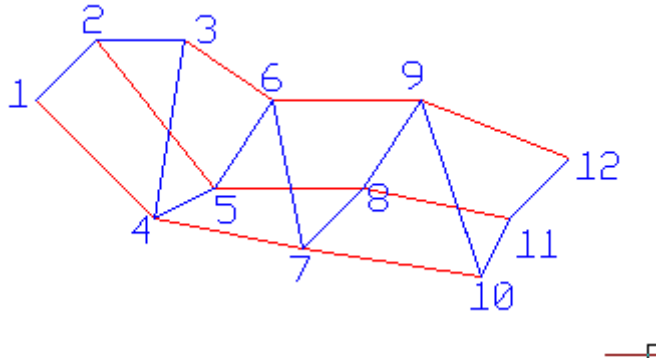
### To Create a Polygon Mesh

First you are prompted to specify the mesh M size and the Mesh N size. This is simply the number of wire intersections wide and long.

Then the program continues to ask for corners or vertices, until you have defined all of the vertices in the one (M) direction and the other (N) direction. Remember that to complete the mesh, you will need to enter M x N vertices. You are prompted for each vertex in order, starting with vertex (0,0), continued with the second vertex in the first column (0,1), and so on. Once all vertices of a

column are specified, the first vertex of the next column is requested. The blue line traces the input order in the example further below.

**Note:** During vertex input, the mesh does not appear onscreen, but is instead being constructed in memory. Terminating the command places the mesh onscreen.

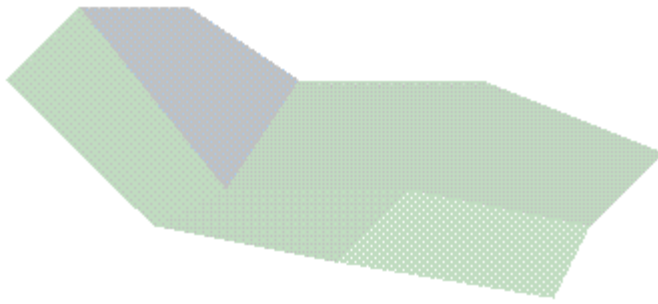


For example, vertex (1,0) is the first vertex (#4) in the second column. In the figure below, the last vertex (#12) is (3,2).

**Note:** Meshes can be open or closed depending on whether the mesh joins in either the M or N direction, or both. A donut mesh would be an example of a mesh closed in both directions. Input for the above mesh would be as follows:

```
> 3DMESH
M-value of the mesh (between 2 and 256): 4
N-value of the mesh (between 2 and 256): 3
Vertex (0,0): 110,170,4
Vertex (0,1): 130,190,4
Vertex (0,2): 160,190,6
Vertex (1,0): 150,130,0
Vertex (1,1): 170,140,0
Vertex (1,2): 190,170,2
Vertex (2,0): 200,120,0
Vertex (2,1): 220,140,-2
Vertex (2,2): 240,170,0
Vertex (3,0): 260,110,0
Vertex (3,1): 270,130,0
Vertex (3,2): 290,150,0
```

Like the section of wire fence, the mesh consists of a "wire" grid, which can be easily seen through. If instead, you want a solid surface in each gridded polygon, you will want to use the 3Dface command, instead. Even so, 3D meshes can be easily rendered as shown in the illustration below, that was simply created using the ARENDER command.



## Creating Tabulated (Extruded) Surfaces

Earlier, we described how to use thickness to extrude surfaces. Using this approach, applying thickness to a circle, for example, will always produce a round cylinder, as the extrusion takes place at right angles to the plane in which the original object lay. Often, however, we want to extrude an object in some other direction and this is when we use the TABSURF (Tabulated Surface) command.

Using this command, we can extrude an entity (in this example we will use an arc) in a direction and for a distance, as represented by the endpoints of a linear feature such a line or open polyline. The linear feature selected, therefore establishes the straight line "vector" in which the extrusion occurs.

For this example, we will draw an arc with a radius of 30 with center at a point 260,140,0. We will also draw a line from the center of that arc to a point 230,120,30. At this point when viewed from above (the default view), our objects look like:



Next, we issue the TABSURF command:

```
> TABSURF
```

Select path curve to revolve: (We pick the arc)

Select direction vector (line or open polyline): (we pick the line)

After the command is completed, we have an extruded arc that from above looks like:



Selecting another viewpoint, our extruded entity appears as:



From the latter view, particularly, the extrusion can be seen to have followed the direction and to the extent as defined by the endpoints of the line.

### Creating a Ruled Surface

A Ruled Surface, created using the RULESURF command, simply creates a surface between two objects. In this exercise, we will draw an arc with center at 260,140,0 and radius of 30, as well as a line from 240,130,10 to 260,160,15. After drawing these two entities our view of them from above looks like:



Next, I will issue the RULESURF command.

> RULESURF

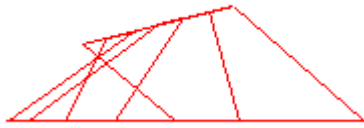
Select first defining curve of ruled surface: (I pick the curve at the lower end)

Select second defining curve of ruled surface: (I pick the line at the lower end)

The surface is drawn and from above it now looks like:



Viewed from a side, this looks like:



Here. We see the arc as a straight line at the base of the extruded surface and the line (with different endpoint elevations as the top of the surface).

Note however that where the entities are picked is very critical to the surface created. In the above example, we picked the visually "lower" ends of the arc and line. If, however, we were to pick the lower end of the arc and the upper end of the line, we would have a surface that from above looks like:



And from the side looks like:

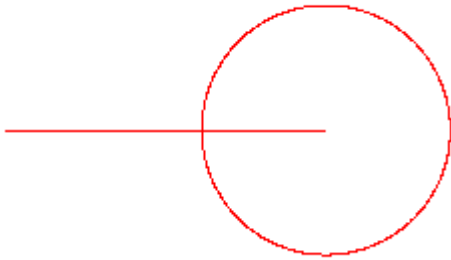


## Creating a Surface of Revolution

We can also create surfaces using the REVSURF command that rotate an entity around an axis, thereby defining the new surface. In this exercise, we will use REVSURF to create a ring.

To begin with, we draw a line with a circle centered at one end. The other end of the line will be the center of the revolution, through which we will later draw our axis. At this point, our drawing from above looks like:





Next, we want to rotate the circle 3-dimensionally by 90° so that the circle will lie in the X-Z plane and look exactly like this when we look at it from a "frontal" viewpoint.

> C:3DROT

Select objects: (We pick the circle)

1 selected.

Select objects: (we want to rotate only one object so we press Enter)

\*\*\* 1 selected. \*\*\*

First point of rotational axis <0,0,0>:endp (we indicate an "endpoint" snap)  
of object: (and pick the line near the circle center)

Second point of the rotational axis <0,0,1>:nea (we choose a "Near" snap)  
of object: (and pick the line again at the other end)

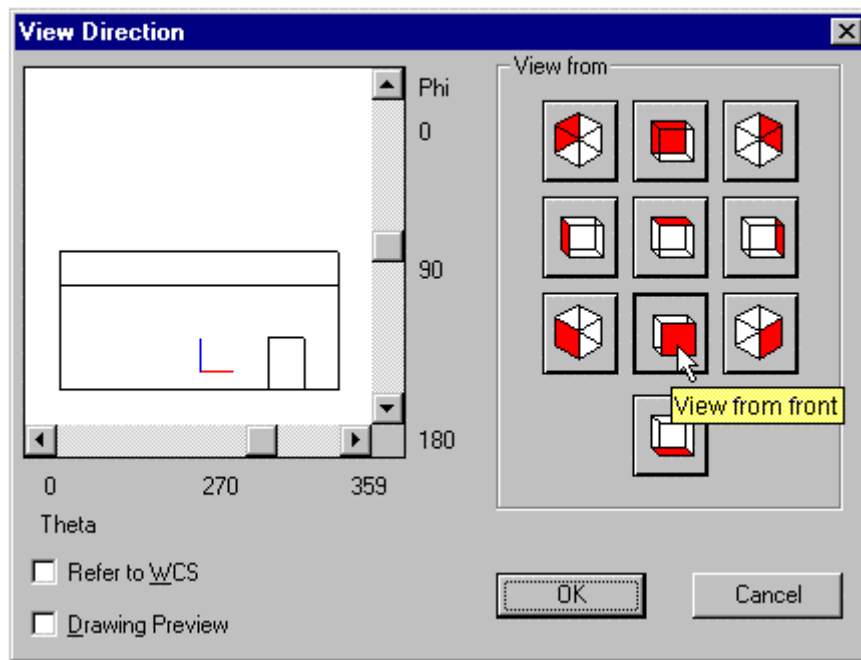
Rotation angle: 90 (We indicate a 90° rotation)

Still viewed from above, our drawing at this point looks like:

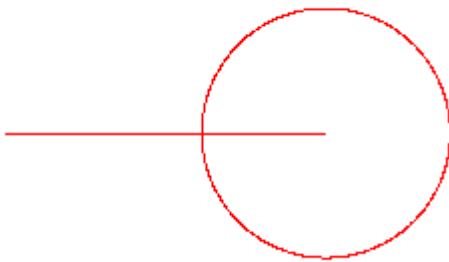


Next, we change our viewpoint to frontal, using the 3Dview Command and we see that our drawing looks as it used to before, with the "above" viewpoint.

>3DVIEW



and:



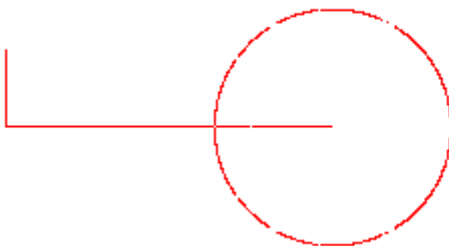
Next, we will draw the axis of rotation as a line from the end of the left end of the radius line (295,175,0) to (295,175,20):

> LINE

From point: 295,175,0

To point: 295,175,20

Now, we are ready to use the REVSURF command and our drawing looks like:



> REVSURF

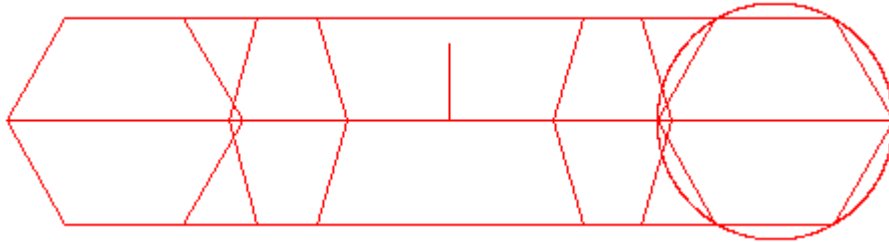
Select path curve to revolve: (We pick the circle)

Select axis of revolution: (We pick the axis line)

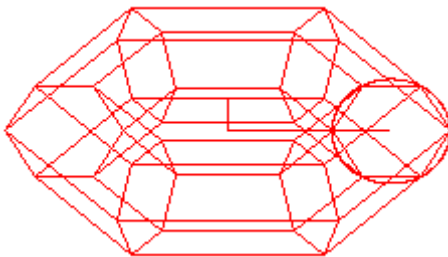
Start angle <0>: (We accept the default starting angle)

Included angle to revolve entity (+ccw, -cw) <360>: (and accept the default end angle)

At this point, our drawing looks like:



When viewed from an oblique angle, however, it looks like:

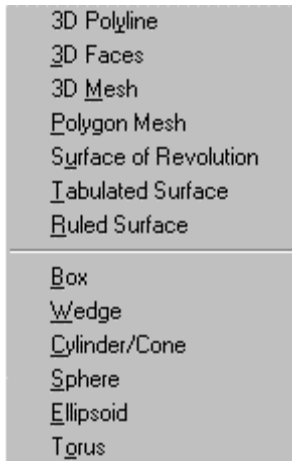


After issuing the ARENDER command to create a rendered version, we have:



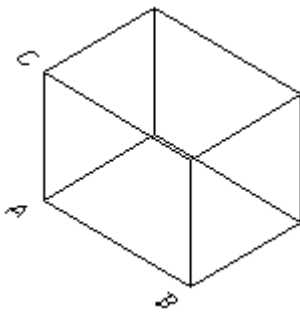
### 3-D Entities

Selecting 3-D from the Draw pull-down menu opens a secondary menu, shown below, that offers the opportunity to draw 3 dimensional entities directly.



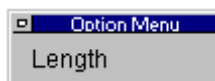
## Drawing a 3D Box

The command 3DBOX facilitates the simple creation of a 3 dimensional box. The command can also be accessed from the Draw - 3D pull-down menu. The box drawn will be parallel to the x, y and z axes, so you may need to rotate the box later, if required.

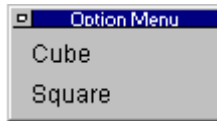


Upon executing the command, you are asked to input the first corner ("A" in the diagram). As with the input of all location definition, you may enter coordinate values in the form x,y,z or you may pick an existing reference point in your drawing. Unless the point you want has a z or height/depth value of zero (default value for 2D objects), be sure that the point you pick has the correct z value.

Once the first corner has been established you will be presented with an option dialog offering you the opportunity to input a length to establish the second corner ("B" in the diagram below). The z value (height) of the second point is the same as the first and the first side is parallel to the x axis (Y values of A and B are equal).

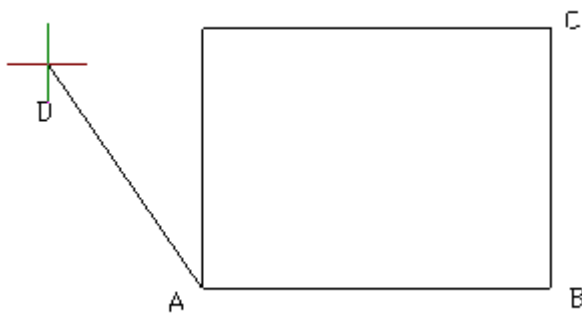


You will be prompted for width if you chose the **length** option and presented with the dialog below:



Selecting **cube** indicates that the x, y and z dimensions are equal to A-B and the cube is completed. Choosing **square** indicates that the x and y values (only) are equal to A-B and the square is drawn with a following prompt for height input, that completes the box.

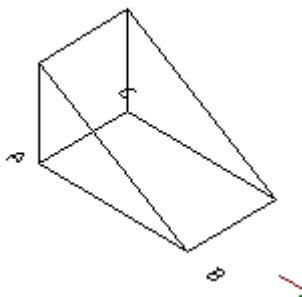
If however, you were to ignore the Length option and input a point with a different y value, after setting the first corner, the second point (picked point "C") would be regarded as the diagonally opposite corner of a rectangle, which would then be drawn. A prompt for height would follow.



For height, an absolute value can be input. Picking a point "D" , however, will set height as being the 3 dimensional distance between points "A" and "D".

### Drawing a 3D Wedge

The command 3DWEDGE permits the simple creation of a 3-dimensional wedge. This command can also be entered from the Draw - 3D pull-down menu.



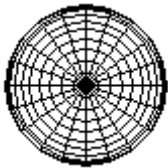
### Drawing a 3D Cylinder or Cone

The command 3DCONE creates a 3D cone, cylinder, prism, or pyramid. This command can also be executed from the Draw - 3D pull-down menu. Object creation is determined by user input. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.



### Drawing a 3D Sphere

The command 3DSPHER allows creation of a 3D sphere (ball). This command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

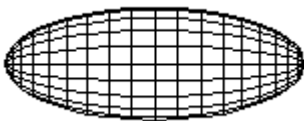


#### 3D SPHERE (BALL)

Command input: after issuing the command, you are prompted to input the Center of the sphere (x,y,z or pick). Next, you are prompted for radius (enter value or pick to establish distance from center). Lastly, you are prompted to input discretization values for Alpha and Theta (values of 20 for both issued in the above example). The sphere is then drawn by the program.

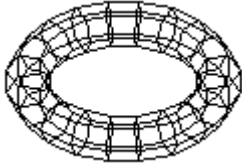
### Drawing a 3D Ellipsoid

The command 3DELLIPS (Draw>3D>Ellipsoid) allows creation of a 3D ellipsoid object. This command may also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.



## Drawing a 3D Torus

The command 3DTORUS allows creation of a 3D torus (3D Donut). The command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.



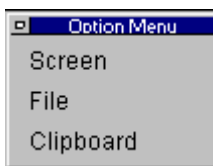
## Rendering

### Rendering 3-Dimensional Drawings

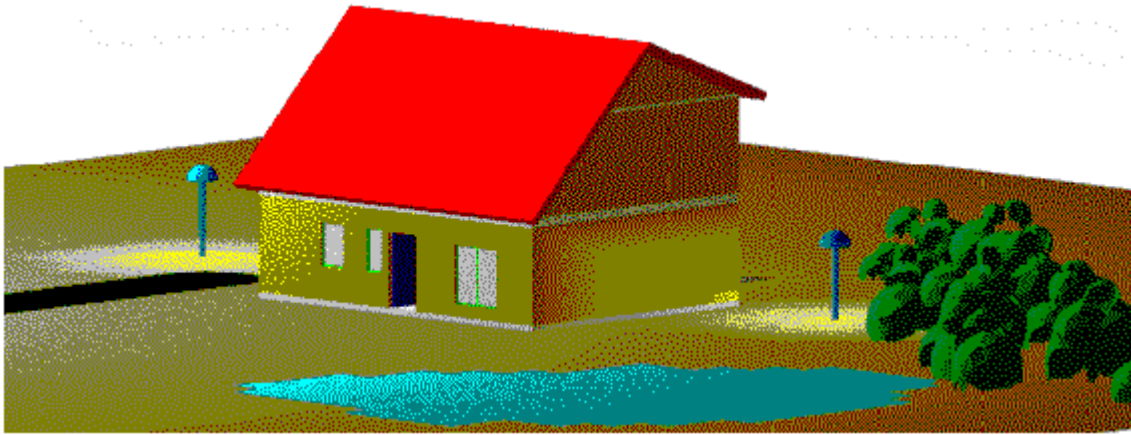
The Render command (View > Render > Render) creates an image of your 3D drawing, called a rendering. The rendering is a realistic illustration of the model. To increase the realism, the rendering can be made to perform smooth shading and evaluate the lights set into the drawing.

You use rendering to visualize and present your 3D geometry, as it would appear as if it has been constructed or built. Note that AllenCad also includes an animated rendering function as detailed below this render description.

To render the current viewport of a drawing, start the Render command. The command prompts you to select where the rendered view should be output.



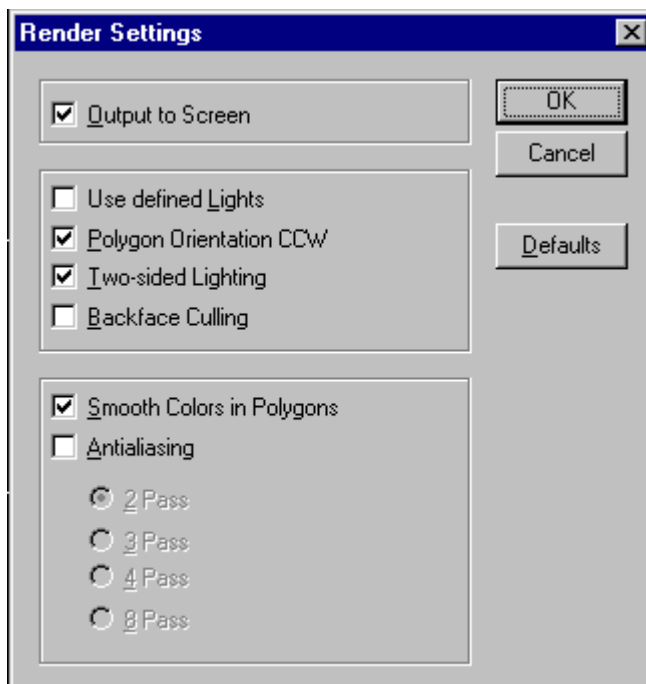
- **Screen:** the rendering is output to the current viewport.
- **File:** the rendering is saved to a .bmp (Windows bitmap) file on disk.
- **Clipboard:** the rendering is copied to the Windows Clipboard in bitmap format; it can be pasted into another Windows application with the Ctrl+V (Edit > Paste) command.



The rendering is based on the preferences set with the RConfig command. The Light command allows you to place lights in the drawing and define their type and settings. The LightEdit command is used to modify the settings of light sources.

## RENDER PARAMETERS

The **RConfig** command (View > Render > Render Parameters) displays the Render Settings dialog box, which lets you control the quality of rendering.



The **Defaults** button resets the preferences to default values (as shown in the illustration above). This is convenient, because a number of settings can influence the performance of render computations dramatically.

1. The **Output to Screen** option controls whether the rendered polygons are output on the screen (to the front buffer) during computations or are written into a display buffer before presenting the rendered image. If this option is not checked, the rendered image is not



displayed before the entire rendered image is computed.

2. The **Use defined Lights** option allows you to activate or deactivate the use of light sources. When disabled, the program generates a simple shading of the 3D model; otherwise, the inserted lights are used for the rendering.

The three other options allow expert users who are familiar with advanced rendering techniques to save time when careful drawing methods have been used during the creation of a 3D model:

3. · Polygon Orientation CCW
4. · Two-sided Lighting
5. · Backface Culling

It is recommended that you use the default values if you are not familiar with these terms.

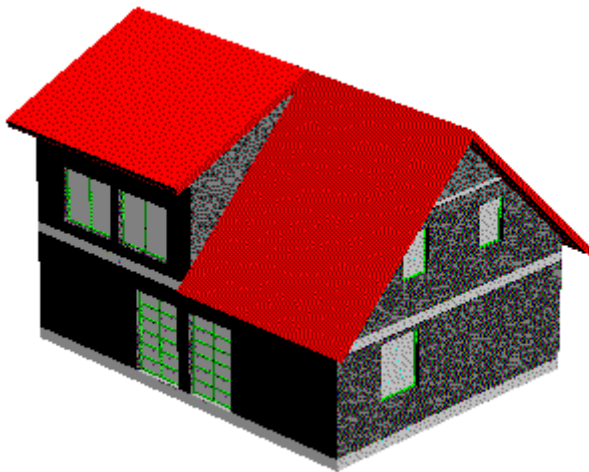
The **Smooth Colors in Polygons** option specifies whether smoothing colors are computed upon individual polygon surfaces. Applying an adequate light environment allows smooth coloring on individual surfaces.

The **Anti-Aliasing** option allows you to control the process of smoothing jagged edges caused by aliasing. Aliasing describes certain display effects, like the staircase effect seen on diagonal lines displayed in a drawing. Anti-aliasing reduces those artifacts. Turn on anti-aliasing for high-quality rendered images.

To improve the rendering significantly you can choose between 2, 3, 4 or 8 passes with the anti-aliasing algorithm. Because multiple passes dramatically increase the rendering time, anti-aliasing should be applied only for high-quality image output (usually when rendering to a file).

## AutoRender

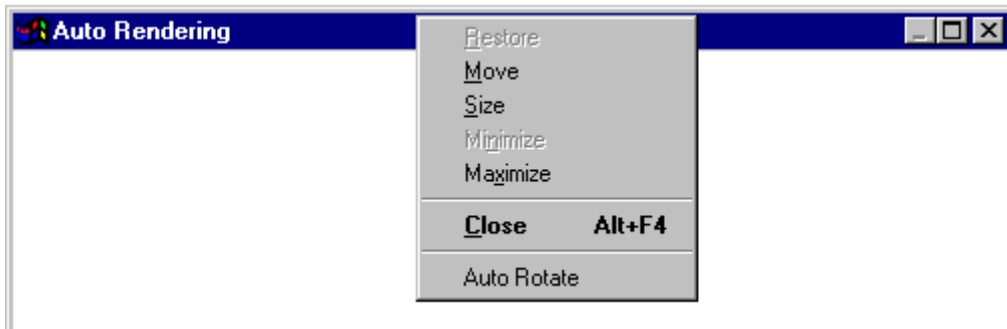
The Arender command (View > AutoRender) displays a rendering of the current drawing in a separate pop-up window.



You can rotate the rendering by situating your pointer inside the Arender window, and holding

down the left mouse button. While the mouse button is depressed, the icon will change to a small hand, allowing you to rotate the image about the screen.

Zooming and panning for Arender is accomplished by holding down the right mouse button. While depressed, you can slide the pointer up or down to zoom and pan the image.



To auto-rotate (spin) the rendering image, right click on the blue command line area and toggle the auto-rotate setting.

For the control of lines and hatches with ARENDER, see ARLINEDISP.

## Render Lights

The Light command (View > Render > Insert Light) allows you to insert lights in the drawing. The lighting environment in a 3D model is set up with one of more light sources. When no lights are explicitly present, AllenCad assume ambient light for the light source.

### Light Types

The render program recognizes four types of light sources:

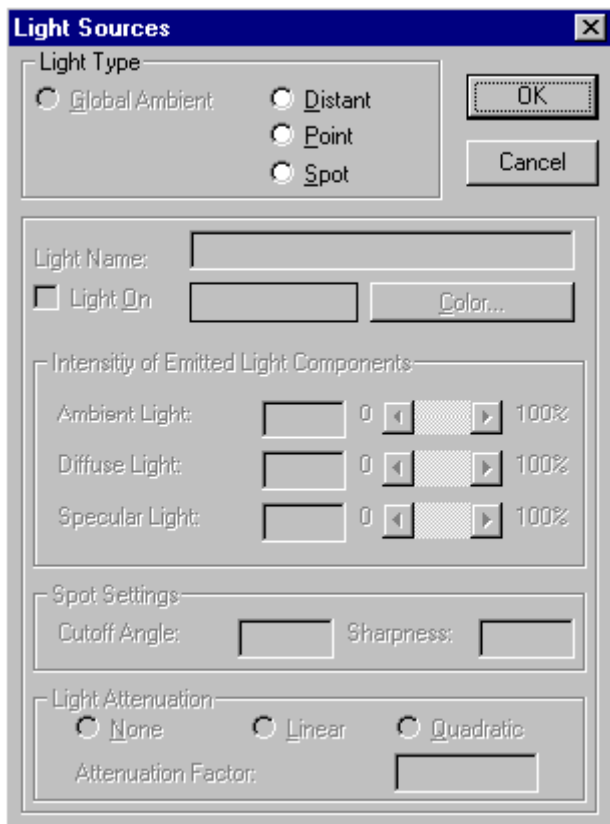
1. An **ambient** light is not really a light source. It is an overall light source that occurs everywhere equally in the scene. The intensity value of ambient light defines an overall value of lightness for the total 3D model. The value increases or decreases the lightness of a rendering.
2. A **linear** light radiates light along a line, similar to a fluorescent lamp. It has a position, intensity, and a color. It is used to simulate sunlight in the model.
3. A **point** light radiates light in all directions. This is similar to a light bulb or an incandescent lamp. It has a position, intensity, and a color.
4. A **spotlight** radiates in a specific direction. It is similar to a stage spotlight. It has a position, an intensity, a color, a direction, and a cone angle.

### Inserting Light Sources

Ambient light is available in all renderings, but the other sources must be placed explicitly in the 3D model. The Light command insert light sources into the drawing.

Although you can insert as many light sources as you want, you can have only eight light sources turned on.

The **Light** command displays the Light Sources dialog box, which allows you to you to define parameters for the light, among them the color and the intensity of the light source.



Lights sources are represented in the drawing as block objects. A light insertion is always placed on a layer with the name "RENDER\_LIGHTS."

### Editing Light Parameters

You modify the settings of an inserted light source later with the LightEdit command (View > Render > Edit Light Parameters). This command requests you to select a light, represented by a block insertion. The Light Sources dialog box then opens, allowing you to alter the current preferences for the chosen light.

### Relocating and Removing Light Sources

To change the location of a light or the lights direction, use the standard modify commands Move and Rotate. To remove a light source from the drawing, use the Delete command.

### Light Sources in Detail

The Light Preferences dialog box is used for all four types of light sources. Depending on the light type, some areas of the dialog box may be grayed out because those parameters do not apply to the specified light type. The following figure shows the properties and intensity values of a point light.

### Ambient Light

Ambient light is not really a light but defines an overall value of lightness for the total 3D model. Ambient light surrounds the entire model to be rendered. It increases or decreases the lightness of a rendering. With ambient light, you can avoid the problem of having the entire model rendered too darkly.

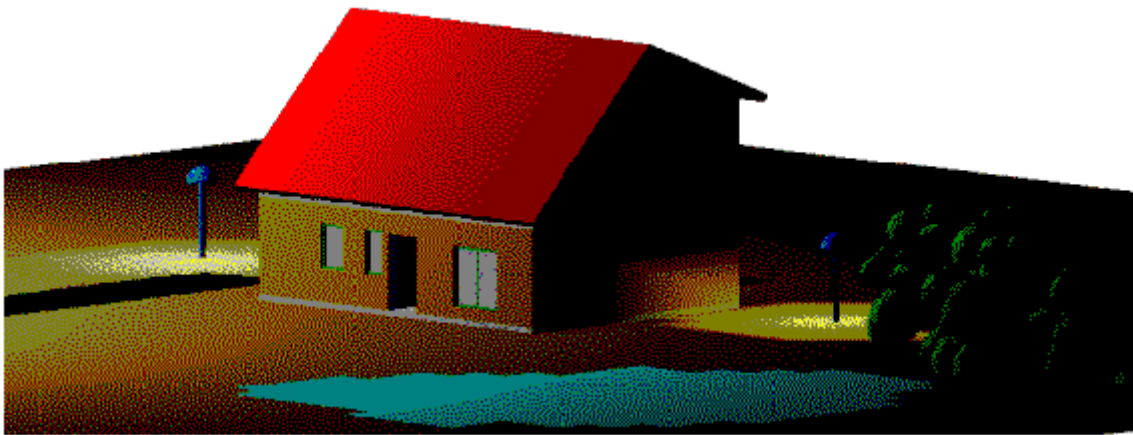
Ambient light has no direction, which means that all surface polygons in the drawing are illuminated uniformly by this type of light (independent of the view direction). In other words, you can understand it as background lightness.

Ambient light is not represented by a light block insertion in the drawing, like the other light types. For that reason it is not covered by the LIGHT command. To modify the value for the intensity of ambient light use the LightEdit command with the Ambient option (View > Render > Ambient Light) to increase or decrease the lightness of a rendering.

The default value for the intensity of ambient light in a new drawing is 20 percent. Increasing the intensity of ambient light tends to saturate the image and give it a washed-out appearance.

The default color of ambient light is white.

Ambient light may also be turned off to simulate absolute darkness. This corresponds to an intensity of 0%. Only other light sources in the drawing will then make drawing objects visible. Turning ambient light off can be utilized to generate special effects when rendering the model.



Ambient light turned off.

### **Distant Light**

Distant light (also called linear or parallel light) radiates light along a line, similar to a fluorescent light bulb. In other words, parallel light emits uniform parallel light rays in one direction only.

Commonly, a distant light is used to model sunlight. It has a position, intensity, and a color. The intensity of the light, however, does not diminish with distance.

Light rays extend infinitely on either side of the insertion point of this type of light source.

### **Point Light**

A point light radiates light rays in all directions from a single point, its insertion point in the model. This is similar to an incandescent light bulb. It has a position, intensity, and a color. The intensity of a point light diminishes with distance.

You set the manner by which the light attenuates with distance (none, linear, or square) and the

factor for distance falloff in the **Light Attenuation** section of the dialog box in.

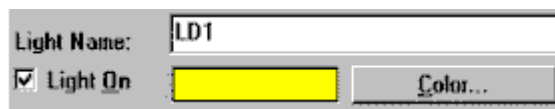
### Spot Light

Spotlights have light rays emanating in a specific direction. This type of light source in a model is similar to a stage spotlight. It has a position, intensity, and a color. In addition, the spotlight has a direction and a cone angle.

You determine the light attenuation (falloff with distance) for spotlights. To determine the direction of a spotlight, you are requested to specify the location point of the light source and the target point of the spotlight when inserting it into the drawing.

### Common Light Settings

Light source insertions have some settings in common, and some settings specific to particular light types. This section describes the in-common settings.



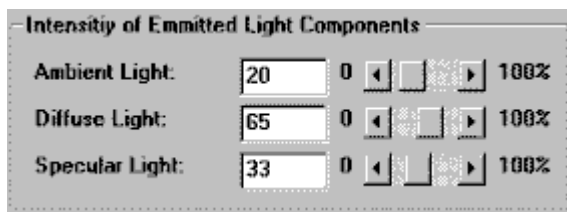
In the Light Name text edit box, you specify a name of a new light source or alter the name of an existing named light insertion. Give each light a unique name. You refer to the light name with the LightEdit command.

The setting of Light On determines if the light is evaluated during rendering. Turning off a light means that the render program does not use the light to illuminate the model. The light settings are retained when turning the light status back on.

Clicking the Color button display in the standard Windows Color dialog box, allows you to define a color for the current light source. The default color for a new light source is white. Note that dark colors affect the light's brightness.

### Intensity Components of Emitted Light

This section of the Light Preferences dialog box allows you to set the intensity of the emitted light. Three components of emitted light are specified in this section: ambient light factor, diffuse light factor, and specular light factor.



1. The **Ambient Light** component is the light that has been scattered so much by the environment that its direction is impossible to determine. The light seems to come from all directions.
2. **Diffuse Light** comes from one direction. It is brighter if it comes squarely down on a surface than if it barely glances off the surface. Once it hits a surface, however, it is scattered equally in all directions, so it appears equally bright, no matter where the eye is located. Any light coming from a particular position or direction probably has a diffuse component.

3. **Specular** Light comes from a particular direction. It tends to bounce off the surface in a preferred direction. You can think of specularly as shininess.

### Light Attenuation

The Light Attenuation section lets you specify distance falloff for point lights and spotlights.



**First**, you determine the manner the light attenuates with distance:

**None:** no attenuation is applied for the point light or spot light.

1. **Linear:** linear falloff is applied, which is commonly used for rendering. The brightness of the light decreases in inverse proportion to the distance from the source.
2. **Quadratic:** square falloff is more realistic, but takes more rendering time. The brightness caused by the light source decreases in inverse proportion to the square of the distance.

**Second**, if you have chosen linear or square falloff, you can specify an **Attenuation Factor** in the edit box. The default value is 1. Light attenuation at a certain location in the model will be:

- Linear falloff: The inverse of the product of distance and attenuation factor;
- Square falloff: The inverse of the product of square of the distance and attenuation factor.

### Specific Spotlight Settings

The **Spot Settings** parameters are set only for spotlights:



1. The **Cutoff Angle** value specifies the angle for the light cone. That is the angle between the line from the location point to the target point (determined during insertion) and the cone edge. Valid values lie in the range of 0 through 90.
2. The **Sharpness** value (or spot exponent) determines the attenuation of light sharpness from the spotlight's target line to the edge of the spotlight cone. Valid values lie between 0 and 128:
3. **Minimum value, 0:** a sharp accentuated light cone.
4. **Any other value:** the brightness is highest at the target line of the spotlight and decreases to the edge of the cone (by the exponent specified).
5. **Maximum value, 128:** the smoothest integration of a spotlight into the environment.

## Productivity Tips

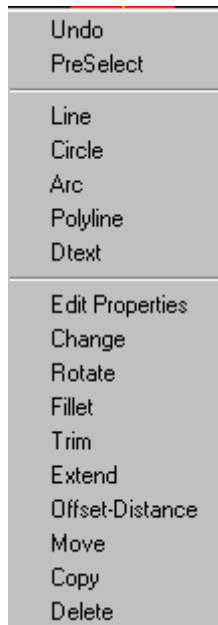
### Wheel Mouse Menu with Dynamic Panning and Zooming

If you have a Microsoft (or compatible) wheelmouse, you have the ability to zoom in and out on the drawing simply by turning the mouse wheel. With the wheel, you can get the exact amount of zooming that you need.

Zooming is based on the current center point in your display, so you may wish to pan so that the area of interest is at the center of your screen before zooming in or out using the mouse wheel.

Using the Alt key in combination with the left mouse button, you also have access to Dynamic Panning, without the need to issue a DPAN command.

The wheelmouse also provides an additional productivity feature – the context menu. By pressing down on the mouse wheel, you call a special drawing and edit menu right onto the screen, saving you the bother of finding the command icons or selections on the desktop or in the menus. This menu contains most of the more commonly used drawing, edit and configuration commands, as can be seen in the following graphic:



To exchange commands on the context menu, you can edit the AllenCad.key file (in the AllenCad \Cfg folder) with a text editor and change the command names listed in the Context Menu section to suit your individual needs. Before editing the key file, be sure, however, to save a backup copy of the file to assist in error recovery, should this be needed.

### Command VNOTE



**Command: VNOTE (VoiceNOTE)**

**Alias VN**

#### **Located:**

- 1. Draw Menu > MarkUp > Insert/Edit VoiceNote**
- 2. Left Side Bar**
- 3. Palette Draw**
- 4. Enter Command or Command Alias Directly at the Command Line**

VoiceNOTES may be embedded into a drawing by using a feature that attaches audio and text to a special comment entity.



### Adding VoiceNOTES

VoiceNOTES can be added to a drawing by selecting Insert/Edit VoiceNOTE from the Draw menu. Select a point in the drawing where you would like to place the VNOTE icon. The dialog box Comment... is then displayed allowing you to enter text and/or to record your voice. Once a VNOTE is entered, select the OK button to place your comment in the drawing.



In the dialog above, simply type your text in the window, or use the recorder buttons to record an audio VoiceNOTE.

### Recording Your Voice

Note that to record your voice, you will need a microphone, audio hardware and audio software installed and properly configured. To playback your recording, speakers are also required. To record your voice as a VNOTE, either add or retrieve an existing VoicenNOTE icon after issuing the



VNOTE command. This will open the dialog box

In the dialog box Comment... there are four audio controls: record, play, pause and stop. Selecting the record control will record audio until the stop control is pressed. Each new recording completely removes the previous recording. You may playback a recording by selecting the play control. When recording or playing back audio, the pause and stop controls can be used as needed.

### Retrieving or Changing VoiceNOTE Comments

VoiceNOTES can easily be retrieved for reading, listening or also editing. Firstly choose Insert/Edit an Audio Comment from the menu Tools or issue the VNOTE command in any of the above-mentioned locations.



After selecting a VNOTE icon in the drawing, the dialog box Comment...is display with the attached text and/or audio recording. You can read the text message (if there is one) in the dialog window, or you can click on the play button to play an audio recording. Alternately, the text edit box may now be edited or a new speech recording made that overwrites a previous recording. The text or audio recording is updated when the OK button is selected.

### Removing VoiceNOTES

VoiceNOTES can be removed by selecting Delete from the menu Edit.

Select the VNOTE icon containing the comment to be removed. Removing this icon will remove the attached audio and/or text comments.

#### Notes:

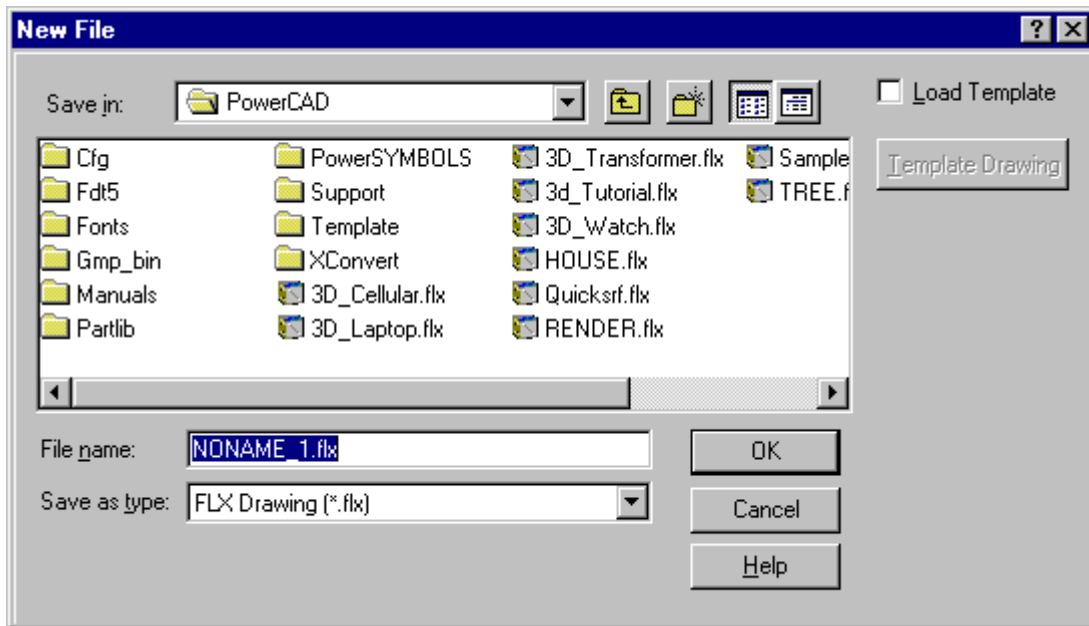
1. VNOTES are stored in Extended Entity Data format and appended to attributes to the fcomment.flx block in the drawing database and externally in the \support directory.
2. VNOTES are primarily for use when the drawing file is loaded in AllenCad. The drawing file can be saved in DWG or DXF format, without losing the recorded information, however this information is not directly addressable from the AutoCAD program. AutoCAD simply ignores their presence in the drawing file.
3. VNOTES stored externally in the \support directory can be exchanged with any other block. Within the drawing file, they are created with the current layer, color and linetype settings. The scale factor of the inserted block is screensize divided by 9.
4. VNOTES are not limited in size, as additional attributes are added for larger sized recordings..

**Tip:** In complex drawings, placing VoiceNOTES on a unique layer can make them easier to locate, especially when the drawing will be shared with mobile AllenCad CE users.

## Template Drawings

If you have drawing standards or common elements that you wish to apply to more than one drawing, it is a very good idea to save all of these settings and elements in a drawing file containing just those elements. By specifying a **template drawing** when you open a new drawing, you will already be setup correctly with dimension variables, text styles, layer settings, title block, etc.

When you open a new drawing file, you are presented with the following dialog box:



Notice the **Load Template** check box in the top right-hand corner as well as the **Template Drawing** button immediately below. These control if a template is used when opening the drawing, and which template, if so.

You may find that the simplest way of using such a drawing would be to save it in your default drawing directory and to call it up as a "Template" whenever you open a new file needing the template's saved characteristics. You may store and use as many template drawing files as you want, in this way.

## Repeating Commands

Often, you find you need to repeat a command. For example, you might be using the Line command repeatedly. To repeat the same command by pull-down menu or keyboard entry gets annoying and inefficient.

You repeat the **last** command by one of three methods:

1. Pressing the Enter key or
2. Clicking the right mouse button or
3. Pressing the Spacebar key.

For example, the first Enter ends the Line command; the second Enter repeats the command:

```
> LINE
From point: P1
To point: P2
To point: P3
To point: [Enter]
> [Enter]
LINE
```

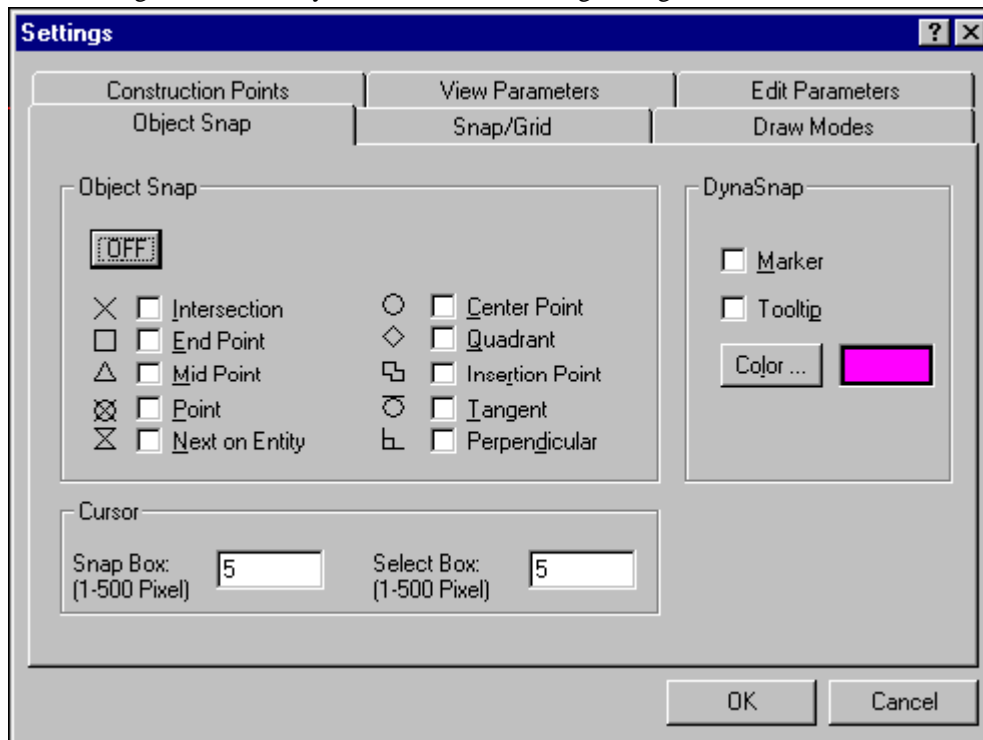
From point: **P4**  
To point: **P5**  
To point: **P6**  
To point: **Close**

**Note:** Some commands, like Point and Trim, automatically repeat themselves.

## Using Object Snaps

As a AllenCad user, you will often want to be able to identify exact points while you are drawing objects and editing them. Imagine having to draw a line from the Center of a circle to the mid-point of a line segment, for instance. To make the job easier, AllenCad provides Object Snaps (Osnaps) that can be switched on and off by using the SETTINGS command or the TOOLS - SETTINGS pull-down menu option.

After issuing the command, you will see the following Dialog box:



In the object snap frame, you will note the different snaps that can be activated by picking on the adjacent boxes. To simplify things, it is recommended that you select only the options that you will likely be using in later operations.

When Osnaps are active, you will notice that a "pickbox" appears on the cursor. When you pick on an object, the closest snap point to the cursor will be selected. If an object is not located within the pickbox, the coordinates of the cursor are recorded. When snaps are located very close to one another, it can be difficult at times to determine which point was selected.

## Manual Mode

If you look again at the Object Snap dialog, you will notice that each command has an underlined letter. These letters call the snap option, which is active only until the next pick has been made. This temporarily over-rides any other snap settings. When you need to be sure that you are getting the correct snap, or when you only occasionally need to use a snap, these options can prove to be very useful.

For instance, if you want to draw the above example of a line from the Center of a circle to the mid point of a line segment, you could enter the LINE command and then any of the following:

- C (set snap for Center of circle)
- X (Pick any point on the circle)
- M (set snap for mid-point)
- X (Pick a point on the line segment near the middle)

At this point, you will have the required line recorded in the drawing.

## Mouse Context

Selecting "**Mouse Context ...On**" located on the Tools pull-down menu allows the operator to make fast modifications to existing entities in the drawing. When switched on, picking an entity with a left-mouse click will bring up an options menu or toolbar entries containing all modifications that can be made to that entity. This saves much time looking for a command that may be out-of-sight on a pull-down menu or un-displayed palette.

Note that with mouse context on, a cursor pick box is not displayed (it would be constantly on, otherwise). Current pick box settings are active, nonetheless, however, so the same object proximity requirements apply, just as if the pick box were visible.

## Using Offset To Advantage



When working with lines, polylines arcs or circles, you will find many instances where parallel entities are required. The Offset command produces these quickly and easily without the need to "draw" the new entity.

## Command Entry Using Palettes

Often, while developing your drawing, you will find yourself in a situation where you are doing a similar function repeatedly (drawing elements, modifying elements, dimensioning, etc.). At such times, it can be very handy to have a palette filled with the appropriate command icons right there on your drawing surface, to speed up your command entry and maintain your focus.



Individual palettes, such as the Edit palette pictured here can be called from the horizontal toolbar icons as pictured below:

Until you become familiar with the icons, you can determine the identity of each palette (and command)

icon by simply holding the cursor over the icon until the tooltip appears with the identification message.

Palettes are scaled to the screen. This means that whether you are zoomed in or out in your drawing, that the palettes retain their manageable and readable size. In addition to the ability to drag and drop the palettes as you move around in your drawing, you also have a "flip" icon on each palette, that flips it to a horizontal or vertical orientation, and then back again, as needed. This ensures that the palette does not interfere with your drawing. When you are finished with them, simply click on the close button on the palette header to remove it from the desktop.

## Automating with Macros

Using macros automates the execution of a repetitive sequence of commands. Users who are familiar with AutoCAD may be familiar with macro-use under the name of Script files. A macro is simply a text file containing a consecutive list of commands and inputs for options, etc. that perform the required function. Once created and saved, a macro can be called and run any number of times, while editing one or more drawing files.

You can create a macro using the Windows notepad program (available by using the File - Resource Manager – Editor command in the pull-down menu. To be recognized as a macro, it is necessary to save the file with an .mcr filename extension.

To run the macro, either:

1. Select File > Resource Manager > Run Macro from the menu, or type Macro at the command prompt or,
2. Select a macro file (extension .mcr) from the file dialog box and confirm the selection by clicking OK. Notice that the macro immediately begins to run.

There are further instructions for creating macros in the Programmer's Guide manual.

## Using Flisp Routines

Like Macros or Scripts, LISP routines perform a series of commands and actions, using a pre-programmed set of instructions. AllenCad uses a proprietary version of common LISP programming code that is written in "FLISP" (Flexible LISP) format. Another common proprietary form of LISP is Autodesk's AutoLISP format. AllenCad contains a built-in FLISP interpreter that will convert AutoLISP code to FLISP format, enabling the user to make use of these programs.

To run FLISP files, use the LOADLISP command (File > Resource Manager > Load LISP File). This allows you to load an FLISP file via a file dialog box.

Detailed information on the built-in FLISP interpreter is found in the Programmer's Manual.

## Drawing Tutorials

### Drawing Checklist

Before we begin a new drawing, there are number of things that we need to consider and

configuration settings that we need to confirm.

1. Program Configuration - Have we properly configured AllenCad with the settings that we will be needing?
2. Drawing Setup Parameters - Have we set the units that we wish to use in the drawing?
3. We also need to plan ahead and consider the elements that we will be using in the drawing and also the environment that we will need to provide, before we get into drawing production:
4. Do we have a template drawing that we can use to set up our new drawing with the text styles, dimension variables, layers, linetypes and title blocks, etc., that we will be using?
5. What coordinate system will we be using and will we be importing other drawing files that are based on a different coordinate system?
6. How do we plan to use drawing layers to assist us and differentiate between the various elements that we will be introducing into the drawing?
7. Will we be using parts raster graphic files or other drawings as external references? If so, we should confirm the location in which these files are stored, as well as their current format is compatible.
8. We should also be clear in our minds as to the goal of the drawing exercise, or the final output format that it needed. Knowing this in advance will greatly assist us as the drawing development progresses.

## Drawing Construction

### Planning

There are often several methods that you can use to produce drawing elements. Some will certainly be faster and simpler than others, and that is why it is a very good idea to plan ahead when you are developing your drawing. In general, the concepts you will want to keep in mind are as follows:

- **Analyse** the known characteristics of the element before attempting to draw it. For example, if it is a line - Where are its endpoints? What is its position relative to other elements in the drawing? At what angle or bearing should it be drawn? The best technique to use in drawing the element will usually be indicated by the elements known characteristics.
- **Avoid** drawing elements more than once, if possible. After analyzing an element's characteristics, check to see if you have other identical elements that can be copied from another location on the drawing. Use the ROTATE command on copied elements as necessary to maximize the use of copied elements.
- **Use** parts and x-references to reduce the number of elements that you need to draw and be sure to use a template drawing to setup your drawing environment and to supply common elements such as title blocks, etc.
- **Set** appropriate object snaps to facilitate and speed up the drawing process.

### Construction Lines

Often while you are drawing, you will need to be able to select a point that has a known relationship to other elements but that has not been defined in your drawing. Before you can use these points to draw from you will first have to define them in the drawing. To do this, you will need to draw construction elements, usually lines or circles, that measure out distances on your drawing and which produce intercepts with other elements that define the points you need. After

they have been used, these construction lines can be erased, as they are no longer needed.

## Creating a Simple Floor Plan

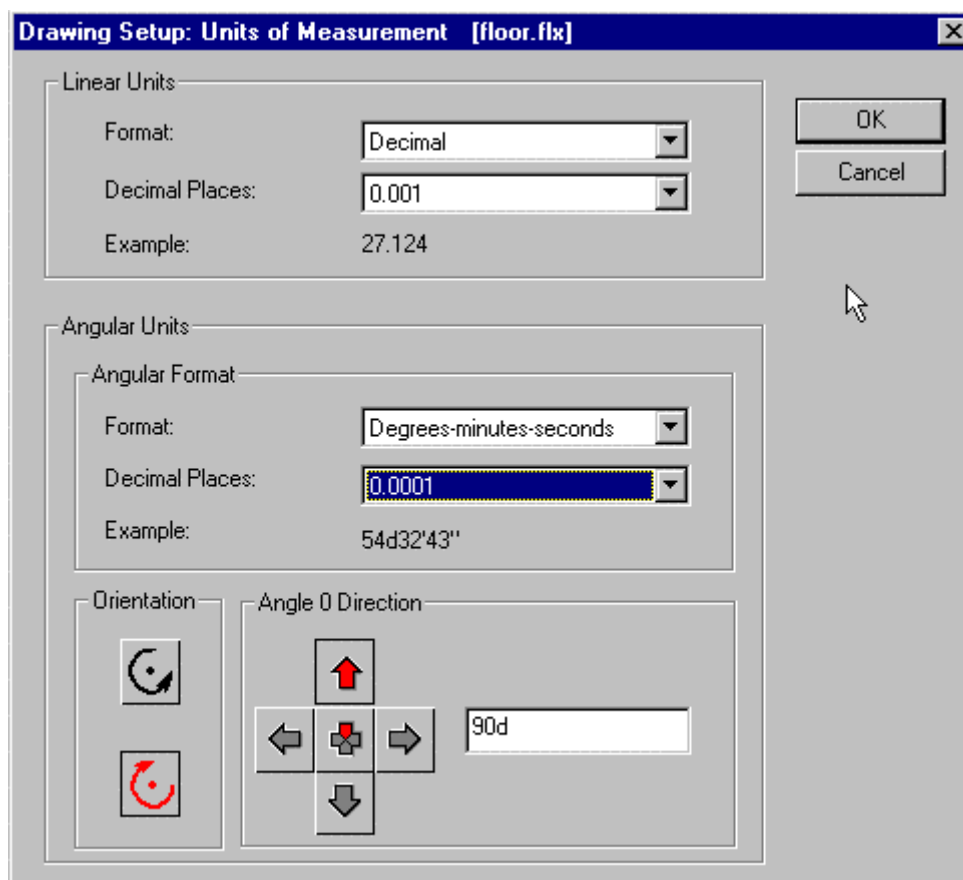
### Introduction

After reviewing our drawing checklist and drawing construction tips, we can use our knowledge of basic element creation and modification to develop our floor plan. After the linework is complete, we will add dimensions to the drawing.

### Getting Started

We will start the exercise by opening a new drawing that we will call Floor. This we do by using the File – New command and entering the new filename. We can elect to use a template drawing in the File – Open dialog, if we have one available with the desired drawing setup.

Once the drawing has opened, we will ensure that the correct units are setup for the drawing file. To do this we will use the UNITSETUP command with which we will be presented with the following dialog box:



To simplify the basic concepts involved, we have chosen to use metric units in this exercise and have therefore chosen a decimal unit format with three places of decimals. This will give us dimensions in meters with labeling accuracy to the nearest millimeter. Please refer to the section Using Architectural Units for a discussion on issues surrounding the use of feet and inches in

AllenCad. For our angular measurement format, we have elected to use degrees minutes and seconds and are using 4 decimal places in our labeling to give us a labeling accuracy to the nearest second. In the bottom section of the dialog, you'll notice that we have chosen "Straight Up" as the orientation for zero degrees from which we are measuring positive angles in the clockwise direction. Generally, zero degrees is set by convention as either "Straight Up" or 90 degrees to that - "Right"

Next, will go to the layer manager and create a layer called wall, on which we will develop the floor plan line work. We are now set to start drawing.

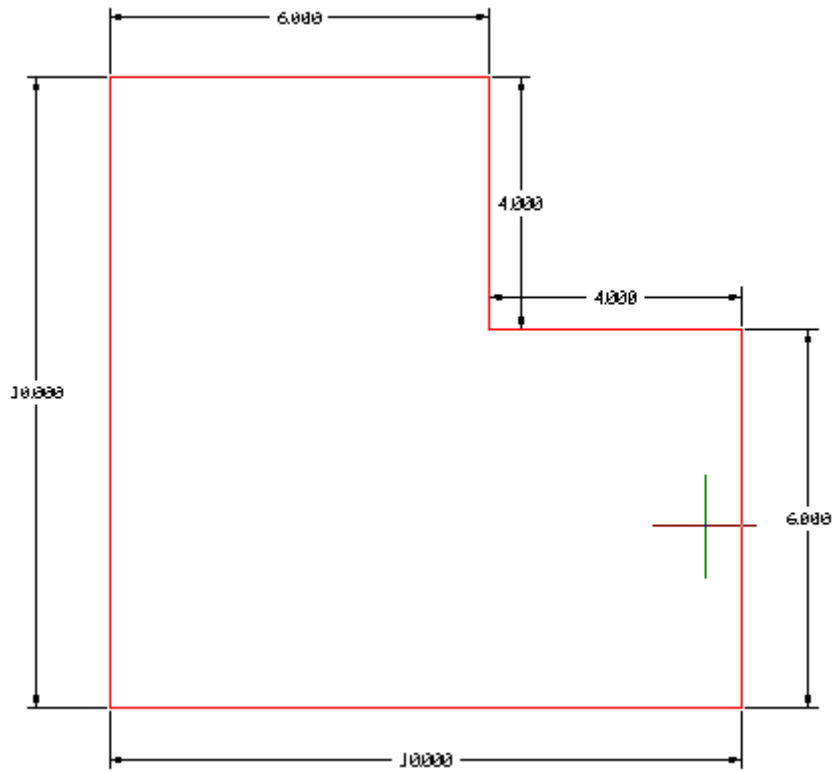
### **Drawing the Floor Plan Outline**

To draw the floor plan outline, we will use the Polyline command. Alternatively, a continuous line could be used, however we have chosen the polyline as it allows us to change line thickness and to treat the entire wall alignment as a single entity. This can be simpler to work with when editing. The polyline command is located in on the Draw pull-down menu. We need to start somewhere, so we will begin in the lower left hand corner of our drawing, which we shall set at coordinates 0,0,0. Having issued the polyline command, we are prompted for the starting point, to which we respond by entering 0,0,0 at the command prompt. After this, we could enter the coordinates of each corner of the floor layout if they were known. Instead, we will draw the floor outline using the distance and angle of each segment to determine the vertices. To do this, we'll respond to each prompt for the next point, by entering the @ symbol followed by the distance followed by the less than (<) symbol followed by the bearing or angle in which to draw the segment. For this exercise then, we will specify the polyline vertices by making command line entries in the following fashion:

```
@10<0  
@6<90  
@4<180  
@4<90  
@6<180
```

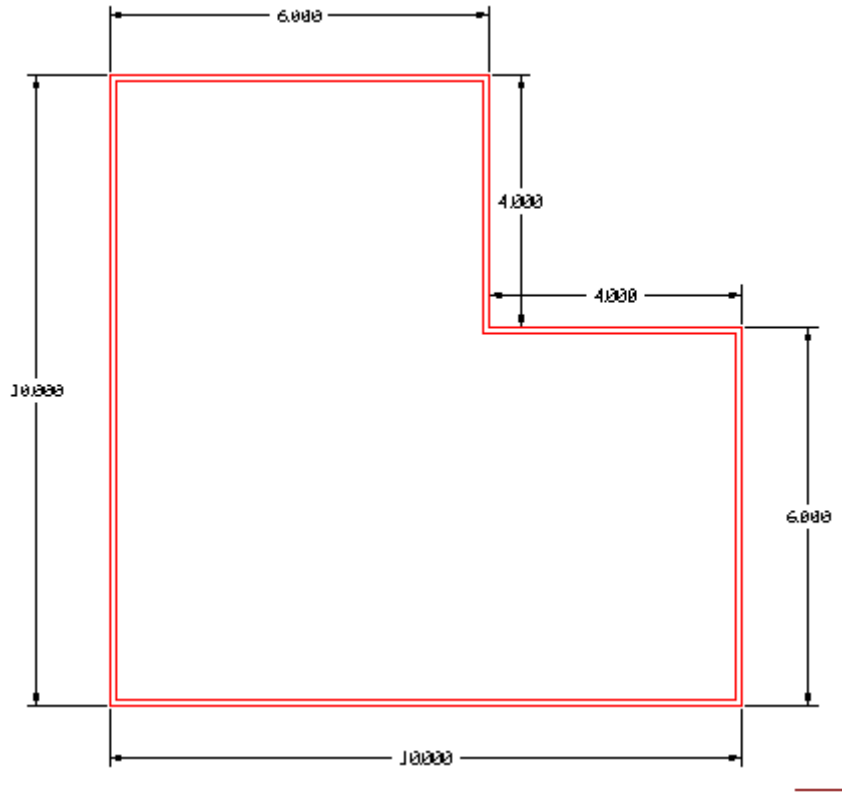
We will then close the polyline to the original start point by selecting Close from the option dialog. At this point, your drawing should look as follows (without the dimensions):





### Offsetting to Define Wall Thickness

Next, we will indicate wall thickness, using the Offset command. As these dimensions are external dimensions, we will need to offset by the wall thickness of 10 cm (0.1m) internally. After executing the command, you're drawing will appear as follows:



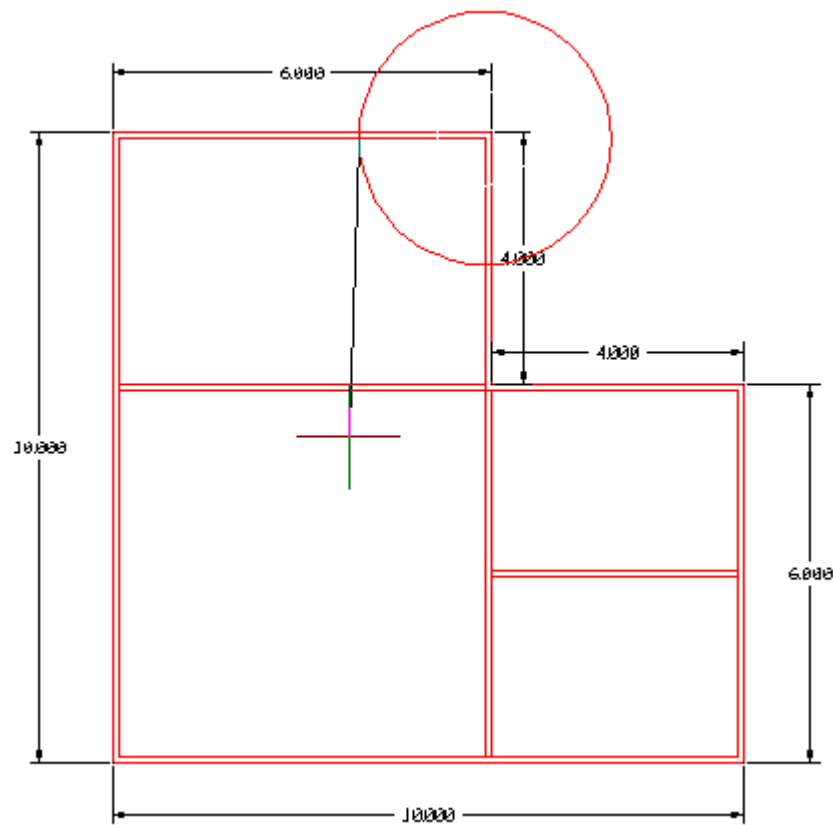
### Drawing Internal Walls

Having completed the external walls, we will now turn our attention to the interior walls. In order to draw these, we will have to establish the correct location for them in relation to the external walls and ensure that we have selectable points to use in the drawing exercise. In some cases, we may need to use construction lines to establish points that define the interior wall.

Prior to drawing the hallway, we will locate the other interior walls by drawing lines from the vertices on the exterior wall to points perpendicular on the opposite wall. In another instance, we divide an area into two equally sized rooms by drawing a line from the mid point of one line segment on another. We have then offset lines to establish wall thickness as we did with the exterior walls.

For the hallway, we will use the intersection of a 2 meter radius circle and the external wall to define one end of the hallway. Once the line is drawn, the circle can be erased. We will draw a line from this point, perpendicular to the opposite wall, as illustrated below. Once offset, as with the other lines, interior walls will be complete.

**Note:** If the walls had been drawn using line segments rather than a polyline, we would have chosen to use the Offset command to offset the external wall to the interior hallway wall location.



To draw this wall, we will select from the Draw pull-down menu or Draw palette:

> POLYLINE

Int (Type INT and then pick the intersection of circle and wall)

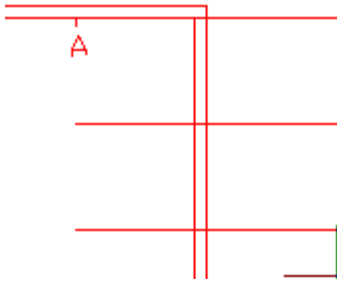
Per (Type PER and then select the opposite wall)

Enter (Press the Enter key to terminate the line)

Having done this we can erase the circle and then offset the line to establish the wall thickness in the drawing.

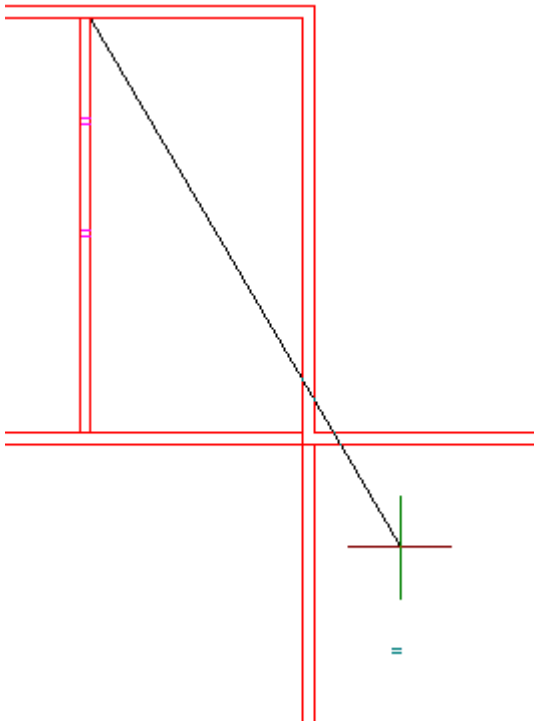
### **Drawing Door Jambs**

Next, we will use another drawing techniques to establish locations for the door jambs.



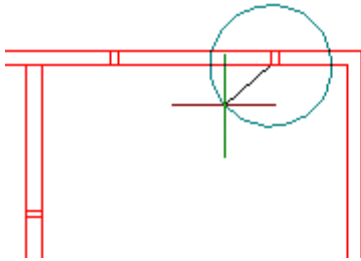
With Ortho mode on, we can begin to draw a line using the Object Snap "Near" the wall at point "A". We then extend the line beyond the corner of the wall to the right. If the jamb is one meter from the corner, we can offset this line one meter to locate one side of the jamb. We then offset that line the width of the jamb to establish the other side. After this, the original construction line can be erased and the wall sides used as cutting edges to trim off the excess sections of the lines.

Rather than redraw each and every door jamb, we can simply use the Copy command to copy elements from one location to another. Be sure to select an originating and destination point that relates identically to each jamb location

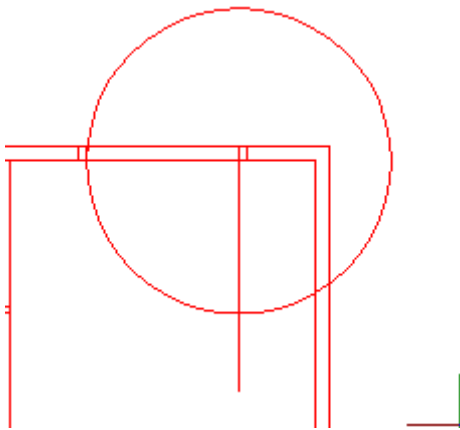


### Drawing Door Clearances

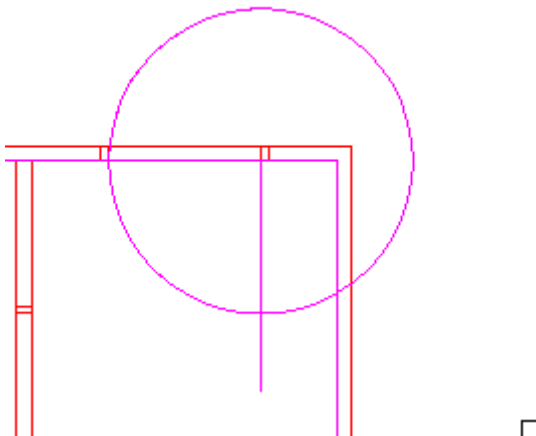
As door clearances consist of an arc and a line representing a door opened to 90 degrees from the Wall, we can simply draw a circle and line and then trim both to give us our symbol.



Here, we have set our object snap to "Intersection" and are drawing our circle with center on the door jamb that the door will be hinged upon. We then drag the circle across to the opposite jamb and click to snap it to the intersection point there.

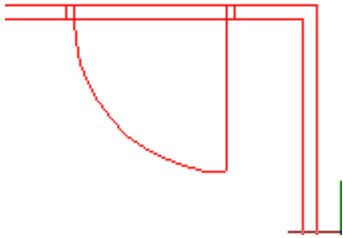


With orthogonal mode on, we can then simply draw our line representing the open door, by clicking on the center of the circle and extending a line to a point outside the circle.

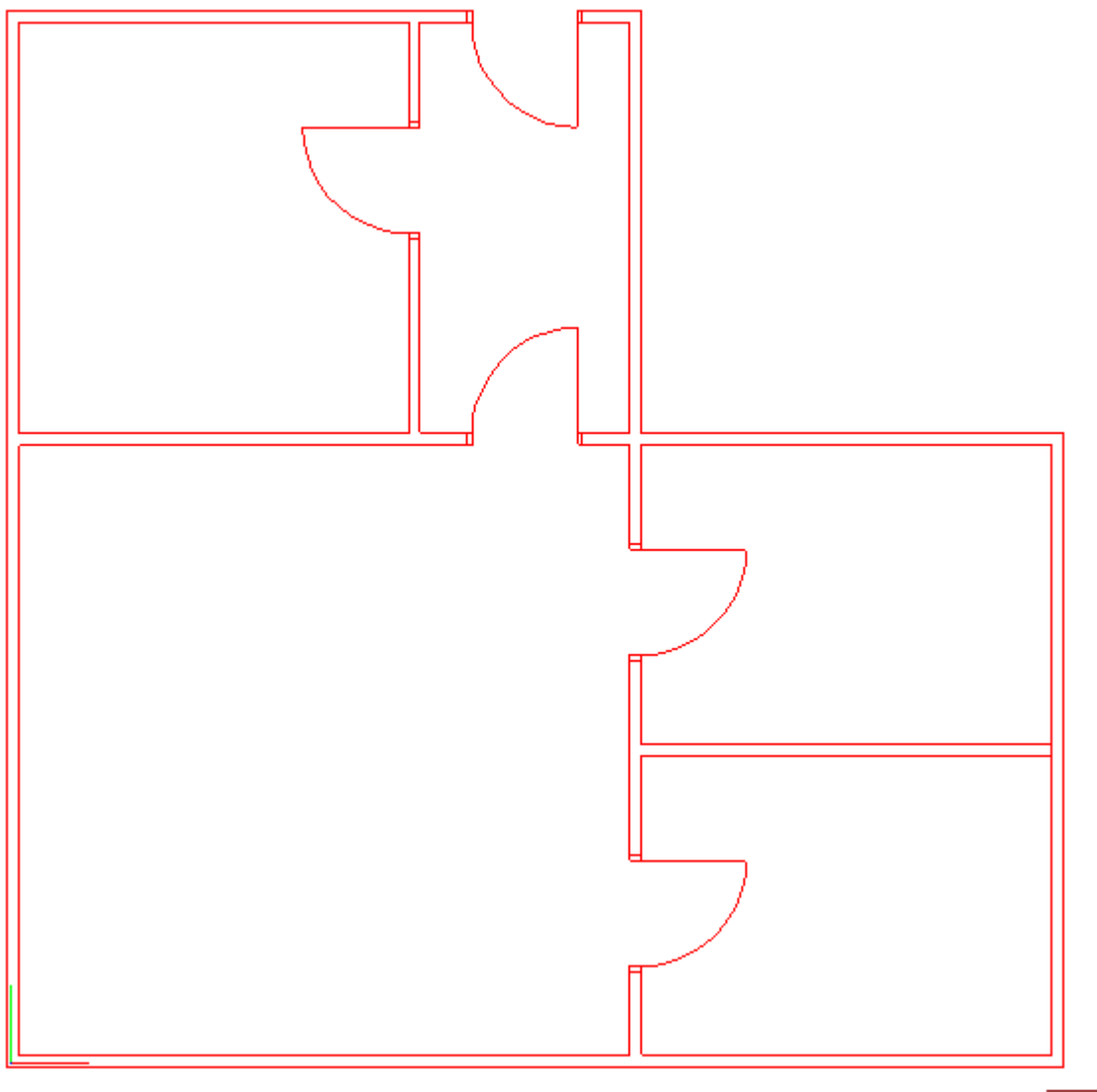


To complete the door clearance symbol, we use the TRIM command and select the circle, the line and the inside wall as our cutting edges to trim off the unneeded sections of circle and line, as

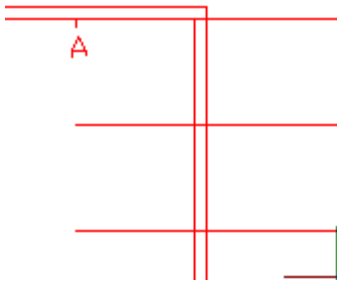
pictured below.



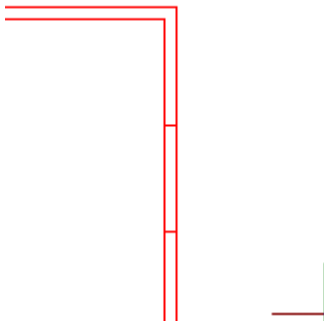
We can use this same procedure, copying the symbol where appropriate, to draw in the other door clearances in our floor plan.



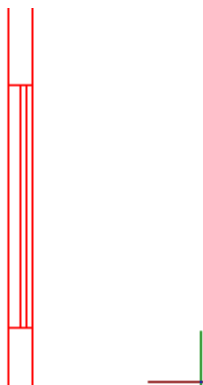
Next, we will use a similar construction method to that used when drawing our jambs, for drawing windows.



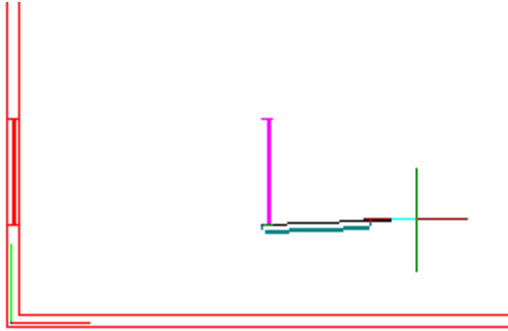
Again, we have Ortho Mode and Object Snap "Near" switched on and have drawn a line from "A" on the inside wall through the wall corner to a point beyond the wall. We can then offset this construction line the required distance from the corner to the edge of window and then offset that edge line the width of the window. We then can erase the construction line and use the wall lines to trim off excess sections for our window symbol, as shown below.



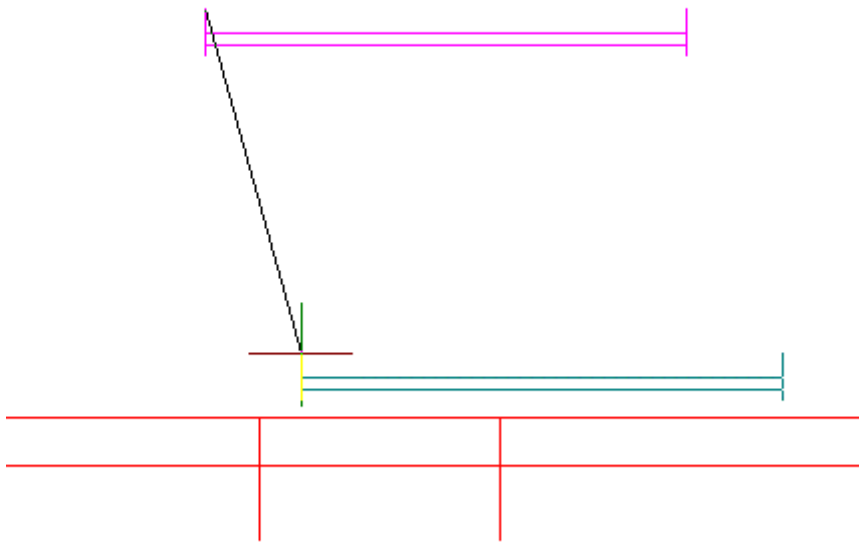
We then set our Osnap to "Midpoint" and draw a line from the midpoint of one window boundary to the midpoint of the other. With a wall width of 10cm (0.1m or drawing units), we can offset that line by one quarter of the wall width (0.025) to complete the window symbol as pictured below.



We can then use the copy and rotate commands with this symbol to place the symbols where we have other windows of similar dimension. Below, we have copied the window elements to a clear point in the drawing and are rotating the elements 90 degrees so that they can be moved or copied onto the adjacent wall.



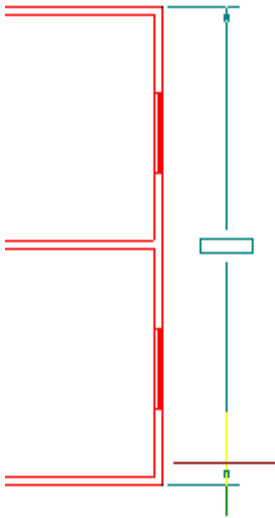
Below, we see that we have used a construction line to locate one window edge on the wall and we are moving the window element into place using the endpoint on the symbol as our origin point and the construction line's intersection point on the wall as our destination point. If more than one window were to be placed on the same wall, we could use the copy command to add additional windows.



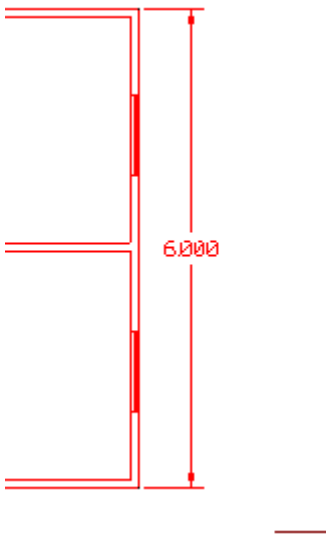
### Dimensioning

Our last remaining task is to dimension the drawing. With our Object snap set to endpoint, we select "Vertical" on the dimension (Dim) pull-down menu and then click on the polyline vertices on the external wall, as shown below. Having clicked on the corners and confirmed the measurement that was suggested (you can change this to any alpha-numeric value IE 6.0m), we are showing the note being dragged into a position that will become fixed with a left mouse button click.

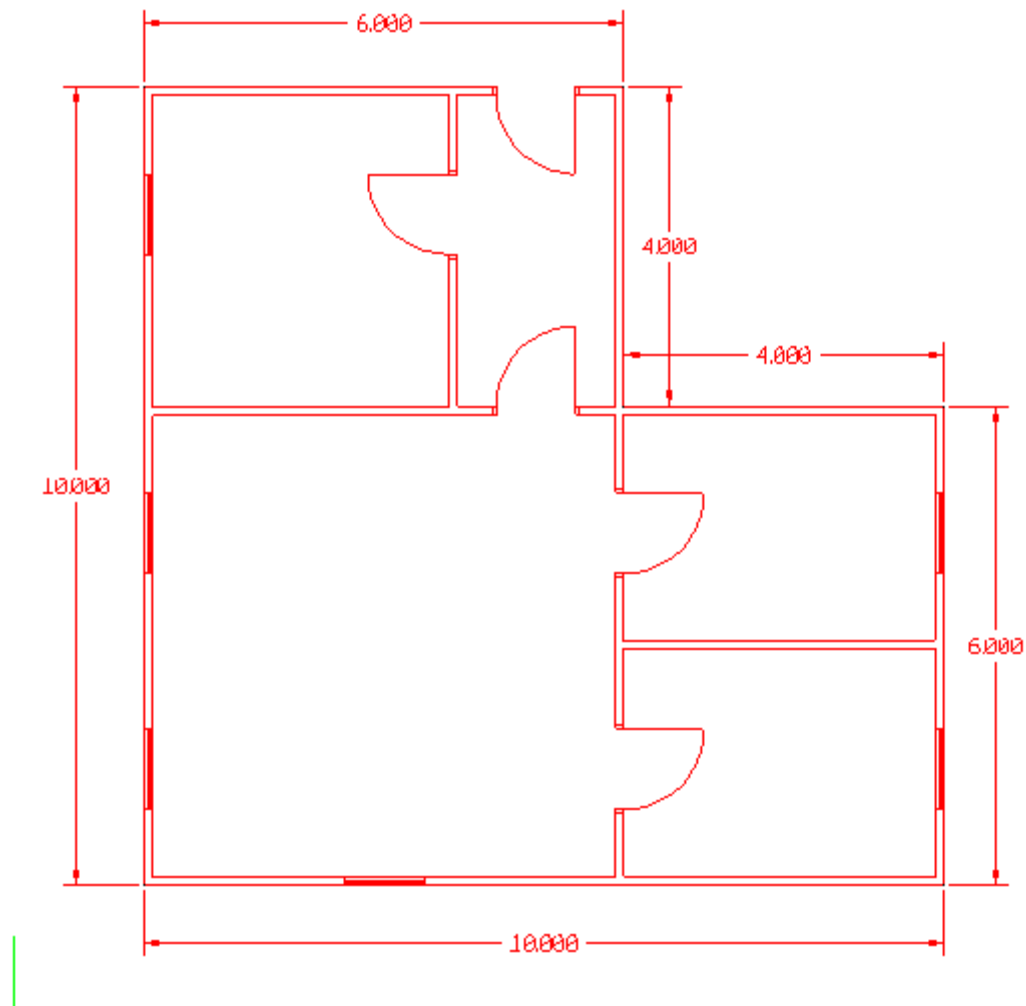




Once placement is confirmed, the dimension note will appear as below.



We can then complete the dimensioning as shown below, using (in this case) both horizontal and vertical dimension commands.



This completes our floor plan exercise.

## Adding Text and Plotting the Floor Plan

### Introduction

In this exercise we will label the rooms that we defined in our previous floor plan drawing exercise and then plot the drawing.

### Adding Text

To add text to the drawing, we will use the TEXT command (Text on the Detail pull-down menu). When the command is executed, we are presented with the following dialog:

**Text**

LIVING ROOM

X: 0.000 Y: 0.000 Z: 0.000 ☒ Interactive

**Options**

Height: 1.00

Angle: 90d

Font: STANDARD

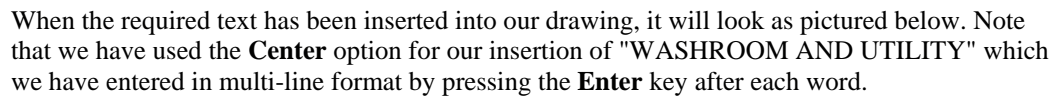
**Insertion Reference**

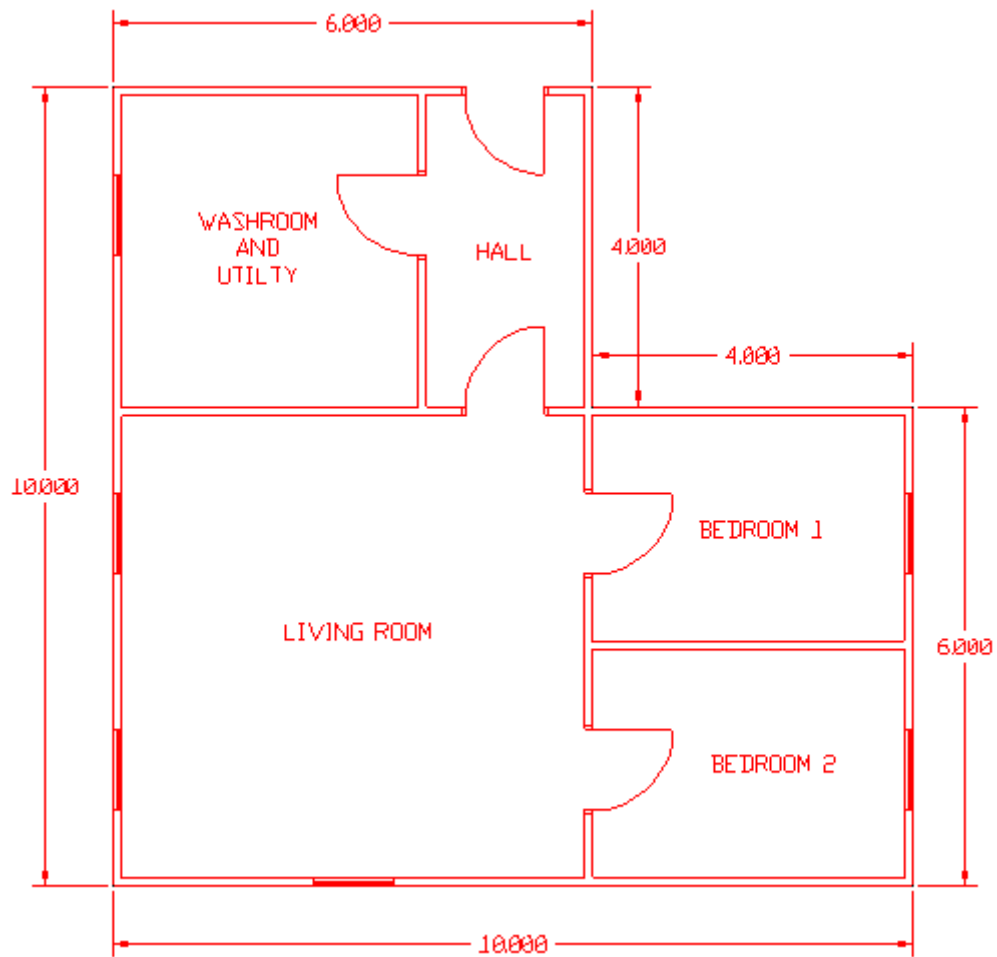
☐ TL ☐ TC ☐ TR  
☐ CL ☐ CC ☐ CR  
☒ BL ☐ BC ☐ BR  
☐ UL ☐ UC ☐ UR

☐ Align ☐ Fit ☐ Center

OK Cancel

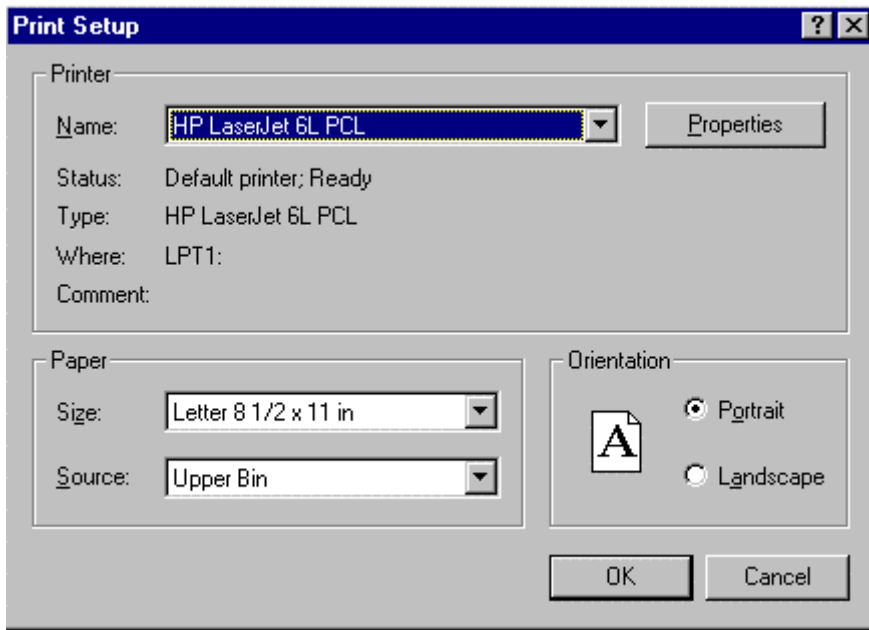
Here, we can enter multi-line text in the text window and control height, orientation and font. With the Interactive box checked, we can place the text in our drawing using the cursor as shown below. Note that the text box attached to the cursor is the size of the text to be inserted and that text position is outlined during the insertion process. Text can alternatively be inserted at a specific location by un-checking the Interactive box and specifying the coordinates of the insertion point in the specific fields provided on this dialog.



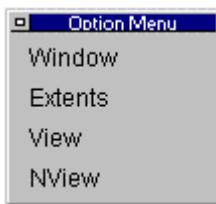


### Plotting our Drawing

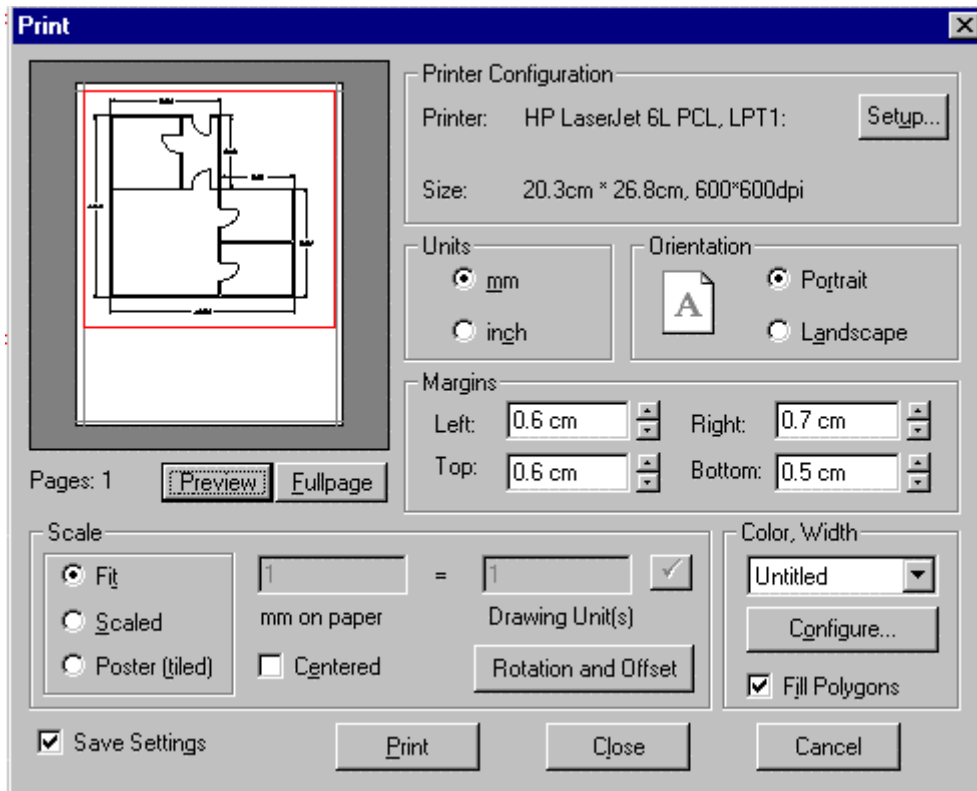
When we are ready to plot our drawing, we can confirm our present Windows print settings using Print Setup (command PSETUP) located on the File pull-down menu. We can make alterations as required and as we would do in other Windows applications.



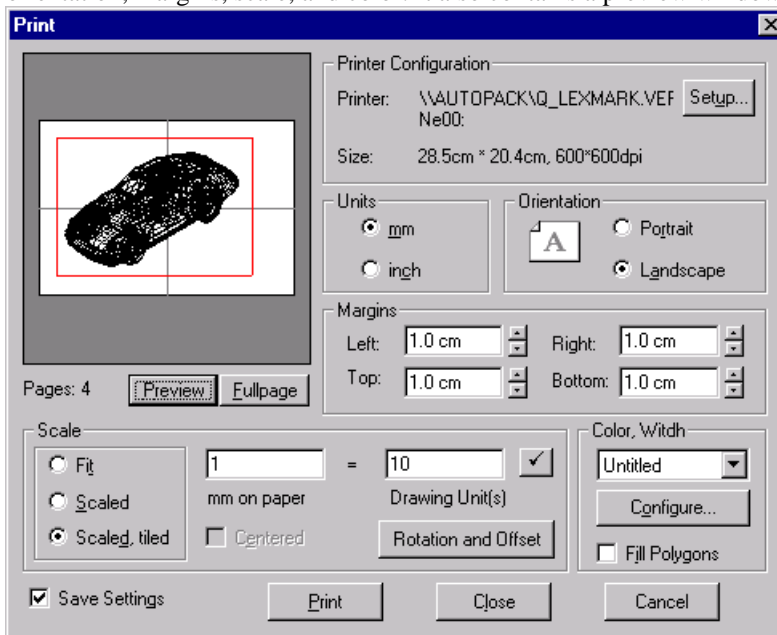
In AllenCad, we use the PRINT command, regardless of whether the output is a printer or plotter. The program makes no distinction between the two and relies upon Windows to administer print devices. When we issue the PRINT command (File – Print), we are presented with the following dialog.



Here we can elect to print a windowed area on the screen that we are asked afterwards to define; the full extents of the drawing, whether the full drawing is onscreen or not; the current onscreen view of the drawing or lastly, a saved and named view of the drawing. Having made our selection and after defining a window or selecting the name of the view (if we have selected these options), we are presented with the Print Dialog as shown below. This dialog offers the use of the current default printer (selectable in Windows under Start – Settings – Printers )



This dialog allows you specify parameters specific to AllenCad, such as configuration, units, orientation, margins, scale, and color. It also contains a preview window. This window



assists you in seeing the effect of changing settings in layout and scaling.

- The **Setup** button will execute the PSETUP command and call up the printer setup as described above.
- The **Units** radio buttons allow you to select the units that you are measuring paper dimensions, margins, etc. in.

- You can select between **Portrait** and **Landscape** plotting. Portrait prints the drawing upright on the paper where Landscape causes the printer to print sideways on the paper.

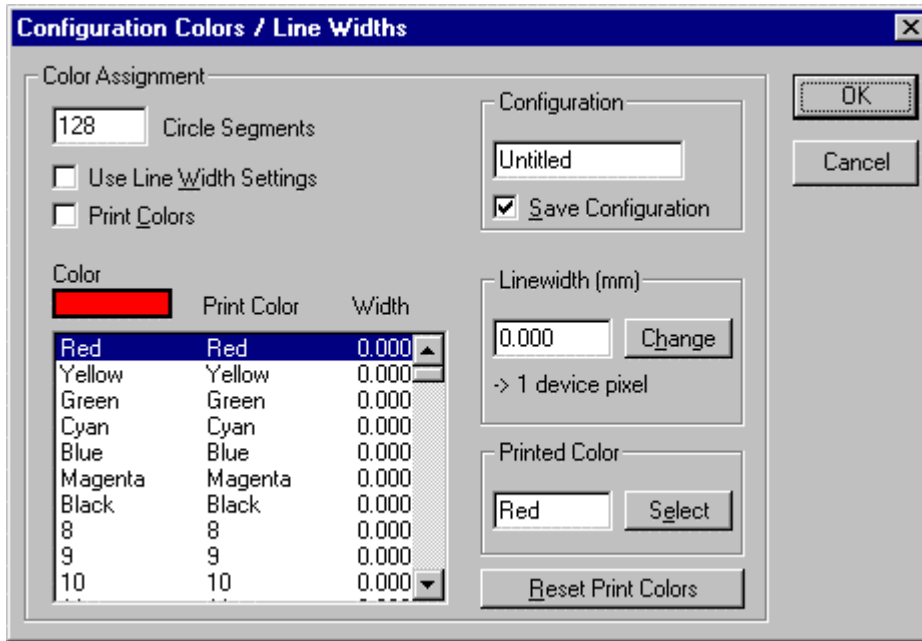
Margins allow you to reserve additional margin space around the printed drawing, as required. Zero margin widths will cause the printer to print to the full width it is capable of on the paper. Regardless of the printer used, there will be a narrow boundary left around the drawing where the printer cannot physically print.

In the **Scale** section of the dialog, there are three options available:

1. **Fit** – With this option, the printer will produce the drawing information selected at whatever scale that would be necessary to fit the full length or width of the paper (whichever dimension would govern). There is therefore no direct control on the scale produced.
2. **Scaled** – With this option, you can directly control the plotted drawing scale. You can establish plot scale to specify as follows: a. Where paper is specified in millimeters and 1 drawing unit represents 1 meter, the scale plotted if 1 paper unit to 1 drawing unit were specified would be 1mm : 1000mm or 1:1000. b. If we want to produce a plot at a 1:250 scale, we would have to specify 1 paper unit : 250/1000 drawing units or 1 paper unit to 0.25 drawing units. This could also be expressed as 4 paper units: 1 drawing unit if you prefer.
3. **Poster (Tiled)** – This option is very useful when the drawing size and required plot scale makes the drawing larger than a single sheet of paper. This option will send up to 32 different sections of the drawing to the printer with a single print command. The sections can afterward be spliced together to create the full drawing. The number of pages that will be printed is shown immediately below the drawing preview window
4. **Centered** – This option works only in conjunction with the Fit option. Where Fit will scale the drawing to fit on the more restrictive of the paper's length or width (not both), the Centered option will center the drawing in the other (non-restricted) direction. IE where the drawing width extends across the full width of paper, the drawing is centered on the paper lengthwise.

**Color, Width** – This area is for assigning plotted line widths to line colors used in the drawing. By assigning line widths, you enhance the appearance and legibility of the drawing. To set up colors and line widths, click the **Configure** button, which displays the **Configuration Colors / Line Widths** dialog box.





To set up line widths, you associate the line widths with color numbers. When you created the drawing, every object was assigned a color, or, more accurately, a color number (also called the color *index*). For detailed instructions on the definition of colors for layers or for objects, see "Layer and Object Properties ". For example, red is color #1.

Any color can be given a line width in mm or inches. For example, you can assign a line width of 0.36mm to color red. This means that all objects colored red in the drawing will be plotted 0.36mm wide. The setting of line widths can be saved under a configuration name, and then reused by future printing operations.

- The **Use Thickness Settings** check box determines whether line widths will be used during plotting.
- The **Print Colors** check box switches the color printing on or off.
- The **Color / Print Color / Width** list box allows you to coordinate screen and plotted colors and plotted width, by first selecting the screen color in the left column and then changing print color and printed width in the boxes to the right. The Reset Print Colors button restores default values to all screen colors.

### Configuration

You can save the line widths and color configuration by entering a name in the **Configuration** area, then ensuring the **Save Configuration** check box is on.

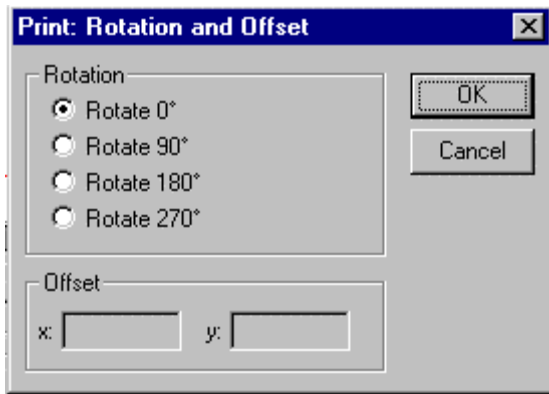
### Fill Polygons

This check box forces AllenCad to plot filled polygonal areas. If the box is not checked (the default), only the outlines of filled areas are output, which can speed up plotting.

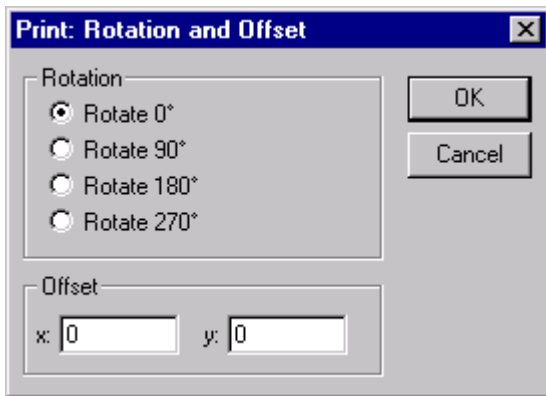
Click **OK** to record the changed line width and color assignments and to return to the plot dialog.

### Print Rotation and Offset

Click the Rotation and Offset button to display the Print: Rotation and Offset dialog box.



1. The **Rotation** area allows you to select a plot rotation in increments of 90 degrees. The drawing is rotated on the sheet by 0, 90, 180, or 270 degrees. Rotation is applied clockwise.
2. The **Offset** option moves the plot origin from the lower left corner (0,0) of the paper (the normal position) to another location on the paper. This lets you plot the drawing in a specific location on the page. Specify the x and y values in paper units (either mm or inch, as determined in the units section of the plot dialog).




All settings are reflected in the preview of the dialog box in the full-page preview. Press the Print button when are satisfied that you have the settings you need for your plot.

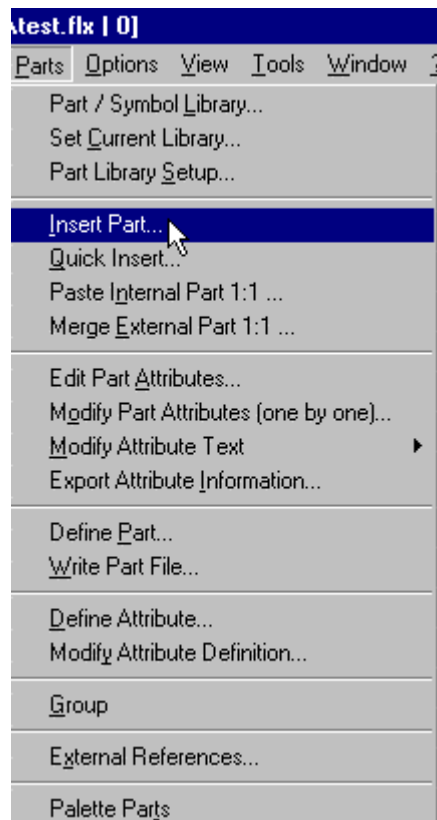
## Inserting Parts Into the Drawing

With AllenCad, you can insert one drawing into another any number of times and specify an insertion point, scale factor and rotation angle. When used in this fashion, external drawing files are referred to as external Parts to the drawing being created and the part name is simply the drawing file name. Parts can also be created within the drawing from a selection of drawing entities and saved as either an external drawing file (Part) or simply defined and named within the current drawing.

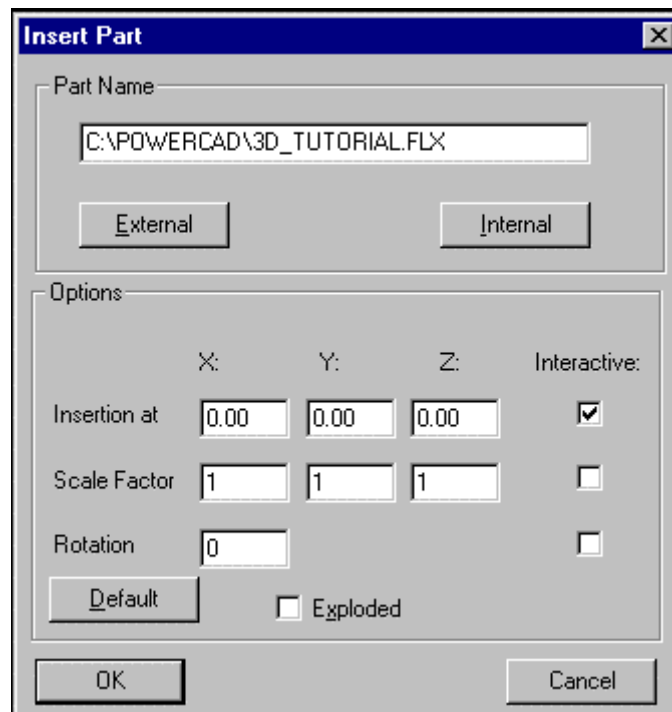
As you can imagine, being able to re-use existing drawing information saves much time as well as repetitive drafting work.

Whether the part is external or internal, you will begin the insertion by selecting Insert Part ... on

the Parts pull-down menu or selecting the  icon on the Parts Palette.

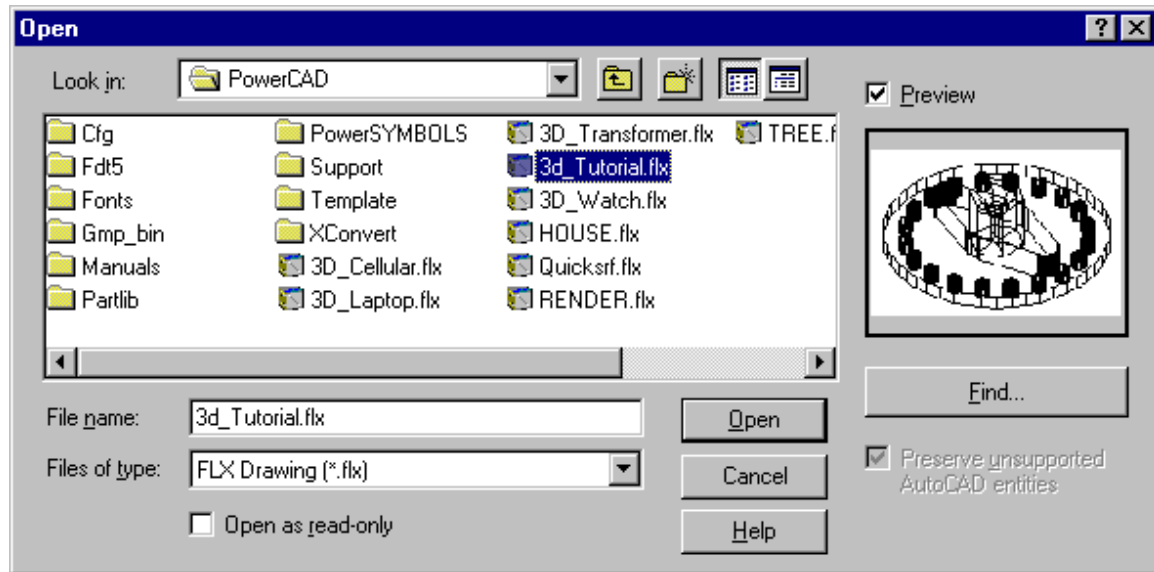


When this command is issued, the following dialog is displayed:



Here, you can re-use an earlier inserted part name indicated in the window, or else choose to insert a different internal or external part by pressing on the appropriate button. Before leaving this dialog, however, you may wish to over-ride default values for insertion point, scale factor and rotation by selecting the "Interactive" check boxes beside each. You may also choose to bring in the part as an exploded entity if you will need to modify individual components within the part. Don't worry if you fail to address these issues at this point, as they can be specified later on, as well.

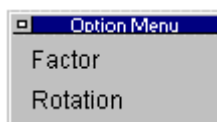
Selecting a new external part for insertion will bring up the following dialog:



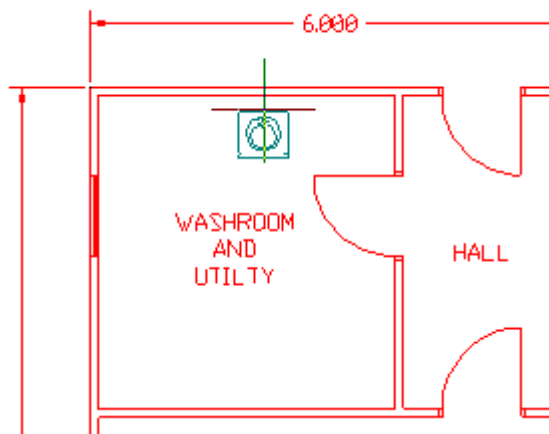
Here, you can browse to find the drawing file you want by picking the "down arrow" next to the "Look In:" window. Clicking on a particular drawing file will result in a view of that drawing appearing in the preview window when the Preview checkbox is checked.

You can elect to display either AutoCAD DWG or DXF, AllenCad FLX drawing files, or else all (\*.\*) files, in the **Files of type** field. Simply select the down arrow to the right of the field to see your options and make your choice. In this case, our external parts are AutoCAD DWG files, so we have made the appropriate selection.

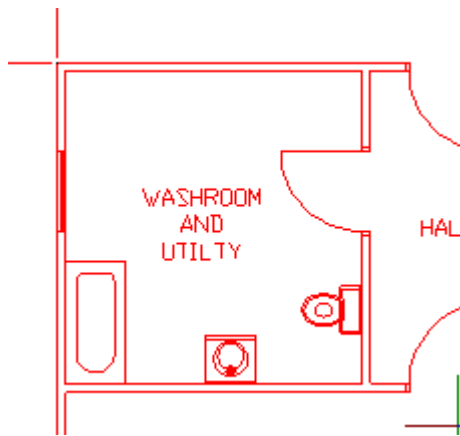
When you select the Open button, you are offered the insertion option dialog, that allows you once again to change default values for scale factor and rotation angle for the inserted drawing information.



At the same time, an image of the part appears at your cursor. This is at the current default or selected values for scale factor and rotation. You can use an object snap to fix the insertion point, or else simply click on the drawing at the appropriate point to fix the part's location.



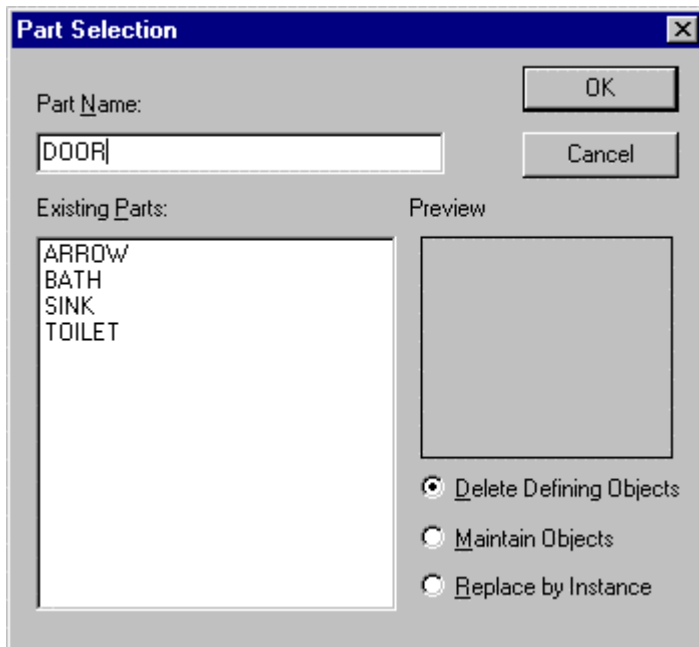
Repeating the process as needed, allows us to complete our exercise with the remaining washroom fixtures:



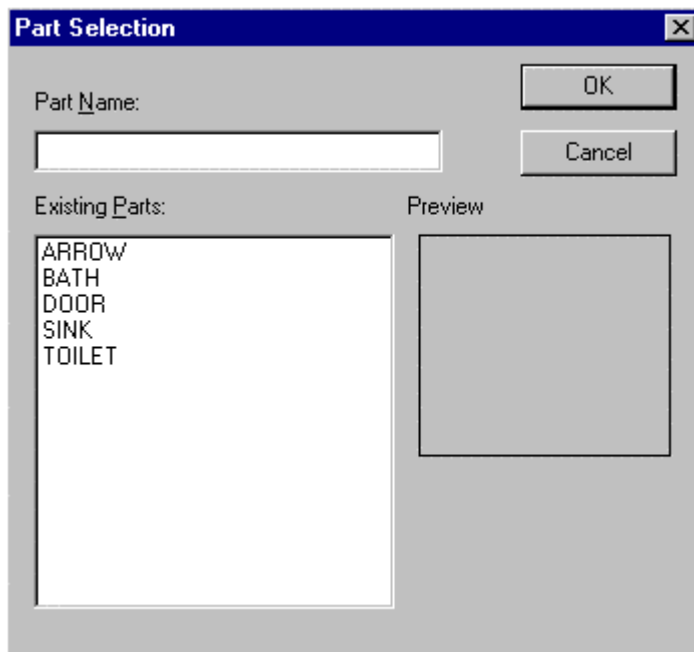
### Internal Parts

Apart from using external parts as we have in this exercise, we can also create internal part definitions within the drawing, whenever we recognize that we will need to re-draw certain elements.

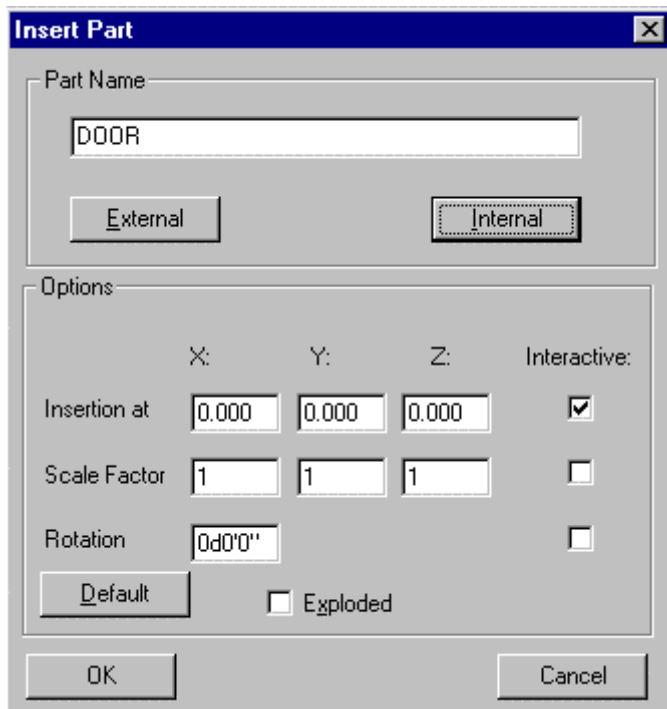
To demonstrate this feature, we will create an internal part that we could use to duplicate the door closure symbols that we used in an earlier copying exercise. To create the part, we use "Define Part" on the Parts pull-down menu and enter our new part name "DOOR" in the name field as shown below. While doing this, we notice that all other parts defined in the drawing are listed below and that a preview window is available. This allows us to re-define an existing part if that should be necessary. All insertions of a part that is being re-defined will be updated to the new definition, which is another significant labor saving feature..



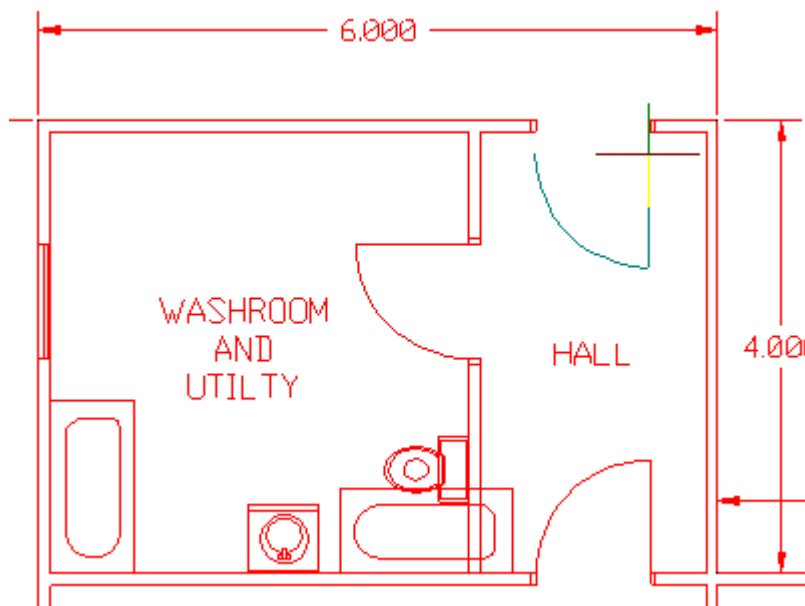
After specifying the part name to be created, we are asked to define (using Object Snaps) the insertion point for the new part as well as to select the entities that will comprise the new part. The internal part is created and is ready for insertion once all entities have been selected and the command terminated with a final press on the Enter key.




looking at our drawing, we notice that the door closure that we used in our part creation exercise has disappeared. We then insert the part "door" in it's old position as well as in any other new positions as is required. We again use the Insert Part selection on the Part pull-down menu and notice that "Door" has become the default part since it was the last part created or used.




We can then simply confirm the selection with OK and then drag the symbol to its old location as shown below.



The Define Part command can also be accessed from the Part palette when we use the  icon to launch the command.

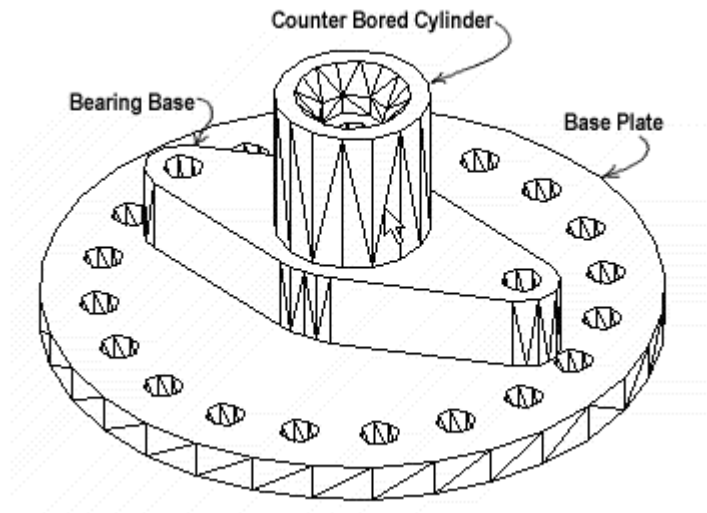
If we want to create a new external drawing file for use in other drawings, it is not necessary to open a new drawing file that we would save and use as a drawing part. We can simply select entities in our drawing and create an external part (drawing file) by selecting "Write Part File on

the Parts pull-down menu, or else access the command through the use of the  icon on the

Parts palette. This will create an external drawing file without the need to quit the current drawing.

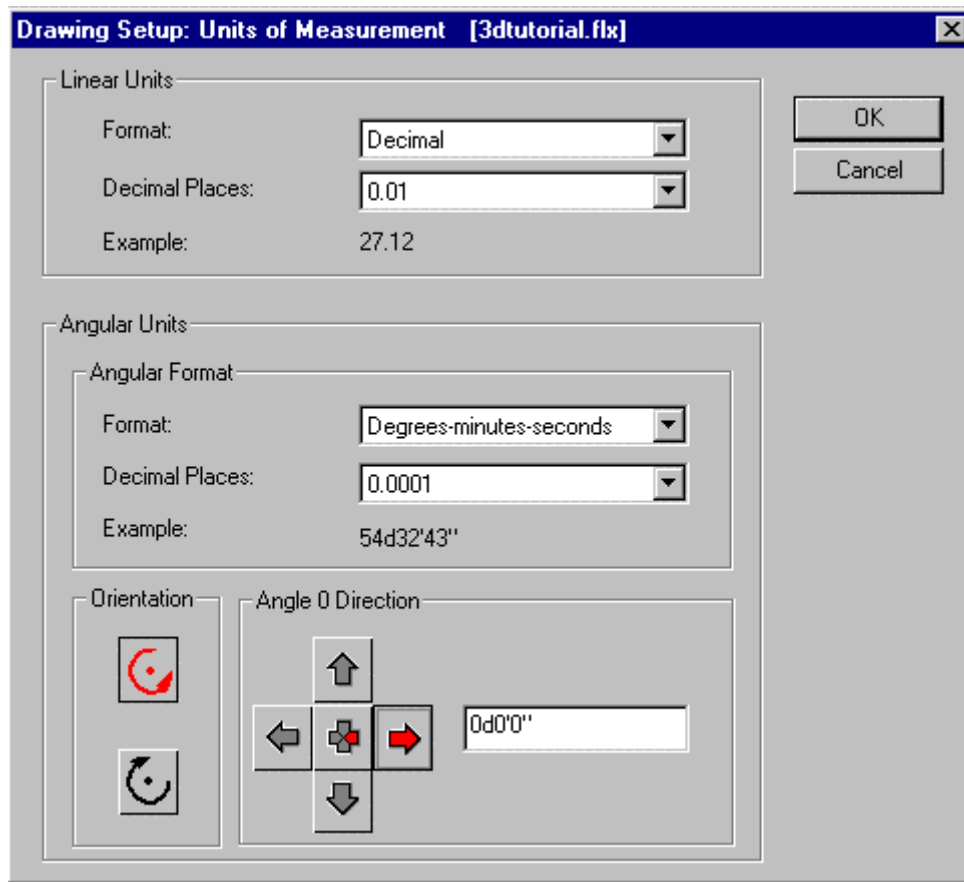
## Creating a 3-D Model

During this exercise, we will use our drawing tools to create a 3-D model of a mechanical part as pictured below:

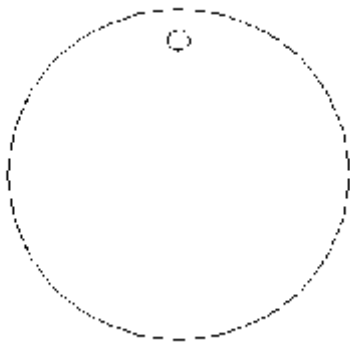


To begin with, we use the UNITSETUP command to ensure we are beginning our drawing using the correct unit settings as indicated below:



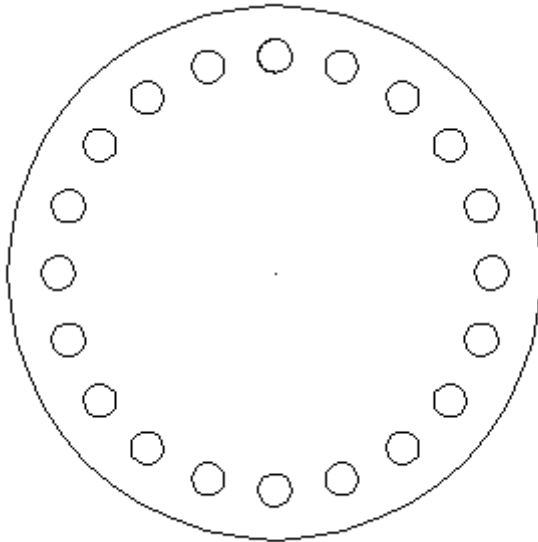


Next, we begin to draw our baseplate by drawing a circle at center (0,0) with radius 80. We then draw a second circle to represent a bolt hole at center (0,65) and radius 5, so that our drawing appears like this:

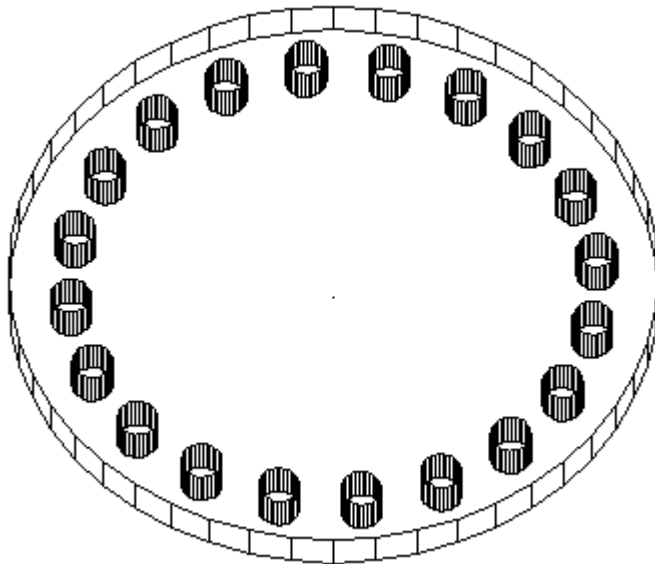


We want a total of 20 bolt holes arranged equally spaced around the baseplate, so we will use the ARRAY command to copy the bolt hole in this pattern. We will need to use the circular array and as we will be arraying circles, rotation is not required, so we will simply use the CircNRot array option. After issuing the ARRAY command, we select the bolt hole circle and then the center of the baseplate (Using the CEN osnap and picking the baseplate circle) as the basepoint. There are 20 instances in which this will be placed (including the original location), so we respond with 20 when prompted. We also want these arranged around the complete baseplate so

we input 360o when prompted for the angle to fill. When the command completes, we have a drawing as pictured below:



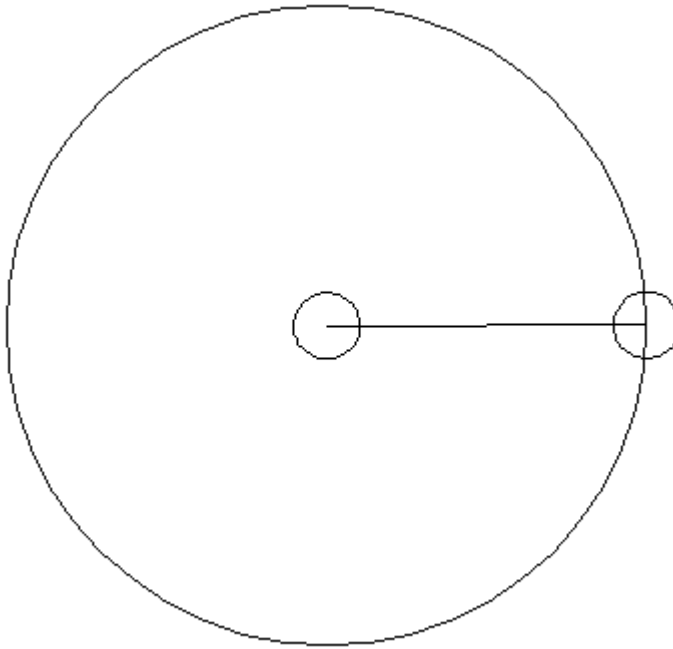
Next, we use the PROPEdit command to change the thickness of the baseplate entities and then use the 3Dview option on the View pull-down menu to see the result. For this, we will specify a 300o rotation IN the X-Y axis and a 35o rotation TO the X-Y axis



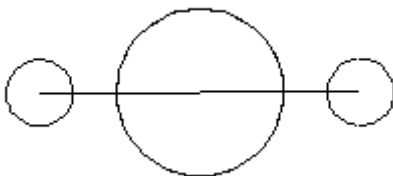
At this point, having confirmed that we have achieved the desired result, we will revert to a setting of 0o In and 0o To the X-Y axis to carry on with the next phase of model construction.

Moving away from our model, we will next build the next phase to the model, which will later be added to first phase. To do this we draw a circle with radius 10mm clear of the existing drawing. We need next to establish a point 96mm away from the center of this circle. We therefore offset the circle 86 units and then with ORTHO set to ON, we will draw a line horizontally from the

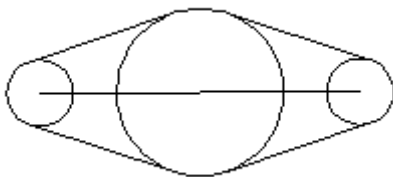
center of the circles beyond the circumference of the 96 mm circle and then use the circle as cutting edge to TRIM the line to length. We then draw a 10mm circle at the end of the trimmed line. At this point, we have a drawing that looks like:



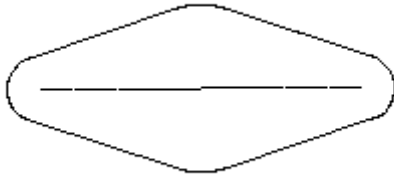
We can now DELETE the larger 96mm circle that was used to establish the center of the second 10mm circle. Using the line between circle centers, we draw a 25mm radius circle with center at the midpoint of the line. We now have a drawing that looks like:



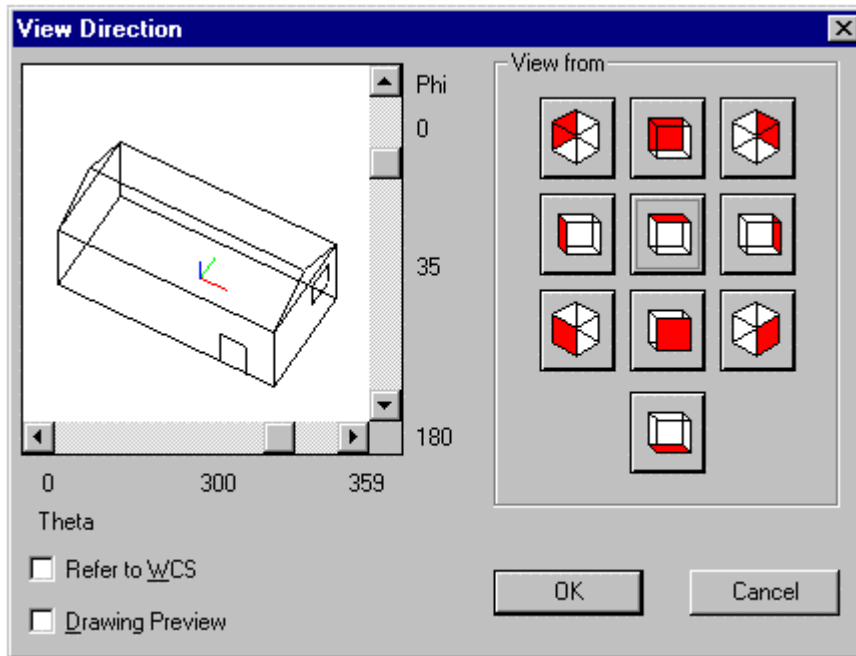
Next, we will set our Object Snap Tangent to ON and draw four lines tangent to the large and smaller radius circles:



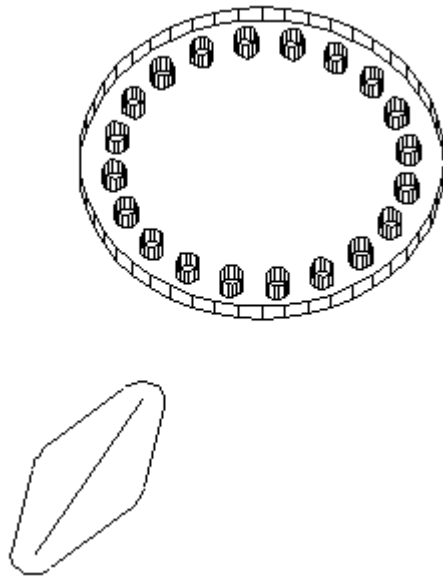
Then, using the line segments that we have just drawn as cutting edges, we will trim off the unneeded sections of the circles to give us a drawing that has the following appearance:



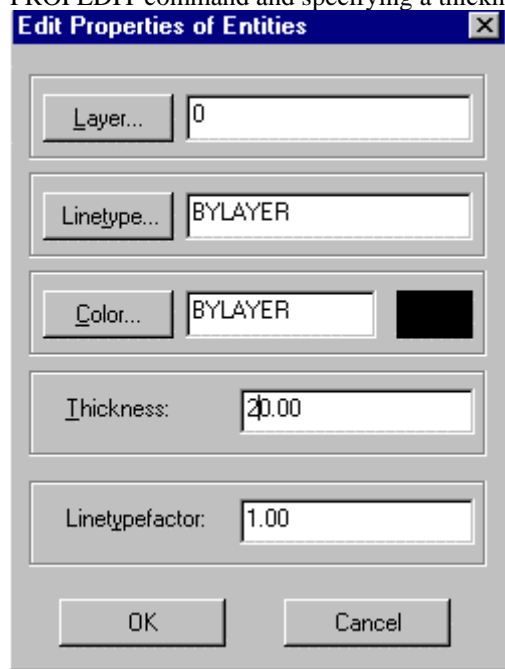
Next, we use the Edit Polyline option on the Modify pull-down and select any line segment on the outline as the polyline entity to edit. Then after confirming that we want this line segment converted to a polyline, we select the JOIN option and then select the remaining arcs and line segments around the perimeter to be joined to form a closed polyline around the perimeter. At this point, we choose the 3DVIEW option from the View pull-down menu and adjust the Theta value to 300 (IN the X-Y direction) and the Phi value to 35 degrees TO the X-Y plane, using the sliders:



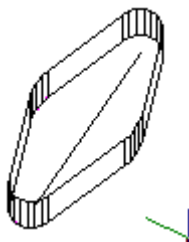
And our drawing suddenly changes to this:



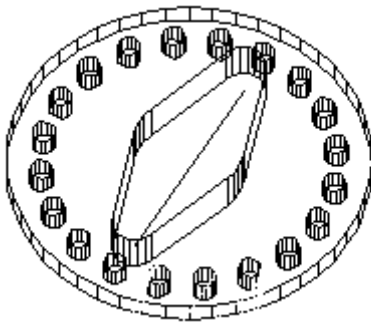
At this point, we extrude the bearing cover perimeter that we have just drawn using the PROPEdit command and specifying a thickness of 20.



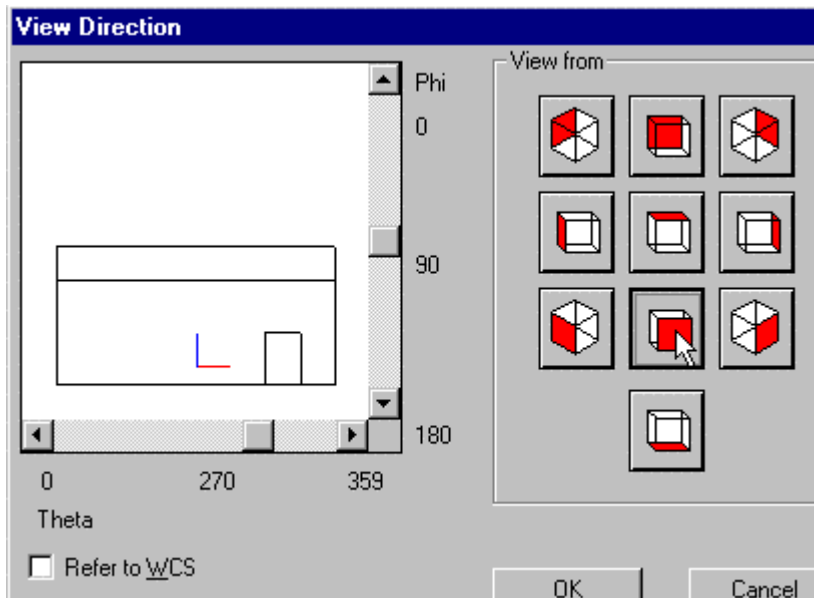
And we have :



Next, we will move the bearing cover onto the baseplate and center it by selecting mid point of the cover and 0,0 (the coordinates of the center of the base plate) as from and to points in the move command. At this point, our drawing appears as:



Next, we will want to draw the third segment of our drawing, the shaft. To do this, we will want to orient our view to a point sideways to the baseplate and we will therefore select a Front view of our model, with the 3DVIEW command.

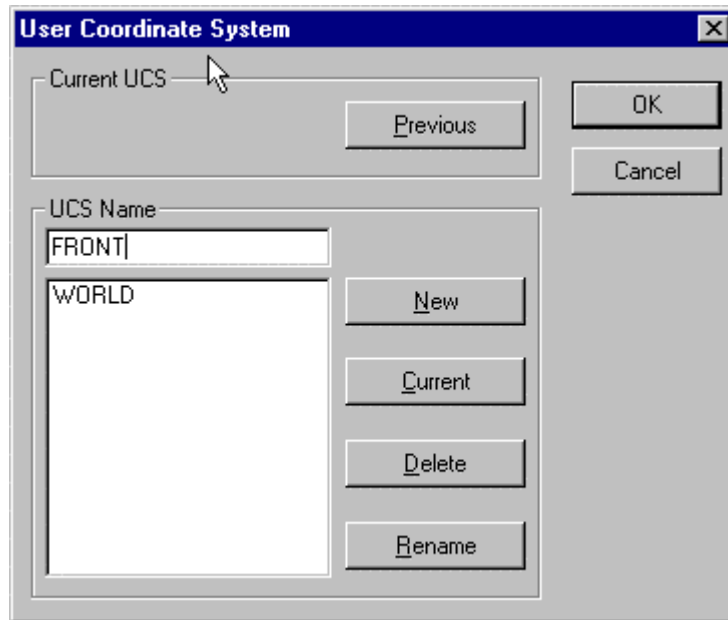


The Front view is easily selected from the 3Dview dialog by pressing the indicated option. Having done this, our drawing appears as:

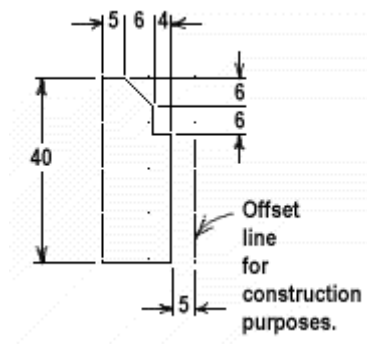


From this perspective, we can easily construct the shaft, using lines and offsetting them in this plane as we did earlier in the X-Y. Before we begin drawing however, we will have to setup a UCS that corresponds to the current view. This we do by selecting the View option that is offered after issuing the UCS command. Because we may want to return to this view for later editing or additions, we could save the UCS and name it for faster setup. To do this we go to the Options Menu and select User Coordinate System, then select UCS Manager. In the dialog that is presented below, we will enter our new UCS name – FRONT and press the New button. When

we do this, the FRONT UCS joins WORLD in the lower window that shows all coordinate systems that have been saved and are available for use in the current drawing.



In the sketch below, we can see the elements and their dimensions that we need to draw in order to construct the counter bored cylinder. These elements consist of a closed polyline and a single line segment that we will use as the axis of rotation when we generate the cylinder.



To simplify the drawing process, we will draw the cylinder separately from the existing drawing information and simply move it into place when it is complete. With simple dimensions to work with, we elect to enter our points using coordinates and have chosen a point (130,0,0) as the start point for our polyline (the bottom right corner).

Where we normally enter coordinate values in (x,y,z) format in the world coordinate system, we must enter them in (x,z,y) format when we have oriented our UCS to a frontal view. To draw the polyline then, we will issue the command: POLYLINE (Draw – Polyline) and respond to prompts for vertices as follows:

```
>POLYLINE
130,0,0
130,28,0
126,28,0
126,34,0
120,40,0
```

115,40,0  
115,0,0  
Close Option – to close the polyline to the start point.

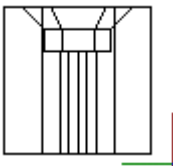
We then can draw the line segment that will be our axis of rotation by issuing the command:

>LINE  
135,0,0  
135,28,0

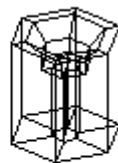
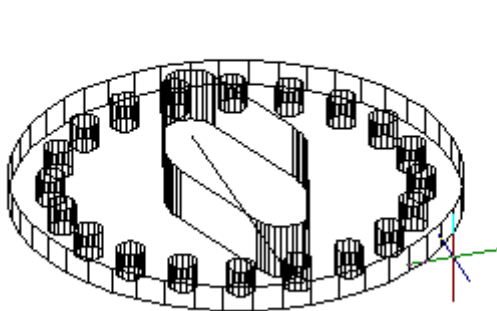
At this point, we should have our polyline and axis appearing as follows:



Next, we will construct our cylinder using these elements, using the REVSURF command (Draw – 3D – Surface of Revolution). When we issue the command, we are prompted first for the "Path Curve to Revolve" to which we respond by picking the polyline. Next, we are prompted to "Select Axis of Rotation" at which point we select our axis – the line segment. When the command completes, our cylinder appears as below.

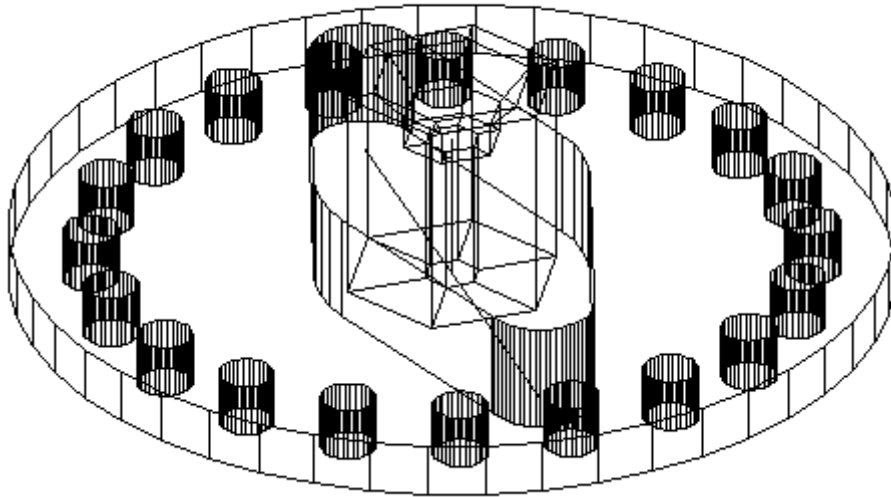


Next, we need to move our cylinder into place on the base plate, but first we need to change our view back to about 300° on the X-Y axis (Theta) and 35° TO the X-Y Axis (Phi), using our View – 3D (3DVIEW command). As we can see below, this makes it much easier to select our "to" and "from" points in the MOVE command (Modify – Move).

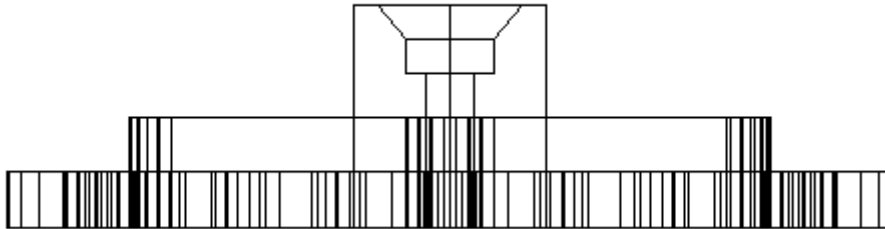


When we issue the command, we first select all of the cylinder entities when prompted for entities to move. Next, we will pick the bottom endpoint (using Osnap ENDP) of our cylinder's central axis, and then the midpoint (Osnap MID) of the bearing cover baseline.





When the MOVE command completes, our drawing should appear as above. We can quickly check that entities have been placed correctly by changing our viewpoint to Front again (View – 3DVIEW – Front) as pictured below:



# Chapter 2 Users Guide

## Configuration

### Hardware Configuration

#### Hardware Configuration Summary

##### Hardware Configuration:

Before you concern yourself too much with Program and Drawing configuration, you will want to first address your computer hardware setup. The areas of concern include:

1. Video Resolution Settings
2. Printer Setup
3. Tablet Configuration

##### See Also:

##### Program Configuration:

1. Program Configuration
  - Default Open and Save File Formats / AutoSave
  - Default Program Support Directories
  - Alias Command Setup
  - Hotkey Command Setup
  - Program Color Setup - Drawing Background, Cursors, etc
2. Desktop Layout Configuration
3. Redefining Menus, Dialogs and Palettes
4. Redefining Toolbars (including Left Sidebar)
5. Changing Menus

##### Drawing Setup

1. SETUP – To specify feet and inches or decimal drawing units, angular measurement type (degrees/minutes/seconds, grads, radians), as well as angle zero orientation.
2. LIMITS – To set drawing boundaries.
3. SETTINGS – Setting drawing parameters and drawing mode configuration.
4. LOADLTYPE – Loading linetypes that you will be needing in the drawing
5. LAYER – This command calls the Layer Manager, which allows you to create new layers and setup or change layer parameters (default color, linetype, etc.).
6. FONT – This command helps you set up text styles (fonts and their characteristics) for use in the drawing.
7. DIMTYPE – This command helps you to set the various dimension parameters that you will want, when you dimension your drawing.

#### Video Resolution Settings

As with other programs, the settings that you have made for your video resolution affect both the size and clarity of your onscreen image. Ideally, you will want to set the resolution as high as possible to get the best definition, without making onscreen elements (icons, text, etc.) too small for comfortable viewing. Resolution is set and controlled by Windows, and users who are unfamiliar with changing these settings, should consult their Windows manual for further details.

Apart from the clarity and size of onscreen elements, the viewer's ability to see all of the AllenCad



## Printer Setup

**Command: PSETUP**

**Alias PS**

**Located:**

- 1. Pull-down Menu: File – Printer Setup**
- 2. Or Enter Command or Alias at the Command Line**

The command PSETUP (Printer/ plotter Setup) or the selection of the option **Printer Setup...** from the *File* menu will start the selection and installation of the desired printer or plotter.

This procedure for installing printers calls up the standard Windows printer/plotter installation sequences.

The user will find detailed instructions regarding the operating mode of the specific options for the printer installation and for installing and connecting a printer or plotter in the Windows Manual or in the output device manual. Only the most important steps for printer/plotter installation are reviewed in this text.

The command will open a dialog box.

### Dialog Control Elements

#### Printer

In the field Printer/ Plotter the user will specify the printer/ plotter or plotter to be used in the output of the drawing. Select either one of the standard devices defined in the Dialog Box control, or select another printer or plotter that is available under Windows. Available output devices are displayed in a list in the selection area under the heading **Special Printer**. To see the complete list, select the arrow at the right edge of the window. Select from the list of the installed output devices the desired printer.

#### Orientation

Select the format **Portrait** or **Landscape** for the desired alignment of the paper.

#### Paper

The size of the paper as well as the feed from a particular printer or plotter is controlled from the field **Paper**. Select the proper size and feed.

#### Options

The **Options** switch (at the right edge of the Dialog Box) opens another Dialog Box that contains parameters that are mainly a function of the capability of the device to be connected.

### Drawing Sheet Reference

The drawing sheet sizes that you can use are determined by the paper size limitations that are specified in your printer or plotter documentation. To determine the size of your drawing, you must also consult this documentation to determine your printer's "clip limits". Generally speaking, printers can only print to within a certain distance from the edge of the paper. The "margins" around the edges of the drawing sheet that cannot be printed on are referred to as the "clip limits" and these vary according to the specific printer being used. The area inside the clip limits is commonly referred to as the "drawing frame". Always ensure that your drawing lies within the dimensions of the drawing frame.

There are two different sets of drawing sheet standards. The American Standards Association recognizes the following standard drawing sheet sizes:

Sheet Ref.	Size (Inches)	Size (Millimeters)
------------	---------------	--------------------

"A"	8 1/2" x 11"	215 mm x 279mm
"B"	11" x 17"	279 mm x 431mm
"C"	17" x 22"	431mm x 558mm
"D"	22" x 34"	558mm x 862mm
"E"	34" x 44"	862mm x 1116mm

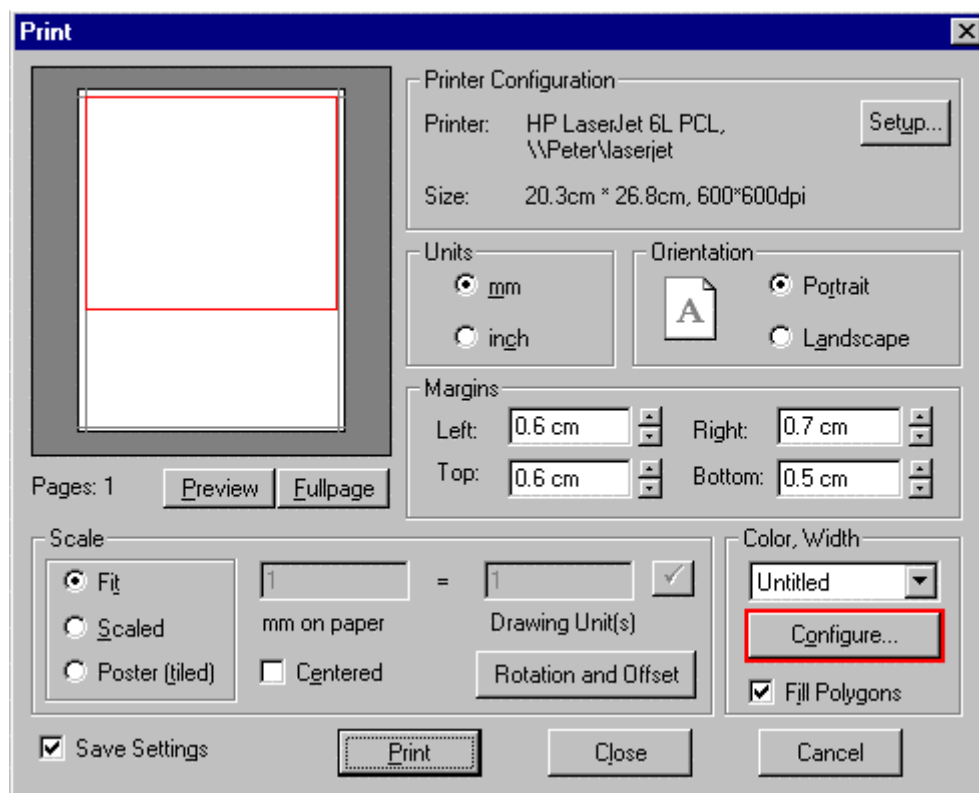
Another common paper size is the "Legal" sheet, that is 8 1/2" x 14" (215mm x 355mm).

The other commonly used set of drawing sheet standards are the ISO (Metric) standards. These include:

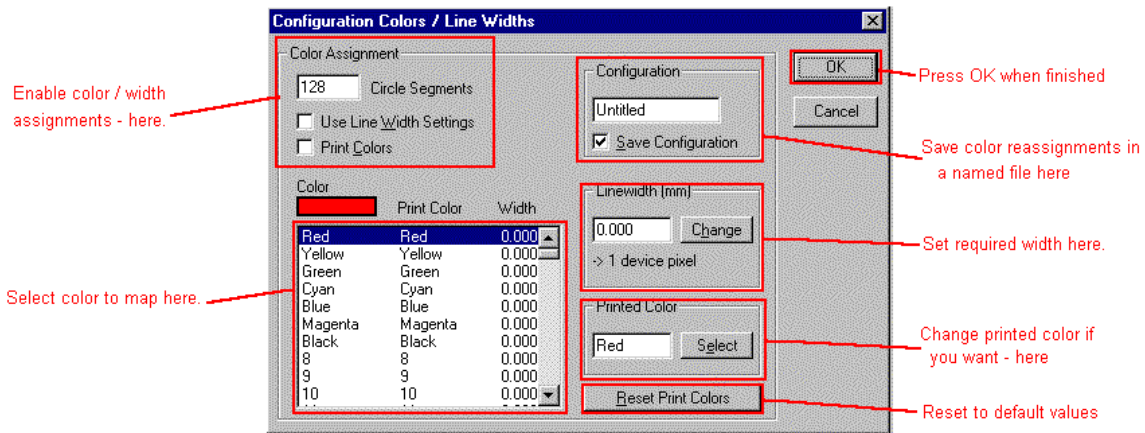
Sheet Ref.	Size (Inches)	Size (Millimeters)
"A0"	33.11" x 48.81"	841mm x 1189mm
"A1"	23.39" x 33.11"	594 mm x 841mm
"A2"	16.54" x 23.39"	420mm x 594mm
"A3"	11.69" x 16.54"	297mm x 420mm
"A4"	8.27" x 11.69"	210mm x 297mm

## Print: Color / Width Configuration Dialog

*Area Color / Width* defines the widths of the stroke for the output of the printer/plotter or plotter. This choice of line widths permits the structuring of a drawing and improving its legibility. You set the color - line width associations by clicking on the Configure button on the PRINT command dialog.



You will then be presented with the following dialog:



The definition of stroke width is a combination of width and color (or color identification number). The current setting of stroke widths can be saved under a name and then, like a parameter, used again for future printing operations.

Every object has been assigned a color that is defined by a color identification number (also called color index) already. For detailed instructions on the definition of colors for layers or for objects, see the Help section "Layer and Object Properties".

During the preparation of the print output, any defined color can be given a stroke width defined between 0 and 50 mm.

Select the area *Color / Width* of the Dialog Box *Print* and then select *Config*. The dialog box containing *Color/Width Configurations* will appear.

## Switching Line Width Selection On/ Off

Independent of the correlation of color identification numbers and stroke widths, the option to output different line widths can be switched on or off. Select the control field *Use Thickness Settings*. A cross in the control field displays the active status of the option.

There is a second control field with the name *Print Colors*. This field will switch the color printing On or Off. Of course, the output device must have color capability to print in color.

To activate or de-activate the option *Print Colors* select the control field. A cross in the control field displays the active status of the option.

## Attach stroke widths

Use the selection area for coordinating color and width.

Select the desired color or its color index (color identification number). In the selecting window the color identification number and the present stroke width is displayed. In the preview window *Color Index* the user can preview the selected color.

Enter the desired value in the input field *Stroke Width*. A range from 0 to 50 mm is possible. Attach the entered stroke width to the previously selected color by pressing the key *Change*.

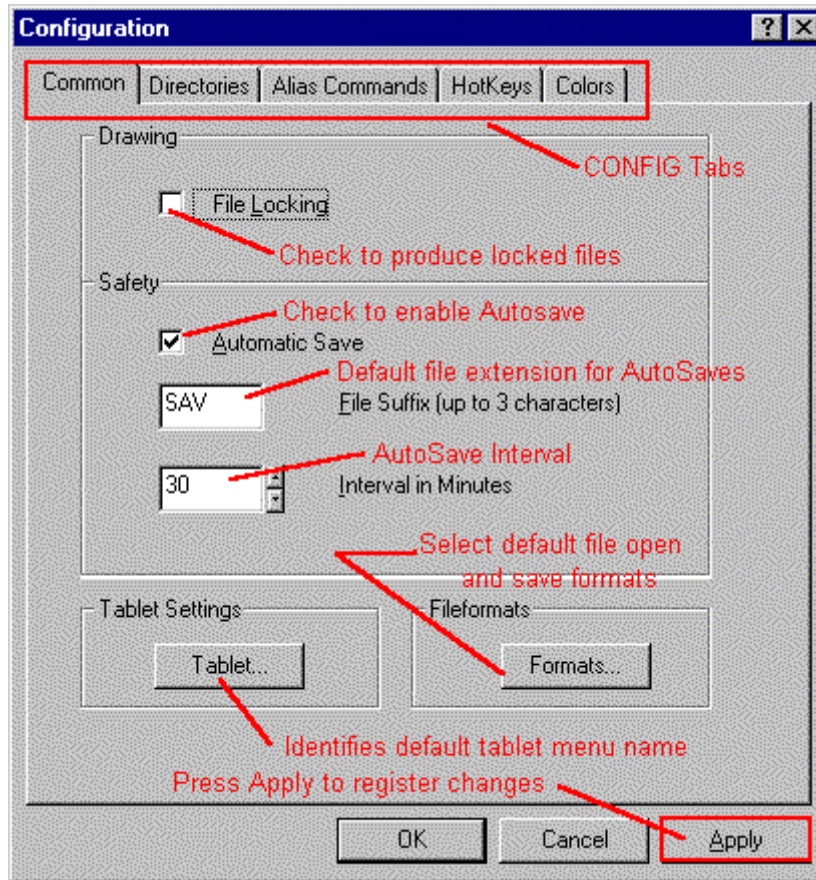
## Save Assignments

The program allows the user to save the correlation between line widths and colors.

To do this, enter a name in the area *Configuration* for the current setting of the stroke widths in the input field there. Switch on the function *Save Configuration* by selecting that control field. A cross in the control field displays the active status of the option.

Confirm the settings for the stroke widths by OK.

### Configuration: Tablet



By choosing the Tablet... button from the *Common* tab in the *Configuration* dialog, a dialog will appear. You may define the tablet menu file name for auto loading. *Start tablet menu automatically* can be switched on or off.

**See also:**

Calibrating the Tablet  
Loading Tablet Menu  
The TABSECTION command

## Program Configuration

### Program Configuration Summary

**Program Configuration:**

After setting up your hardware to run the program (see below), you will want to address program configuration. These are configuration elements that will remain active while you move from drawing to drawing. These elements include:

1. Program Configuration
  - Default Open and Save File Formats / AutoSave
  - Default Program Support Directories
  - Alias Command Setup
  - Hotkey Command Setup
  - Program Color Setup - Drawing Background, Cursors, etc
2. Desktop Layout Configuration
3. Redefining Menus, Dialogs and Palettes
4. Redefining Toolbars (including Left Sidebar)
5. Changing Menus

**See Also:**

**Hardware Configuration:**

1. Video Resolution Settings
2. Printer Setup
3. Tablet Configuration

**Drawing Setup**

1. **SETUP** – To specify feet and inches or decimal drawing units, angular measurement type (degrees/minutes/seconds, grads, radians), as well as angle zero orientation.
2. **LIMITS** – To set drawing boundaries.
3. **SETTINGS** – Setting drawing parameters and drawing mode configuration.
4. **LOADLTYPE** – Loading linetypes that you will be needing in the drawing
5. **LAYER** – This command calls the Layer Manager, which allows you to create new layers and setup or change layer parameters (default color, linetype, etc.).
6. **FONT** – This command helps you set up text styles (fonts and their characteristics) for use in the drawing.
7. **DIMTYPE** – This command helps you to set the various dimension parameters that you will want, when you dimension your drawing.

## Configuration



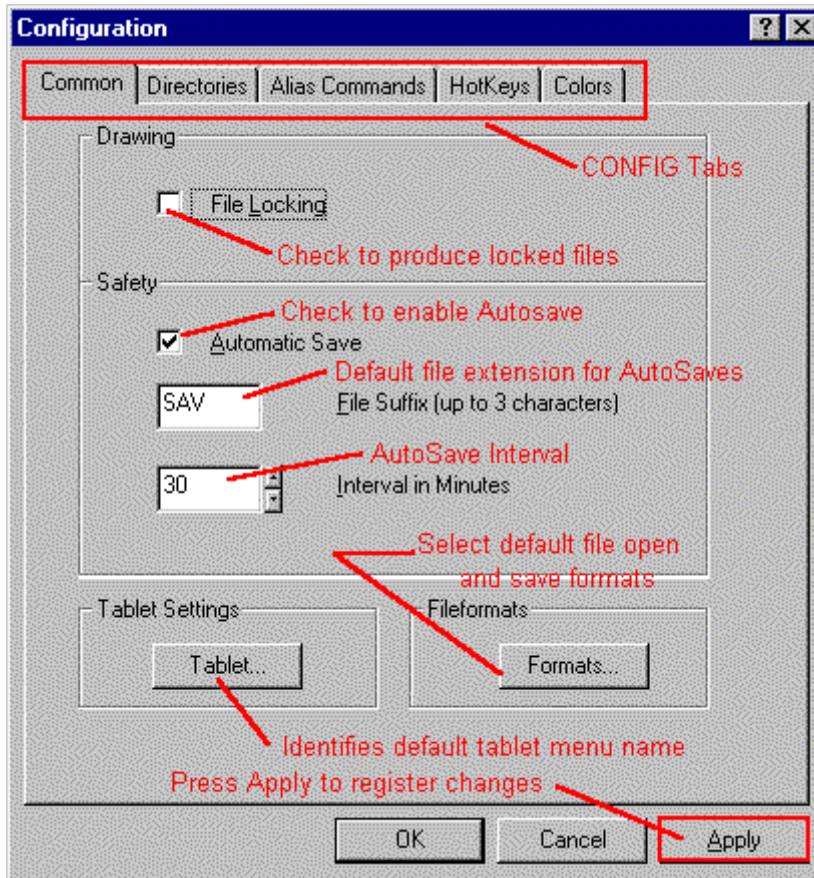
**Command: CONFIG**  
**Alias CFG**

**Located:**

1. **Pull-down Menu: Tools – Configure**
2. **Or Enter Command or Alias Directly at the Command Line**

By calling the CONFIG command or selecting Configure on the File pull-down menu, the Configuration dialog will appear.



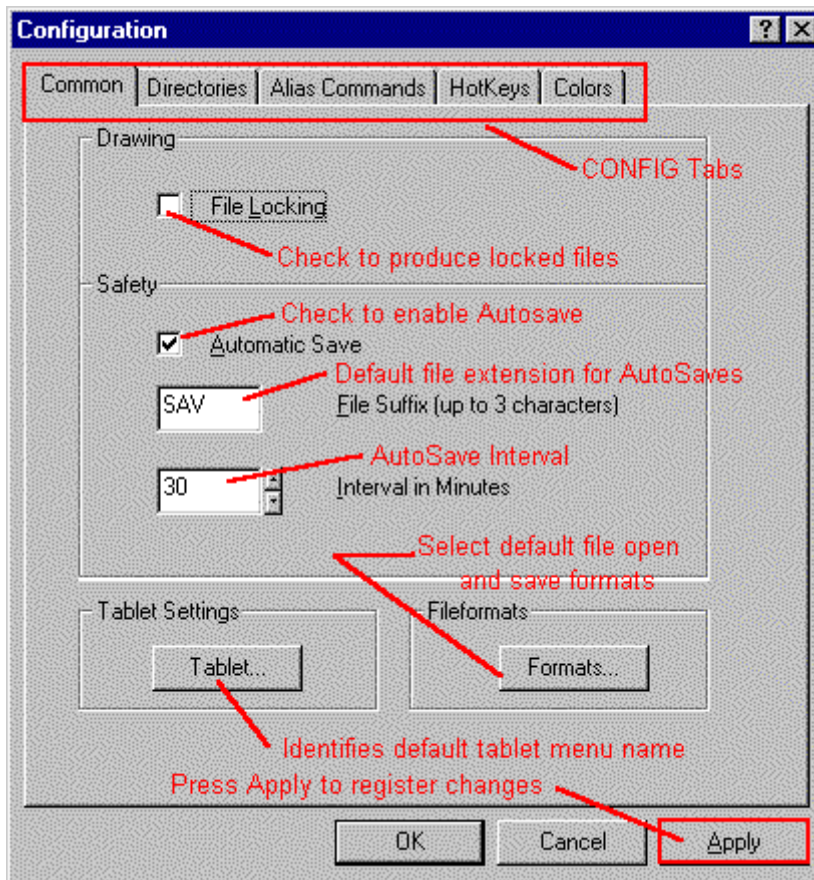


As can be seen, this dialog contains five tabs, each presenting separate dialogs under the headings: Common, Directories, Alias Commands, Hotkeys and Colors.

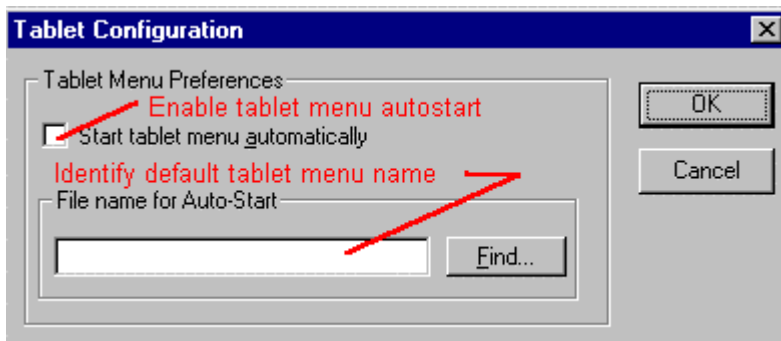
Note: Changes made in these dialogs will only become effective after the APPLY button has been selected. No changes will become effective if you close the dialog window by pressing OK, the Cancel button or ESC key.

### Config - Common

As can be seen below, the Common section enables file locking and configuration for Auto Saving your drawing file as you work. Lock files (with the extension .FLK) are backup files that are produced whenever you open a new drawing. These files are deleted whenever your drawing is correctly closed and are meant to ensure you do not lose your original file if there should be an unexpected hardware or software problem while running the program.

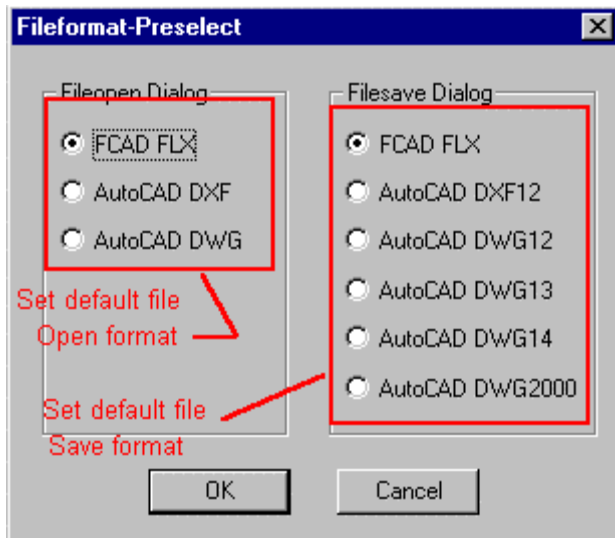


Below, a Tablet Settings button will display the following digitizing tablet settings dialog:



This dialog ensures that a named tablet configuration file is loaded on program startup. See also Command DIGICONFIG.

A second Button on the Common dialog, File Formats, presents you with the following dialog:

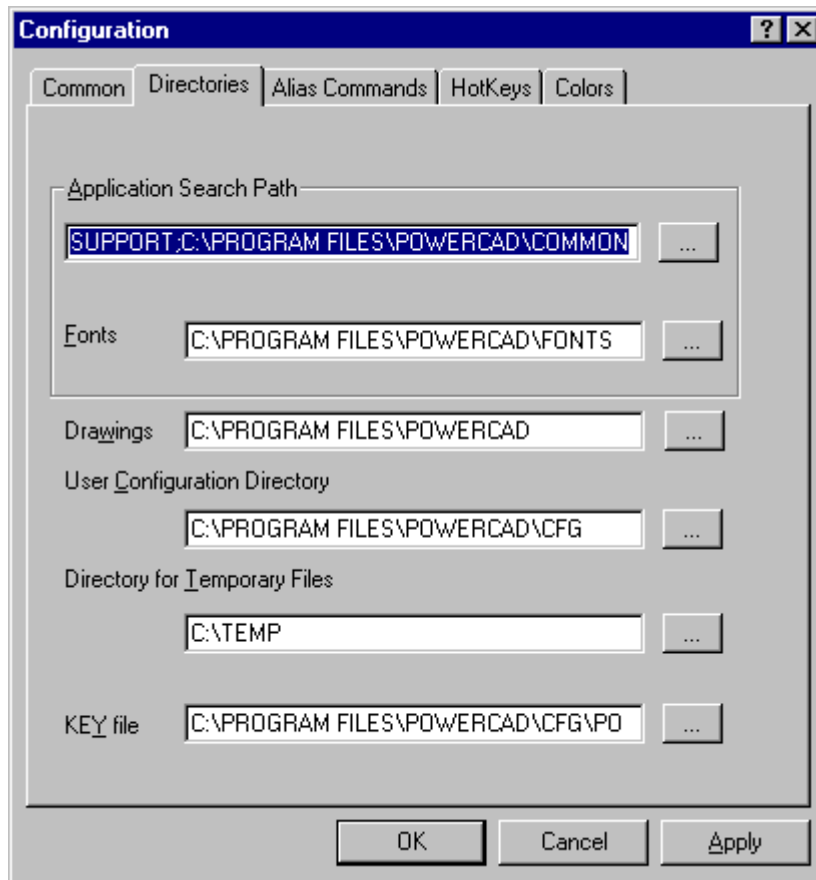


In this dialog, you can specify the **default file format** for the program to use when opening and closing drawings. Note that options are presented at these times allowing you to select options other than the defaults specified here.

## Config – Directories

The Configuration Directories dialog allows you to specify directories that the program will use when filing and retrieving its support files. In some instances, more than one directory can be entered for a specific file type. The order in which alternate file locations are entered here, establishes the search order that the program will use when retrieving needed files.

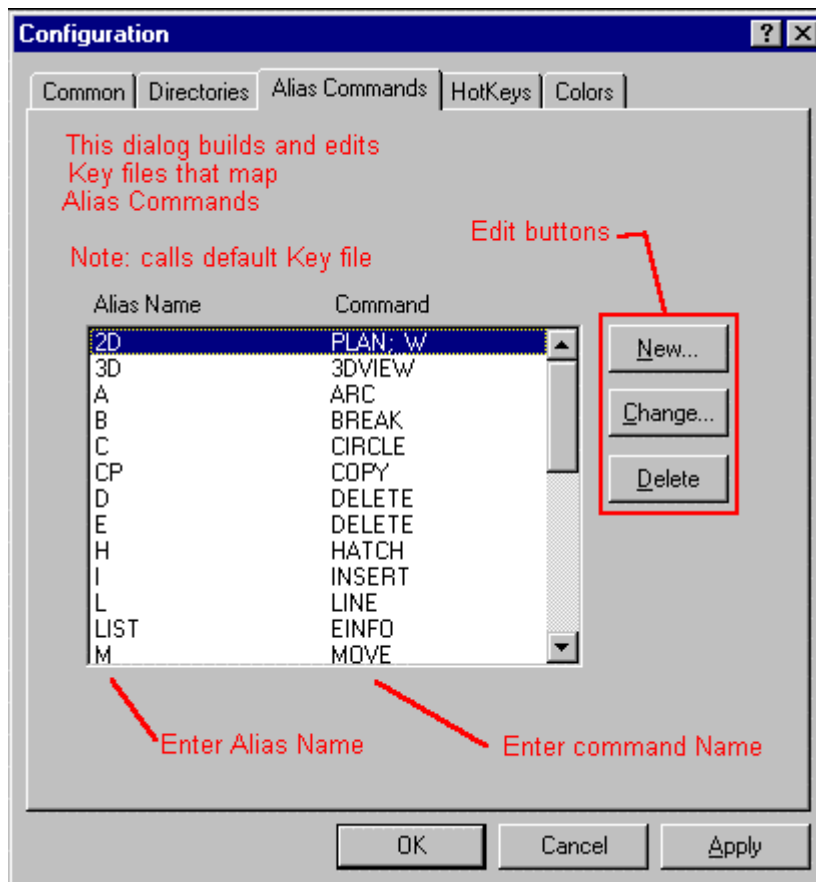
Note that apart from default directories that the program will use, this tab also names the default key file, that defined the alias commands that will be active in the running program.



### Config - Alias Commands

Many users prefer to use alias commands when drawing to speed up the drawing process. Alias commands are simply alternate command names that can be used to execute AllenCad commands. If you are familiar with using another CAD program, this capability also allows you to equate command names that you are familiar with to AllenCad functions.

In this dialog, you simply add new alias names to command functions by selecting the "New" button and entering new values. Similarly, "Change" and "Delete" buttons enable you to edit entries already made. These aliases are stored in a ".Key" file that is specified in the Configuration Directories Dialog.



Alias-commands can contain command entries that are not only made up of a command itself, but also corresponding command arguments, such as parameters. Alias-commands can be defined freely so that LINE 0,0 210,0 210,297 0,297\_C could be replaced with the alias command A4. Using this method it would no longer be necessary to input the parameters to draw an oblong to represent an A4 sheet of paper, the same result could be achieved by simply typing A4.

This method of operation not only increases the speed and efficiency with that the command is carried out, but also greatly reduces the risk of input errors.

Alias-commands can be defined by selecting the Alias-command option from the configuration window.

#### Dialog Box Control Elements:

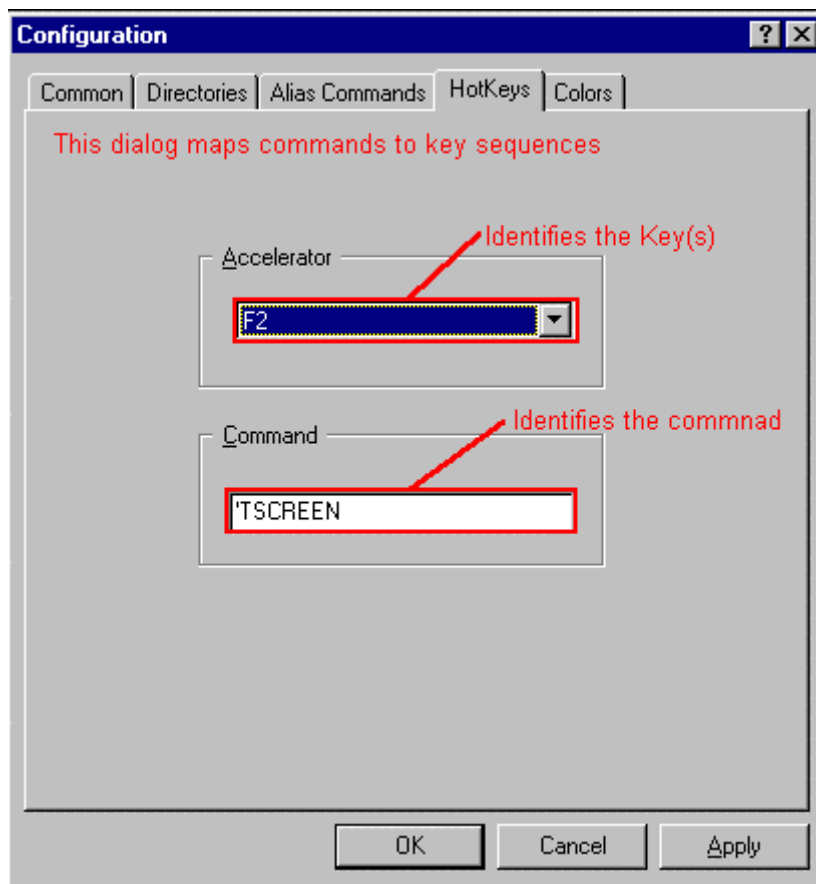
- New
- Change
- Delete

See Also: Transparent Alias Commands

## Config – Hotkeys

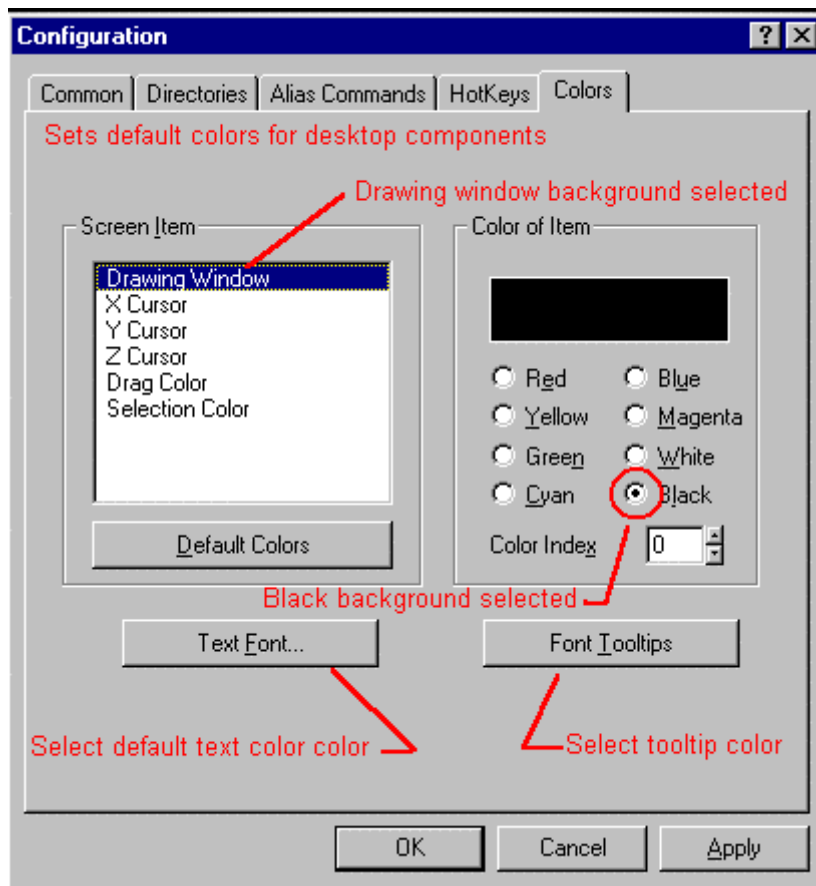
Like Alias commands, Hotkeys permit assigning commands to designated keyboard keys. These assignments are made using this dialog and are available for use whenever the program is running. To enter values, select the key to use (Function key 2 is shown in this example) in the Accelerator window, then enter the command name in the window below.

Note: Remember that context-sensitive Help uses the F1 key and that this key should normally be reserved for this use only.



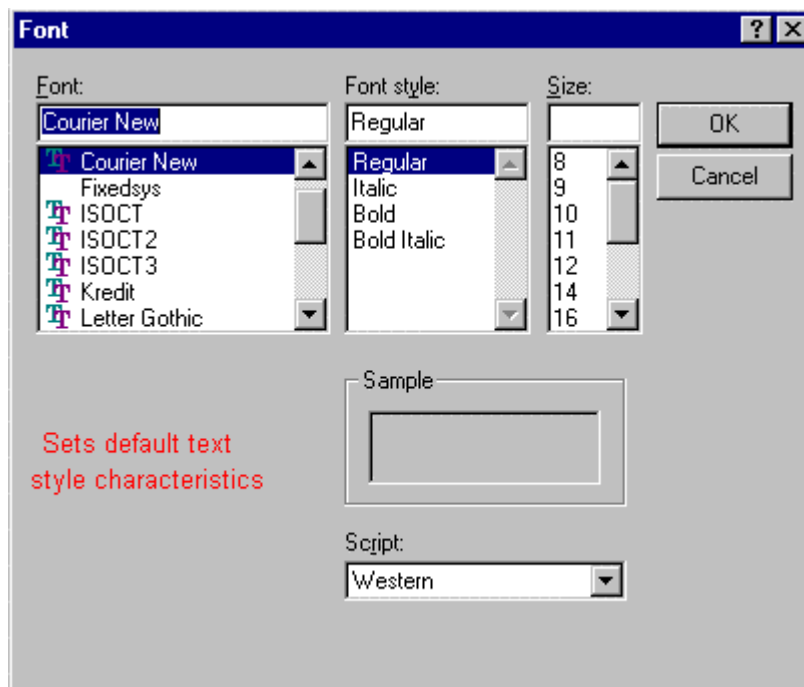
### Config – Colors

The Configuration - Colors dialog pictured below, simply allows you to change the colors of AllenCad drawing tools such as the cursors, etc.



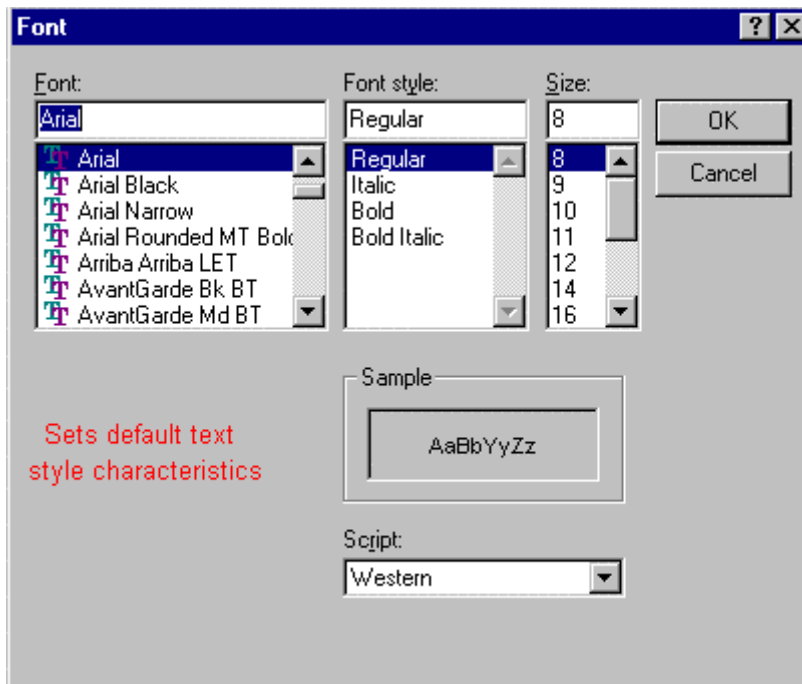
There are also buttons in the lower section of the dialog that permit changes to default values for Text fonts and Tooltips.

Pressing the Text Font button calls the following dialog:



In it, you can specify a different font and parameters for the program to use when it sends you messages.

Pressing the Font Tooltips button will call the dialog below:



Here you can simply specify the font and parameters that the program uses when presenting the user with tooltips that identify program buttons.

## Desktop Configuration

### Command DESKCONF

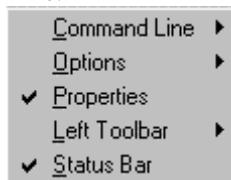
#### Alias DSK

##### Located:

1. Pull-down Menu: Tools – Desktop Manager
2. Or Enter Command or Alias Directly at the Command Line

## Modifying the Desktop Layout

The DESKCONF command's pop-up menu enables you to re-arrange the desktop to suit your needs. This pop-up menu allows you to set various layout options for the programs user interface, including the display of the Command Window, Options Menu or Options Bar, Property Bar, Left Sidebar, and Status Line.



### The Desktop Layout pop-up menu

**Caution!** The following sections describe how to turn off desktop elements. The user should exercise caution in doing so, however, as these actions will remove visual references to command inputs. If all desktop elements are turned off and the user wishes to reinstate them, the DESKCONF command will need to be retyped and executed with the relevant options selected. Pressing F2 will call the text window where the user can see the command prompt, as needed. Likewise, online Help can



be called as needed, by pressing the F1 key.

#### Turning desktop items on and off:

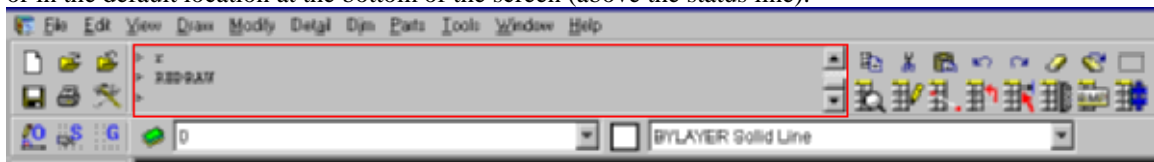
To achieve more space to display drawings on the desktop, or as an advanced user, who is familiar with the names and options of the commands, you may want to turn off certain items of the desktop. You can turn off and on (at runtime) the display of the command line, the Options Menu or context toolbar, the property bar, the left toolbar, and the status bar as the figure above shows.

#### How to open the *Desktop Configuration* pop-up menu:

You can call the *Desktop Configuration* menu by typing the DESKCONF command or selecting the Desktop Configuration selection on the Tools pull-down menu.

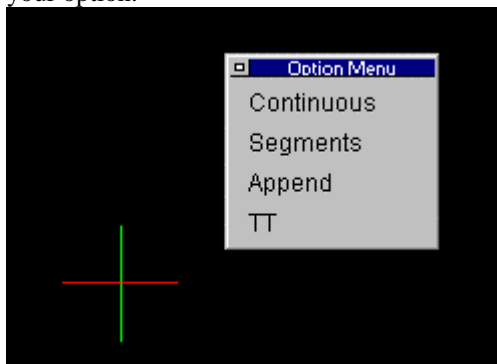
You can also open the *Desktop Configuration* pop-up menu by clicking the right mouse button in a "non-functional" area of the program's desktop. Such area is either the background area upon where the drawing viewports are displayed or areas in the property bar, Options Bar, or toolbars that are not covered by buttons or drop-down lists. If necessary, reduce the size of the current drawing(s), cascade the drawing(s), or minimize the drawing(s) to get some free space of the drawing area background. Right-click to this area to open the *Desktop Configuration* menu. You exit this pop-up menu by left clicking to some other area or component on the desktop.

The **Command Line** can either be placed at the top of the screen (below the menu bar as pictured below), or in the default location at the bottom of the screen (above the status line).



Where you place the bar is strictly a matter of personal preference, as there are no specific advantages in choosing either location. If you are an advanced user, who is familiar with the program's command set, you might even turn the command line off to provide a larger drawing edit window. In this instance, you can always check for program messages or inputs by toggling the text window on/off using the F2 key.

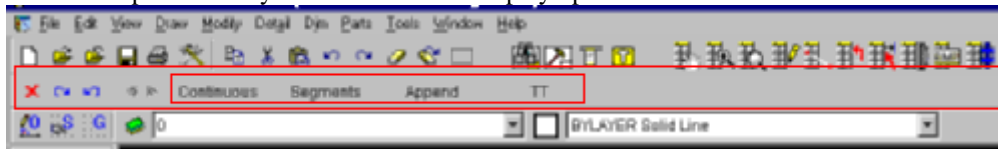
The items of the **Options** sub-menu allow you to determine whether you want to be prompted for command options via an *Options Dialog Menu* or within a *Context Toolbar*. By default, command options are displayed in a pop-up menu that appears in a dialog window in the edit window. These command option dialogs (as pictured below) will disappear or change to offer new options, once you have selected your option.



An Options Dialog Menu - The benefits to this option are that it places the options right on the editor where you are working, and that it doesn't reduce the overall size of your drawing edit window. The drawback is that you sometimes may have to move (drag and drop) the dialog if it is in your way as you draw.

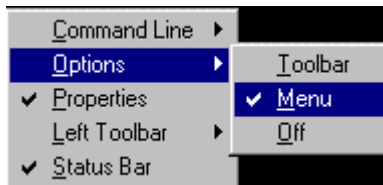
If you choose to display options on an *Options Bar*, you will see a new toolbar inserted between the

horizontal toolbar and the Properties Bar, as pictured below. This toolbar will remain open to display command options until you choose another display option.

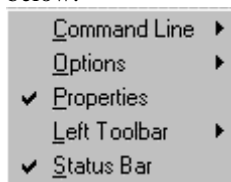


The Options Bar - Note the options presented on the bar.

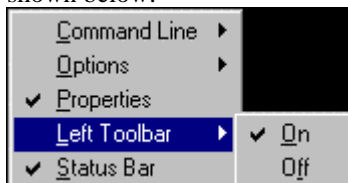
Note: You can recall the display of the context pop-up menu by selecting the option *Options>Menu* from the *Desktop Configuration* pop-up menu.



Both the **Properties Bar** and the **Status Bar** can be toggled off and on by clicking on the selection in the Desktop Configuration dialog. When toggled on, the selection will appear with a check mark as shown below:



The **Left Sidebar** can be switched off and on by clicking on the on or off options that appear in a secondary window when you click on the Left Toolbar option in the Desktop configuration dialog, as shown below:



See Also: Emulating the AutoCAD Desktop

## Modifying the Desktop On-The-Fly

At any time when AllenCad is running, you can change the elements of the user interface (called the *desktop*). All items of the desktop (except pull-down menus) can be turned on and off at run-time. The toolbars can be easily customized on the fly:

Simply drag and drop palette buttons to the toolbar.

Define or modify a toolbar button by clicking it with the right mouse button, which brings up the command configuration dialog box.

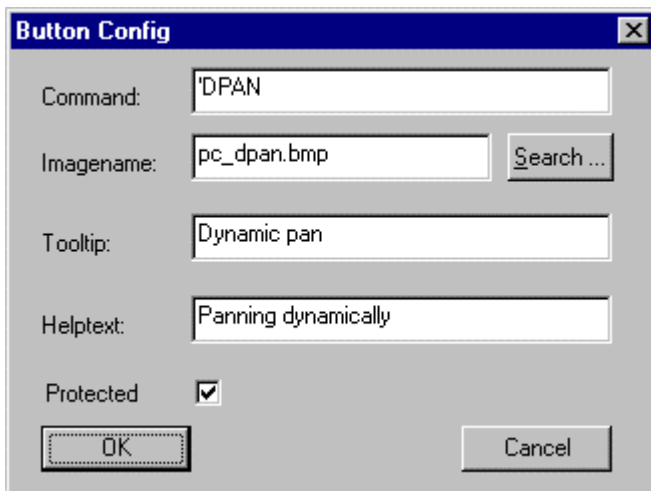
Icon menu support is included to allow users to visually select options.

## Reassigning Toolbar Commands/Icons

If you find you want to change the toolbars around to make them more functional for your particular purposes, this is easily done in AllenCad. Complete instructions may be found in the Customization and LISP Programming Guide, as well as the online Help that accompanies the Desktop and Menu Editor program.

Icons and command assignments for both the Horizontal Toolbar and Left Sidebar are stored in the desktop.cfg file, found in the AllenCad\CFG directory. You could edit this file with Windows' Notepad (or any other text editor), in order to change button assignments, however direct reassignment on the desktop using "drag and drop" is a much simpler and safer approach.

To change a command button location directly on the desktop, simply click on the icon with the left mouse button, keeping the button pressed until you reach the new location with the cursor. While moving the button, be sure to keep the Shift key depressed as well. If you don't hold the Shift key down, you will simply create a copy of the command button in the new location. To edit or assign new values to a button, simply right-click on the button to raise the following dialog, and then enter or edit the values in the dialog windows. Please refer to the Customization and LISP Programming Guide for directions on dialog entries.



### Notes:

1. To prevent accidentally losing command buttons, all buttons are "protected" by default. In order to release them for editing, right mouse click on the button and remove the check mark beside the "Protected" selection on the dialog that appears. After editing or moving, you may wish to reinstate button protection.
2. Command buttons can be copied from the palettes to the toolbars, but not vice-versa. A master palette containing all command buttons is supplied for this purpose and can be found on the Tools pull-down menu.
3. Menus, Palettes and program dialogs must all be modified using the Desktop and Menu Editor program.
4. A backup copy of the default Desktop.cfg named Desk.cfg has been provided in the CFG directory. This file can be copied and renamed Desktop.cfg to replace a scrambled desktop.cfg file without the need to reinstall the program
5. To clear a button, right click on it and erase all dialog entries.

## Desktop Layout

AllenCad Pro ships with an AutoCAD emulated desktop, as well as a AllenCad desktop. These desktops include all of the toolbar icons as well as toolbar visibility, that are default to each option. Each of these desktops can be further customized, if you wish, through the DESKCONF command, which controls bar visibility and AllenCad Pro further allows you to move and replace the command icons, through both drag and drop and dynamic button reassignment. You may also use these features to emulate other CAD desktops, or you create your own unique desktop layout, should you prefer.

Note: Both AutoCAD 15 emulation menu is also available in addition to the default AllenCad menu, through the MENU or PULLDOWN commands.

If you wish to change the desktop layout between AllenCad and AutoCAD emulation after installation, you will need to note the following:

The current desktop is defined by the desktop.ini file that is located in the program's cfg directory. Select the appropriate menu file using the MENU command from within the program. See the Help documentation on the command for further details on the use of this command.

Tool bars are available in 3 flavors (AC\_\*) for autocad tool bars, (PC\_ and FC\_) for AllenCAD tool bars.

## Desktop and Menu Editor

### Desktop and Menu Editor – Command DLGEDIT

#### Alias DM

##### Located:

1. **Pull-down Menu: Tools – Resource Manager – Run Desktop and Menu Editor**
2. **Or Entered Directly at the Command Line**

The Desktop and Menu Editor (DME) design and lay out your menus and palettes in a visual environment. Knowledge of programming is not required.

Please refer to the Command Button Chart for input information when modifying palettes.

For more information on the DME click here: DME Help 

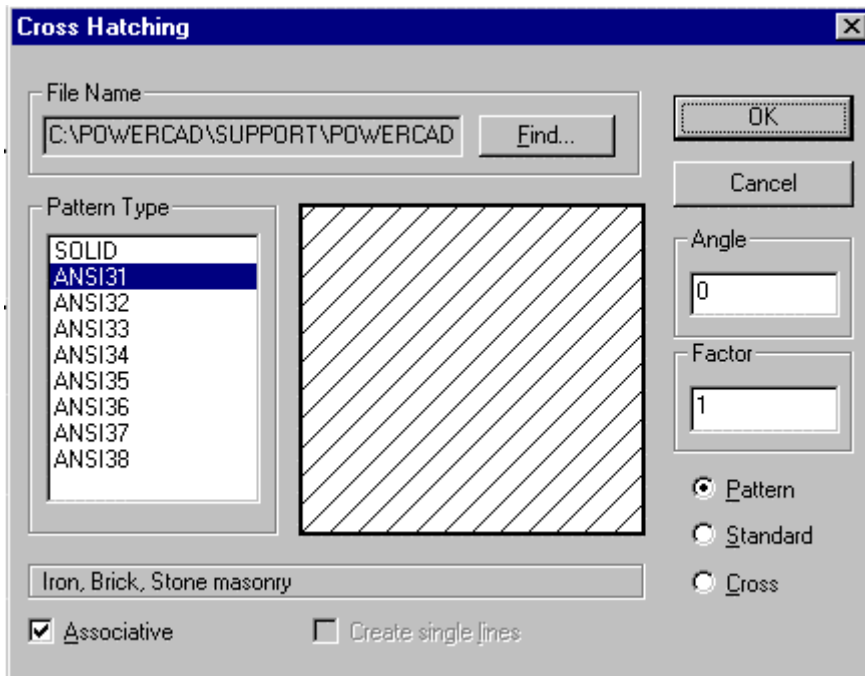
## Icon Menus

### Command ICONMENU

#### Located:

1. **Pull-down Menu: Tools – Resource Manager - Icon Menu...**

The Icon menu command is a programming tool that presents the user with selections that are supported by images. Icon menus use groups of images as graphic labels. Icon menus also have a list box containing text labels associated to each image. This combination allows the user to select options



Icon menus are a valuable visual tool, as in the example above, when selections of hatch patterns or text styles should be customized. Also, icon menus can be useful in creating symbol library selection menus.

With this command, a dialog presenting an assortment of graphical options can be created. A slider bar is added automatically to the dialog if the number of images exceeds display limits.

You can customize the number of columns and rows to be displayed for each icon menu.

## Palette Manager

**Command PALMAN**

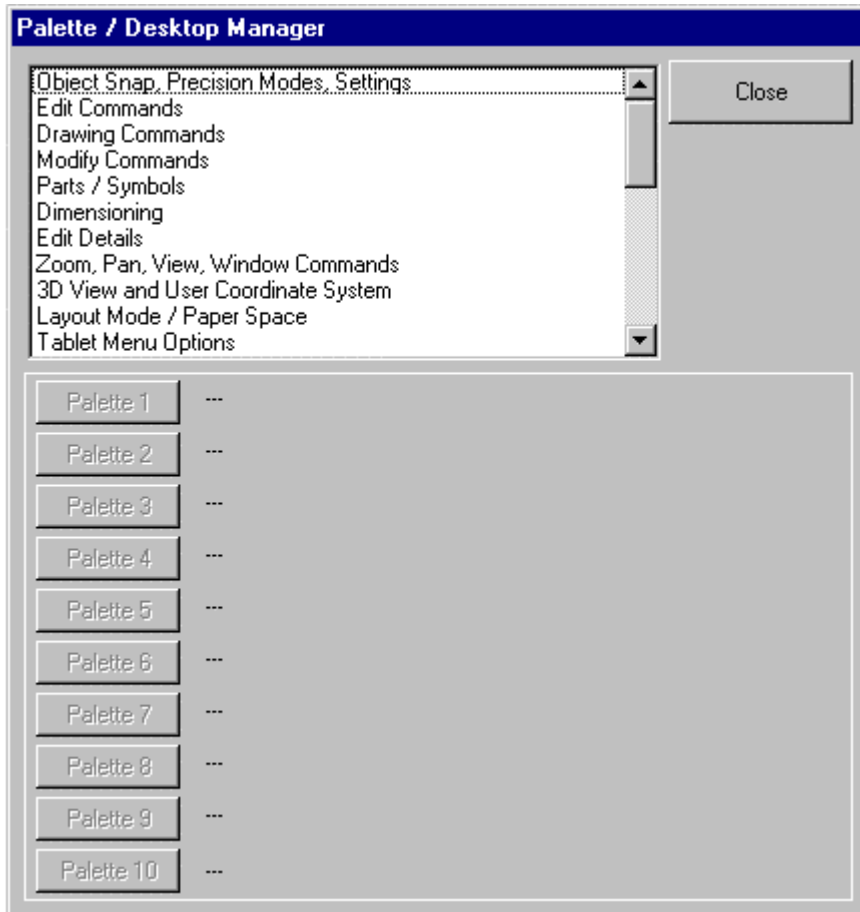
**Alias PM**

**Located:**

1. **Pull-down Menu: Tools – Palette Manager**
2. **Or Entered Directly at the Command Line**

Most users will elect to use the palette icons on the Toolbar to activate the various palettes, however others may prefer to select *Palette Manager* from the menu *File* and choose the action that they want in a more interactive fashion.

In the selection area of the Dialog Box, all of the palettes that are currently available are shown:



Select the desired palette. Underneath the selection area are three switches with the designations "Palette 1", "Palette 2", and so forth, to "Palette 10". By selecting one of the ten switches, the user can assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID-number already opened, this palette will be closed and replaced by the new palette having that ID number. When the palette is opened, the selecting window will close. If no palette is to be opened or replaced the user may close the window by selecting *Cancel*.

If the user wants to include customized palettes in the Palette Manager she/ he should proceed as follows: Edit the file *<Menu name>.pal* that is situated in the \SUPPORT directory. For instance, if the user menu is called *Pcad.mnu*, the user must edit the file *Pcad.pal*. The structure of the file is:

Palette description 1

palette1.mnp

Palette description 2

palette2.mnp

and so forth.

The palette description will be found in the list shown in the Palette Manager Dialog Box, and the line following will always name the palette file. The user can complete or delete the entries as he or she wishes.

### **PALETTE1 ... PALETTE<sub>n</sub>**

These variables return the filename of a palette number *n* currently loaded and displayed on the desktop. The variable returns in a string the filename including drive and path. If no palette has been loaded indexed as *n*, the variable returns a null string ("").

#### **Properties**

*Type*                      String

*Default*        ""  
*Status*        Read-Only  
*Example*       "C:\APX\AP\_DRAW.MNP"

#### **Commands**

PALETTE, PALMAN, MENU

### Turn Maximized Command Line Window On/Off

**Command:** 'TSCREENFULL

**Alias** TSC

#### **Located:**

1. **Entered Directly at the Command Line**

Toggles the maximized Command Line Window on or off.

Both, the commands TSCREEN and TSCREENFULL, allow you to display the command line area in an independent window. TSCREEN shows the text window that covers only parts of the screen, whereas TSCREENFULL shows the text window covering the entire display.

Drawing Navigator...



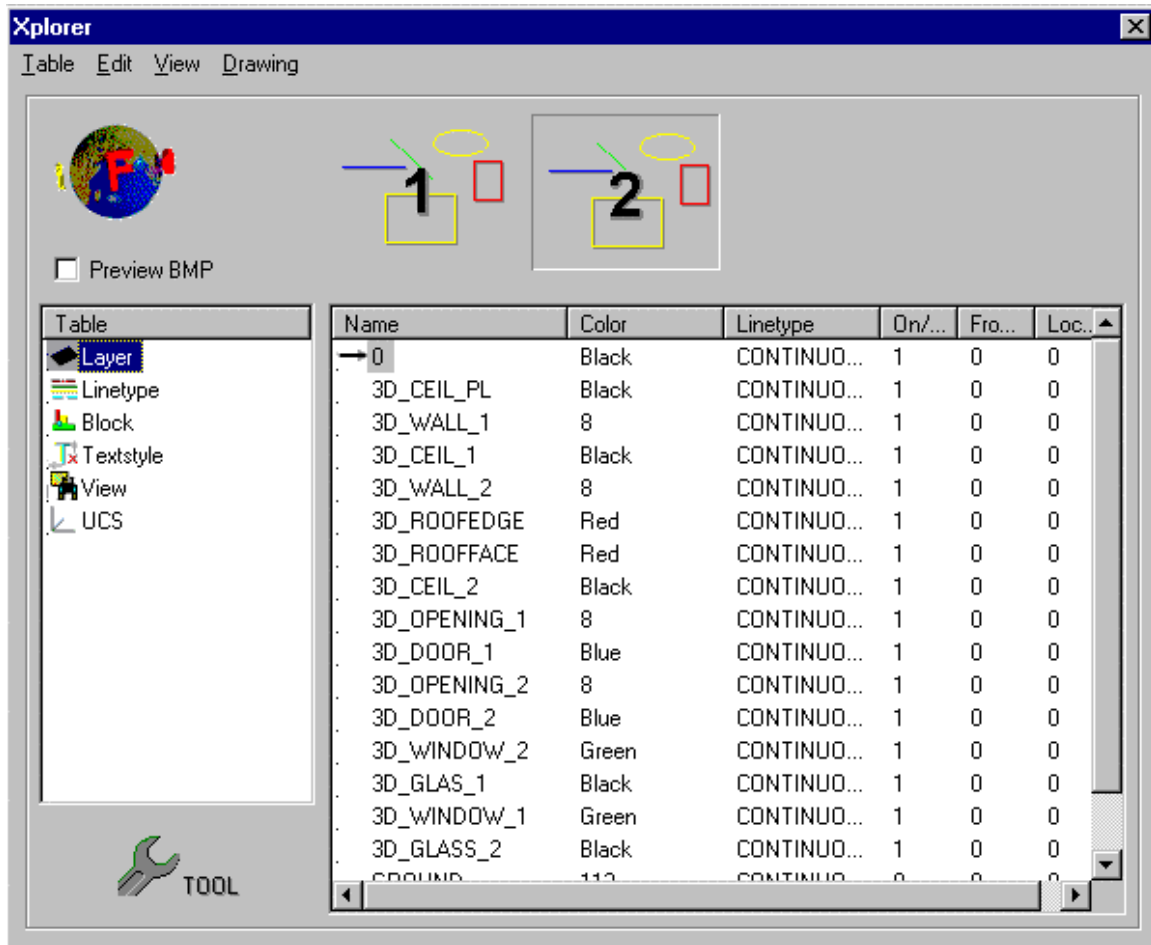
**Command:** XPLOERER

**Alias** XPL

#### **Located:**

1. **Pull-down Menu: File– Drawing Xplorer**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The Drawing Xplorer offers you fast and uncomplicated access to important information on the currently open drawing(s). Furthermore, you can use the Xplorer to alter various properties of the current drawings.



More precisely, the Drawing Xplorer is a tool to manage the named objects in the drawing database tables. With it, you can manage the tables and table entries of layers, linetypes, textstyles, views, user coordinate systems, and blocks.

Type in the command `XPLORER` to activate this tool or select *Drawing Xplorer* from the *File* menu.

The *Xplorer* window displays the preview bitmap of the currently open drawings in the area underneath the menu bar. You switch to another drawing table, click the appropriate bitmap. A ToolTip over the corresponding bitmap displays the drawing name including drive and path.

The left-hand list box titled *Table* serves to select and access a specific drawing database table. The table entries with all its named objects and characteristics will then be displayed in the right-hand list box.

In detail, the drawing Xplorer permits the following:

- Delete unused, not referenced table entries (allowed only for the active drawing)
- Rename table entries (allowed only for the active drawing)
- Set the current layer, current linetype, current textstyle, current user coordinate system, or current named view (allowed only for the active drawing) by double clicking to one of the table entries
- In case the *Block* table is the current one, you can insert a block by double clicking to its name.

Clicking the *Tool* button will invoke the main corresponding function associated to the active table.

For example, if *Layer* is the Drawing Navigators current table, you will call the Layer Manager dialog window by clicking the *Tool* button. This way, you might create new layer entries.



## THICKNESS

### System Variable - Thickness

#### Alias TH

##### Located:

1. **Pull-down Menu: Tools – Entity Extrusion - Set Thickness or No Thickness**
2. **Or Enter Command or Alias at the Command Line**

Sets the value for the default thickness (extrusion). Specifying the value 0.00 means that no thickness is applied.

#### Properties

<i>Type</i>	Real
<i>Range</i>	Any positive or negative number
<i>Default</i>	0.00
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### See also

System variable ELEVATION

## Turn Coordinate Symbol on/off

### Command: 'TICON

#### Alias TI

##### Located:

1. **Entered Directly at the Command Line**

The transparent command TICON toggles the Coordinate Icon on or off.

## Load Menu



### Command: 'MENU

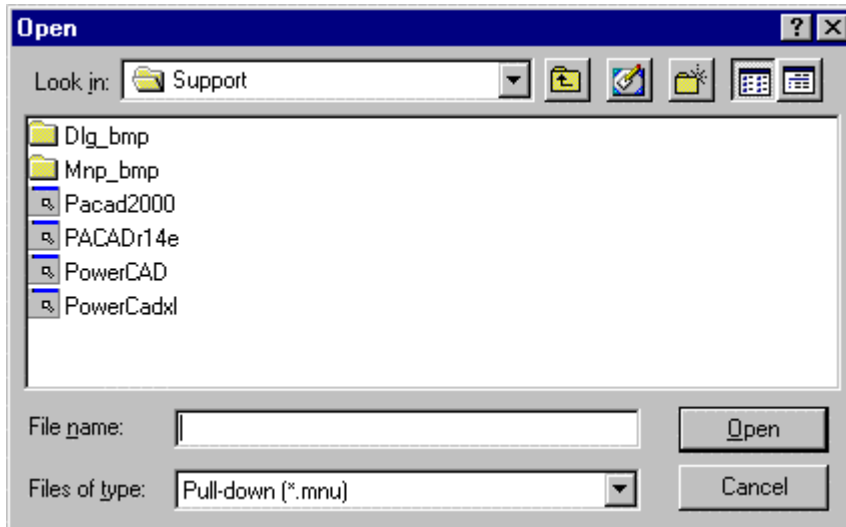
#### Alias MU

##### Located:

1. **Pull-down Menu: Tools – Resource Manager – Pull-down Menu**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The program is designed so that it is possible to work with different pull-down menus. AllenCad comes complete with both its own AllenCad menu, as well as AutoCAD and FelixCAD emulation menus. Although the AllenCad menu offers the greatest functionality, users more familiar with other CAD programs may prefer to use the emulation menus provided.

The pull-down menus are stored in menu files with the extension **.mnu**. To switch to using another stored menu file, simply execute the MENU command and select the file from the listing.



### Template Drawings

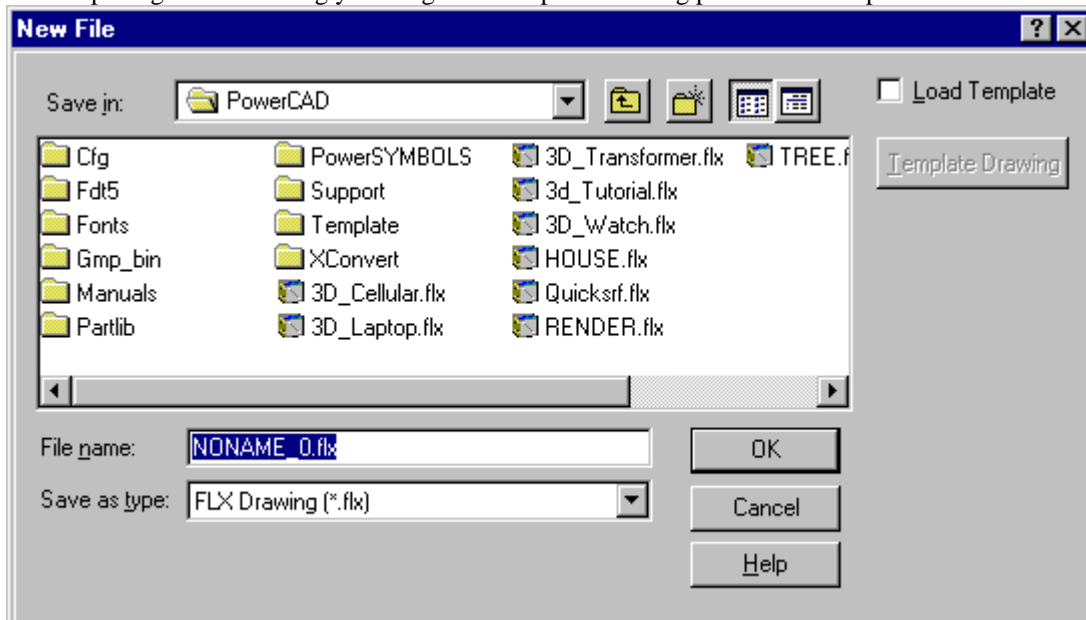
#### Command TEMPLATE

#### Alias TP

#### Located:

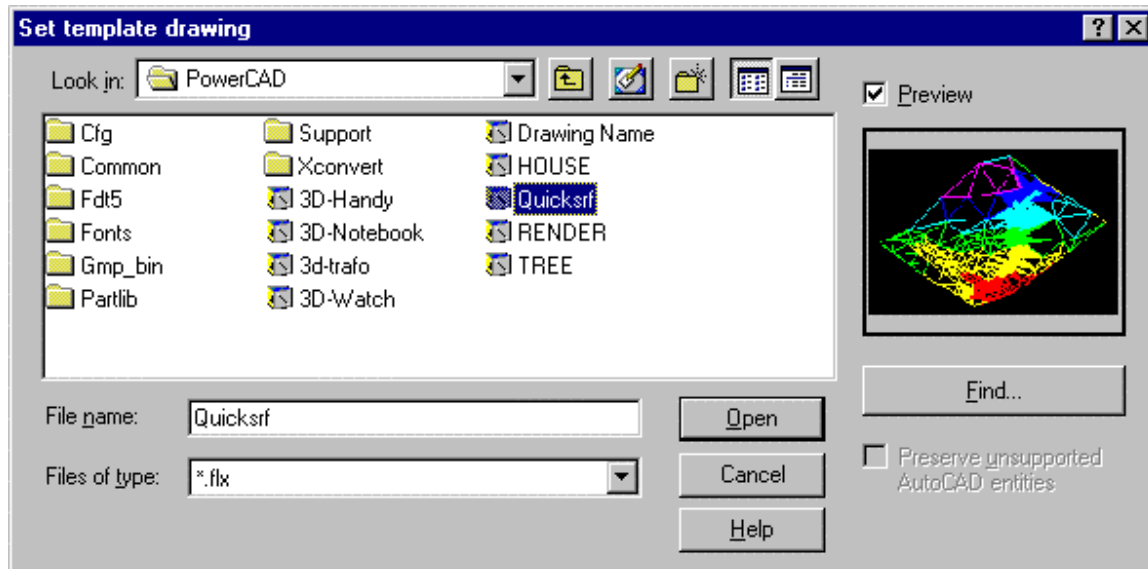
1. Pull-down Menu: Tools – Resource Manager – Set Template Drawing
2. Or Entered Directly at the Command Line

When opening a new drawing you are given the option of using pre-defined templates.



This is the standard file OPEN dialog, where you name your new drawing or choose to over-write an existing drawing with a new blank file. In the top right corner, you see the option to Load Template. If selected, this will load the default template drawing into your new file. If you wish to load another drawing as template, (with Load Template selected) select the Template Drawing button and browse to select another drawing file as template.

To set a new default template drawing, however, you must use the TEMPLATE command from within the program. All subsequent new drawing files will have the option of using this new template that you have just made default.



When you issue the **TEMPLATE** command, you will be presented with the dialog shown above. You simply navigate to the drawing you want as template, select it and press the **Open** button. That drawing will then be the new default template drawing until changed, using this same procedure.

#### Notes On Templates:

1. Templates are a form of specimen drawing that either act as a basis for your finished work, or set up the required drawing environment.
2. The main advantage of using templates is that all elements and settings of an already existing drawing can be adopted and as such do not need to be re-entered.

This is particularly useful for parameters such as drawing boundaries, frames, guidelines, measurement settings, views etc. In order to facilitate this, a number of local variables containing stored settings are loaded along with the template. These settings are then used to display the drawing, drawing elements and objects.

For a listing of commands that can be used in template drawing setup, please refer to **Drawing Setup**. Alternatively, if you have a drawing file that has all your standard setup elements in it, you can simply delete any drawing elements you do not wish to have in your template drawing and **SAVEAS** a new drawing file, to be used as the template.

Further information on the use of template drawings can be found in Chapter 4 of the **Customization and Programmer's Guide**.

## Accelerator Key Assignments

As with AutoCAD, AllenCad allows you to map "hot key" assignments to key sequences. These key sequences will save you a great deal of time when using commands repetitively. When you install AllenCad out of the box, the function key mapping is similar – but not identical — to that of AutoCAD. The following table illustrates the differences:

#### AllenCad's Default Function Key Assignments

Function Key	AllenCad Command	Comments
F1	Help	Defined by Windows.
F2	'Tscreen	Flipscreen
F3	...	Not defined.
F4	...	Not defined.

F5	...	Not defined.
F6	...	Not defined.
F7	'Tgrid	Grid toggle.
F8	'Tortho	Ortho toggle.
F9	'Tsnap	Snap toggle.
F10	Menu bar.	Defined by Windows

For function key assignments identical to AutoCAD, use the Config command to display the Configuration dialog box; select the Hotkeys tab. Here you define function keys and other accelerator keys, such as the Ctrl+Alt combination. Function and control key definitions are stored in the same .key file as command aliases.

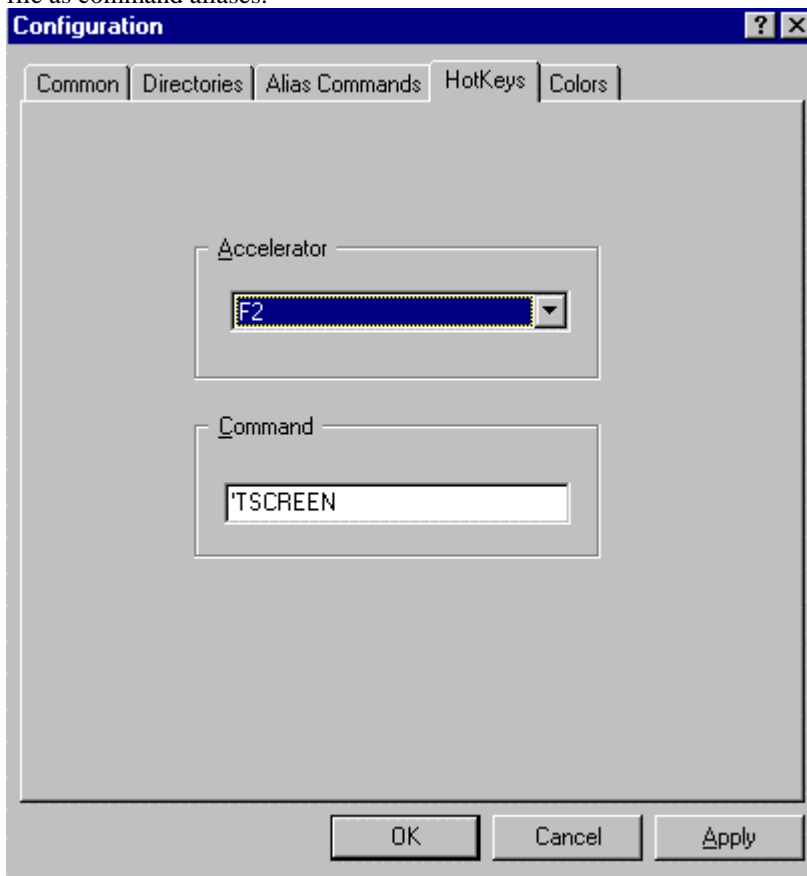


Figure 2.3: The dialog box for creating and editing function and accelerator keys.

AllenCad defines several Ctrl keys in the same manner as AutoCAD. The following table lists the default control-key assignments.

#### AllenCad's Default Control-Key Assignments

Control Key	AllenCad Command	Comments
Ctrl+B	'Tsnap	Snap toggle.
Ctrl+C	Copy to Clipboard.	Defined by Windows.
Ctrl+G	'Tgrid	Grid toggle.
Ctrl+L	'Tortho	Ortho toggle.

Ctrl+N	^^New	Open a new drawing.
Ctrl+O	^^Open	Open an existing drawing.
Ctrl+P	^^Print	Print (or plot) the drawing.
Ctrl+S	^^Save	Save the drawing.
Ctrl+V	Paste from Clipboard.	Defined by Windows.
Ctrl+X	Cut to Clipboard.	Defined by Windows.
Ctrl+Y	^^Redo	Redo the undo command.
Ctrl+Z	^^U	Undo the previous command.

*Note:* The apostrophe ( ' ) prefix indicates a transparent command. The double caret (^^) is the Esc key, which cancels the currently active command.

### Command Button Chart (Reference)

The following chart should be helpful when creating or modifying palettes or toolbars:

<b>PREFIX</b>	<b>Command</b>	<b>Icon</b>	<b>Key Word</b>	<b>Status Bar</b>
^^	(setvar "TILEMODE" 0)(command "_VIEWPORT" "_V")	fc_vpvw .bmp	Layout Mode Viewport Control	Layout Mode Viewport Control
^^	(setvar "TILEMODE" 0)(command "_VIEWPORT")	fc_vport. bmp	Create or Modify Layout Viewport	Create or Modify Layout Viewport
^^	(setvar "TILEMODE" 1)(command "_ARENDER")	fc_arend .bmp	AutoRender	AutoRender
^^	2DFACE	fc_2dfac .bmp	2DFACE	Draw filled plan faces
^^	3DFACE	fc_3dfac .bmp	3DFACE	Draw 3D faces
^^	3DVIEW	fc_3vie w.bmp	3DVIEW	Set 3D Viewpoint for Current Viewport
^^	ARC	fc_arc.b mp	ARC	Draw an arc
^^	AREA	FC_AR EA.BM P	AREA	Calculate area / circumference
^^	ARRAY C	fc_arrcr. bmp	ARRAY C	Circular Array: Copies rotated
^^	ARRAY I	fc_arr_i. bmp	ARRAY I	Individual Array
^^	ARRAY N	fc_arrcn. bmp	ARRAY N	Circular Array: Copies not rotated

^^	ARRAY R	fc_arr_r. bmp	ARRAY R	Rectangular Array
^^	ATTDEF	FC_ATT DE.BM P	ATTDEF	Define Attribute of part
^^	ATTEDIT	FC_ATT ED.BM P	ATTEDIT	Modify attribute location / value
^^	ATTEXP	FC_ATT XP.BMP	ATTEXP	Write attribute information to file
^^	ATTXEDIT	FC_ATT XE.BM P	ATTXEDIT	Edit Attribute infos of a part
^^	BHATCH	PC_BH AT.BM P	BHATCH	Apply Boundary Cross-Hatching
^^	BMPOUT	FC_BM POU.B MP	BMPOUT	Create bitmap file from drawing
^^	BPOLY	PC_BPO LY.BM P	BPOLY	Apply Bondary Polyline
^^	CALLOUT	Pc_callo ut.bmp	CALLOUT	Insert text box
#_	Cen	fc_s_cen .bmp	Cen	Snap center of circle or arc
^^	CHAIN	fc_chain. bmp	CHAIN	Draw a 2D polyline chain
^^	CHAMFER	fc_cham f.bmp	CHAMFER	Chamfering two entities
^^	CHANGE	PC_CH ANG.B MP	CHANGE	Change Object Attributes Or Location
^^	CIRCLE	fc_circ.b mp	CIRCLE	Draw circle(s)
^^	CLOSE	fc_f_cls. bmp	CLOSE	Close current drawing
^^	CONFIG	fc_f_cfg. bmp	CONFIG	General settings for the program
^^	COPY	fc_copy. bmp	COPY	Copy selected objects
^^	COPYPROP	FC_PRO P.BMP	COPYPROP	Copy properties to other objects
^^	COPYSELECT	FC_CLC P.BMP	COPYSELECT	Copy selected objects to clipboard
^^	CUTSELECT	fc_cut.b mp	CUTSELECT	Cut selection set to clipboard
^^	DELETE	FC_DEL .BMP	DELETE	Erase drawing objects
^^	DELPARTIAL	fc_break .bmp	DELPARTIAL	Delete partial segments of entities
^^	DIMA3P	FC_D_A N3.BMP	DIMA3P	Angular dimensioning (defined by 3 points)
^^	DIMA4P	FC_D_A N4.BMP	DIMA4P	Angular dimensioning (defined by 4 points)
^^	DIMALI	FC_D_A	DIMALI	Aligned linear dimensioning

^^	DIMBAS	LI.BMP FC_D_B	DIMBAS	Baseline dimensioning
^^	DIMCENT	AS.BMP FC_D_C	DIMCENT	Center marking
^^	DIMCON	EN.BM P FC_D_C	DIMCON	Continuing dimensioning
^^	DIMDIA	ON.BM P FC_D_D	DIMDIA	Diameter dimensioning
^^	DIMDRAW	IA.BMP FC_DIM	DIMDRAW	Create dimensions
^^	DIMEDIT	.BMP FC_EDI	DIMEDIT	Edit dimensions
^^	DIMHOR	M.BMP FC_D_H	DIMHOR	Horizontal dimensioning
^^	DIMLIN	OR.BM P FC_D_L	DIMLIN	Linear dimensioning
^^	DIMOBL	IN.BMP FC_D_O	DIMOBL	Modify Dimension Orientation
^^	DIMORD	BL.BMP FC_D_O	DIMORD	Ordinate dimensioning
^^	DIMRAD	RD.BM P FC_D_R	DIMRAD	Radius dimensioning
^^	DIMREST	AD.BM P PC_DRS	DIMREST	Restore Dim Style
^^	DIMROT	T.BMP FC_D_R	DIMROT	Rotated linear dimensioning
^^	DIMSAVE	OT.BM P PC_DS	DIMSAVE	Save Dim Style
^^	DIMSER	AVE.B MP FC_D_S	DIMSER	Serial dimensioning
^^	DIMTHOME	ER.BMP FC_D_T	DIMTHOME	Realign Dim Text
^^	DIMTMOVE	HO.BM P FC_D_T	DIMTMOVE	Move Dim Text
^^	DIMTNEW	MO.BM P PC_DT	DIMTNEW	Modify Dim Text Value
^^	DIMTROT	NW.BM P FC_D_T	DIMTROT	Rotate Dim Text
^^	DIMTYPE	RO.BM P FC_PRE	DIMTYPE	Set or modify dimension style
^^	DIMUPD	F.BMP FC_D_U	DIMUPD	Update Dimensions
^^	DIMVER	PD.BMP FC_D_V	DIMVER	Vertical dimensioning
		ER.BMP		

'	DIST	FC_DIS T.BMP	DIST	Measure distance / angle
^^	DPAN	PC_DP AN.BM P	DPAN	Dynamic Pan
^^	DTEXT	PC_DTE XT.BM P	DTEXT	Direct Text Entry
^^	DZOOM	FC_DZ OOM.B MP	DZOOM	Dynamic Zoom
'	EDITPAR	FC_PRE F.BMP	EDITPAR	Parameters for modify commands
^^	EINFO	FC_OIN FO.BMP	EINFO	Info on selected objects
^^	ELLIPSE	FC_ELL IP.BMP	ELLIPSE	Draw an ellipse
#_	end	fc_s_end .bmp	end	Snap end point
^^	EXPAND	fc_expan .bmp	EXPAND	Extend entities to boundary edge
^^	FILLET	fc_fille.b mp	FILLET	Filletting two entities
^^	FLIP	fc_mirnc .bmp	FLIP	Mirror objects (original deleted)
^^	FONT	FC_FO NT.BM P	FONT	Manage Text Styles
^^	FREEHAND	Pc_freeh and.bmp	FREEHAND	Freehand line and text sketching
^^	HATCH	fc_hatch. bmp	HATCH	Create cross-hatching
^^	HPEDIT	FC_HE DIT.BM P	HPEDIT	Revise Existing Hatches
'	ID	FC_ID. BMP	ID	Identify point coordinate
#_	ins	fc_s_ins. bmp	ins	Snap to base point of part
^^	INSERT	FC_INS ER.BMP	INSERT	Part insertion dialog
#_	int	fc_s_int. bmp	int	Snap intersection
^^	INTERSECT	fc_inter. bmp	INTERSECT	Trim/extend two entities to intersection point
^^	LENGTHEN	fc_leng. bmp	LENGTHEN	Lengthen/Shorten line or arc
^^	LIBRARY	FC_PLI B.BMP	LIBRARY	Insert part from library
^^	LIGHT	FC_LIG HT.BM P	LIGHT	Set light for rendering
^^	LINE	fc_line.b mp	LINE	Draw line(s)
^^	MENU	fc_f_mn u.bmp	MENU	Load another menu



_	mid	fc_s_mid.bmp	mid	Snap mid point
^^	MIRROR	fc_mirr.bmp	MIRROR	Mirror objects (original remains)
^^	MOVE	fc_move.bmp	MOVE	Move selected objects
^^	MTEXT	PC_MTEXT.BMP	MTEXT	Multi-Line Text Entry
^^	MTEXTEDIT	PC_MTEXTEDIT.BMP	MTEXTEDIT	Edit Multi-Line Text
^^	NEW	fc_f_new.bmp	NEW	Create a new drawing
^^	NGON	fc_ngon.bmp	NGON	Draw a regular polygon
#_	nxt	fc_s_nxt.bmp	nxt	Snap closest point
^^	OFFSET	fc_offset.bmp	OFFSET	Offset-Copy: Parallel or Complement
^^	OPEN	fc_f_open.bmp	OPEN	Open an existing drawing
^^	PALOPEN Parts Auto fc_part_.mnp	FC_PARTS_WIT.BMP	PALOPEN Parts Auto fc_part_.mnp	Toggle to horizontal (or vertical) palette
	PALSWITCH pc_det.mnp	pc_detail_pal.bmp	PALSWITCH pc_det.mnp	Open Detail Palette
	PALSWITCH pc_dim.mnp	pc_dimension_pal.bmp	PALSWITCH pc_dim.mnp	Open Dimension Palette
	PALSWITCH pc_draw.mnp	pc_draw_pal.bmp	PALSWITCH pc_draw.mnp	Open Draw Palette
	PALSWITCH pc_edit.mnp	pc_edit_pal.bmp	PALSWITCH pc_edit.mnp	Open Edit Palette
	PALSWITCH pc_modi.mnp	pc_modify_pal.bmp	PALSWITCH pc_modi.mnp	Open Modify Palette
	PALSWITCH pc_part.mnp	pc_parts_pal.bmp	PALSWITCH pc_part.mnp	Open Parts Palette
	PALSWITCH pc_rast.mnp	pc_raster_pal.bmp	PALSWITCH pc_rast.mnp	Open Raster Palette
	PALSWITCH pc_snap.mnp	pc_snap_pal.bmp	PALSWITCH pc_snap.mnp	Open Snap Palette
	PALSWITCH pc_view.mnp	pc_view_pal.bmp	PALSWITCH pc_view.mnp	Open View Palette
	PALSWITCH pc_zoom.mnp	pc_zoom_pal.bmp	PALSWITCH pc_zoom.mnp	Open Zoom Palette
#'	PAN	fc_pan.bmp	PAN	Repositioning of drawing window
#'	PANDOWN	fc_down	PANDOWN	Pan Down

#'	PANLEFT	.bmp fc_right. bmp	PANLEFT	Pan Left
#'	PANRIGHT	fc_left.b mp	PANRIGHT	Pan Right
#'	PANUP	fc_up.b mp	PANUP	Pan Up
^^	PARTDEF	FC_PAR TD.BM P	PARTDEF	Define Part by selection
^^	PARTEXP	FC_PAR TX.BM P	PARTEXP	Write objects to external part file
^^	PASTE	FC_PAS TE.BMP	PASTE	Paste selection set from clipboard
#_	per	fc_s_per. bmp	per	Snap perpendicular to ...
^^	PLANVIEW	fc_planv .bmp	PLANVIEW	Plan view on current UCS
^^	POINT	fc_point. bmp	POINT	Set construction points
^^	POINT _M	FC_PM EAS.B MP	POINT _M	Measures entity with points
^^	POINT _S	FC_PSE GS.BMP	POINT _S	Segments entity with points
^^	POLYEDIT	FC_PED IT.BMP	POLYEDIT	Edit Polylines
^^	POLYLINE	fc_pline. bmp	POLYLINE	Draw a polyline
^^	PRINT	fc_f_prt. bmp	PRINT	Print the current drawing
^^	PROPEDIT	FC_CHP RO.BM P	PROPEDIT	Change Layer / Linetype
^^	PROPLAYER	FC_CH LAY.B MP	PROPLAYER	Modify the layer attribute of entities
#_	qua	fc_s_qua .bmp	qua	Snap quadrant of circle or arc
^^	RASTER BMP2RLC	fr_b2r.b mp	RASTER BMP2RLC	Convert monochrome Bitmap file to RLC file
^^	RASTER RBORDER	fr_bord. bmp	RASTER RBORDER	Toggle Raster Frame On/Off
^^	RASTER RCUT	fr_cut.b mp	RASTER RCUT	Cut Region from Raster
^^	RASTER RLC2BMP	fr_r2b.b mp	RASTER RLC2BMP	Convert RLC file to monochrome Bitmap
^^	RASTER RLOAD	fr_open. bmp	RASTER RLOAD	Load Raster File
^^	RASTER RMIRROR	fr_mirr.b mp	RASTER RMIRROR	Mirror Raster
^^	RASTER RMONT	fr_mont. bmp	RASTER RMONT	Assemble / Fit Raster Files
^^	RASTER RMOVE	fr_move. bmp	RASTER RMOVE	Move Raster

^^	RASTER RPROP	fr_prop.bmp	RASTER RPROP	Property Settings of Raster File
^^	RASTER RROTATE	fr_rot.bmp	RASTER RROTATE	Rotate Raster
^^	RASTER RSAVE	fr_save.bmp	RASTER RSAVE	Save Raster
^^	RASTER RUNLOAD	fr_clos.bmp	RASTER RUNLOAD	Remove Raster
^^	RCONFIG	FC_RE_NDC.BMP	RCONFIG	Preferences for Fast Rendering
^^	RECTANGLE	fc_rect.bmp	RECTANGLE	Draw a rectangle
^^	REDO	fc_redo1.bmp	REDO	Redo recent Undo
#'	REDRAW	fc_redr2.bmp	REDRAW	Refresh current drawing window
^^	REJOIN	fc_join.bmp	REJOIN	Rejoin broken lines or arcs
^^	RENDER	FC_RE_ND.BMP	RENDER	Create rendered bitmap
^^	RING	fc_ring.bmp	RING	Draw filled rings / circles
^^	ROTATE	fc_rotat.bmp	ROTATE	Rotate selected objects
^^	RTEXT	PC_RTE_XT.BMP	RTEXT	Text Entry By Reference
^^	RTOP	FR_TOP.BMP	RTOP	Change display order
^^	SAVE	fc_f_sav.bmp	SAVE	Save the current drawing
^^	SCALE	fc_scale.bmp	SCALE	Resize selected objects
^^	SELECT	FC_SEL_EC.BMP	SELECT	Select Objects
#'	SETTINGS	fc_pref.bmp	SETTINGS	Preferences and Modes for Drawing and Editing
^^	STRETCH	fc_strec.bmp	STRETCH	Stretch-Moving of Objects
^^	TABLES	FC_TA_BLE.BMP	TABLES	Info on drawing database tables
^^	TABSERVICE	fc_tserv.bmp	TABSERVICE	Drawing Tables Maintenance
#_	tan	fc_s_tan.bmp	tan	Snap tangent to ...
^^	TCORRECT	PC_TC_ORREC.T.BMP	TCORRECT	Multiple Text Revisions By String
^^	TEXT	fc_text.bmp	TEXT	Generate text in drawing
^^	TEXTEDIT	FC_TXT_ED.BMP	TEXTEDIT	Edit Single-Line Text

^^	TLMODE	fc_layou .bmp	TLMODE	Toggle between Layout Mode and Model Mode
^^	TRAPEZOID	fc_2cone .bmp	TRAPEZOID	Draw a cone (2D)
^^	TRIM	fc_trim.b mp	TRIM	Trim entities to cutting edge
#'	TSCREEN	FC_F_C MD.BM P	TSCREEN	Toggle to/from Text Screen
#	U	fc_undo 1.bmp	U	Undo previous command
^^	UCS	fc_ucs.b mp	UCS	User Coordinate System Functions
^^	UCS C	fc_ucsm. bmp	UCS C	UCS Dialog
^^	UNDELETE	FC_UN DEL.B MP	UNDELETE	Undo last delete operation
^^	VIEW	fc_view. bmp	VIEW	Save or retrieve named view
#'	VIEWPAR	FC_PRE F.BMP	VIEWPAR	Parameters for view commands
^^	VNOTE	Fc_comt .bmp	VNOTE	Insert audio or text comment
^^	VTXEDIT	FC_VT XED.B MP	VTXEDIT	Edit Polyline Vertices
#'	WCASCADE	FC_WC ASC.B MP	WCASCADE	Cascade drawing windows
^^	WMFOUT	FC_WM FOU.B MP	WMFOUT	Create Windows Metafile from drawing
^^	WTILEHOR	FC_WTI LH.BM P	WTILEHOR	Tile Windows Horizontally
#'	WTILEVER	FC_WTI LV.BM P	WTILEVER	Tile Windows Vertically
^^	XPLODE	FC_EXP LO.BM P	XPLODE	Break complex objects down to single entities
^^	XPLORER	fc_xplor. bmp	XPLORER	Xplorer: Drawing Tables Navigator
#	X,Y,Z	fc_xy.b mp	X,Y,Z	Coordinate Filter X,Y,Z
#'	ZOOMBACK	FC_ZB ACK.B MP	ZOOMBACK	Zoom to previous view
#'	ZOOMFIT	FC_ZFI T.BMP	ZOOMFIT	Zoom to drawing extents
#'	ZOOMIN	FC_IN. BMP	ZOOMIN	Zoom In: Nearer to objects
#'	ZOOMOUT	FC_OU T.BMP	ZOOMOUT	Zoom Out: Away from objects
#'	ZOOMWIN	FC_WI	ZOOMWIN	Zoom Window: Specified by box

## **Adding Additional Fonts**

To add additional fonts, run the CONFIG command and note the default font file location specified on the Directories tab. Copy additional .SHX or .FSH font files to this directory, as desired.

Note: For True-Type fonts, consult your Windows documentation for installing instructions

## **Drawing Setup**

### **Drawing Setup**

Assuming that you have properly setup your hardware and configured AllenCad, as outlined in previous sections, you will need to ensure that certain additional configuration settings are currently set (either by default or through use of a Template drawing), that are appropriate to the new drawing. A number of commands present dialogs that allow you to change these parameters and they include:

1. **SETUP** – To specify feet and inches or decimal drawing units, angular measurement type (degrees/minutes/seconds, grads, radians), as well as angle zero orientation.
2. **LIMITS** – To set drawing boundaries.
3. **SETTINGS** – Setting drawing parameters and drawing mode configuration.
4. **LOADLTYPE** – Loading linetypes that you will be needing in the drawing
5. **LAYER** – This command calls the Layer Manager, which allows you to create new layers and setup or change layer parameters (default color, linetype, etc.).
6. **FONT** – This command helps you set up text styles (fonts and their characteristics) for use in the drawing.
7. **DIMTYPE** – This command helps you to set the various dimension parameters that you will want, when you dimension your drawing.

### **See Also:**

#### **Hardware Configuration:**

1. Video Resolution Settings
2. Printer Setup
3. Tablet Configuration

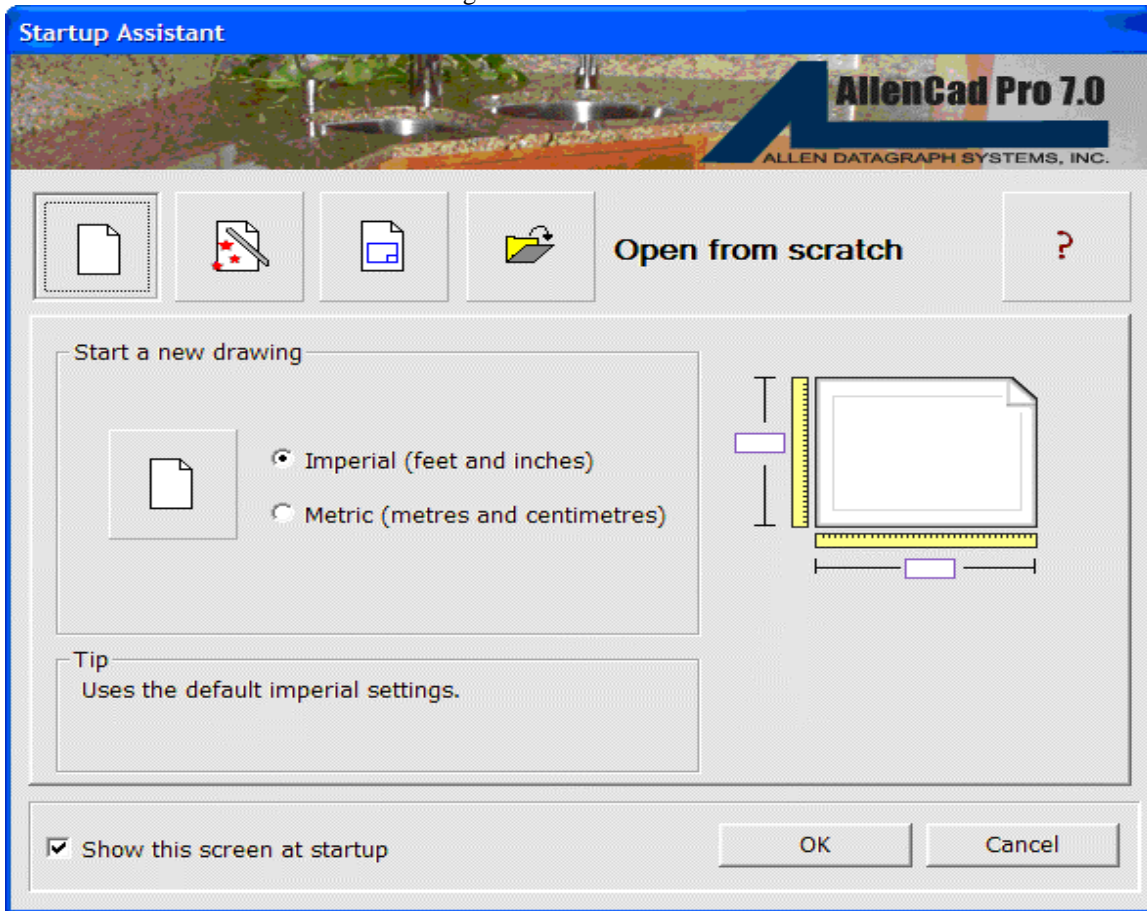
#### **Program Configuration:**

1. Program Configuration
  - Default Open and Save File Formats / AutoSave
  - Default Program Support Directories
  - Alias Command Setup
  - Hotkey Command Setup
  - Program Color Setup - Drawing Background, Cursors, etc
2. Desktop Layout Configuration
3. Redefining Menus, Dialogs and Palettes
4. Redefining Toolbars (including Left Sidebar)
5. Changing Menus

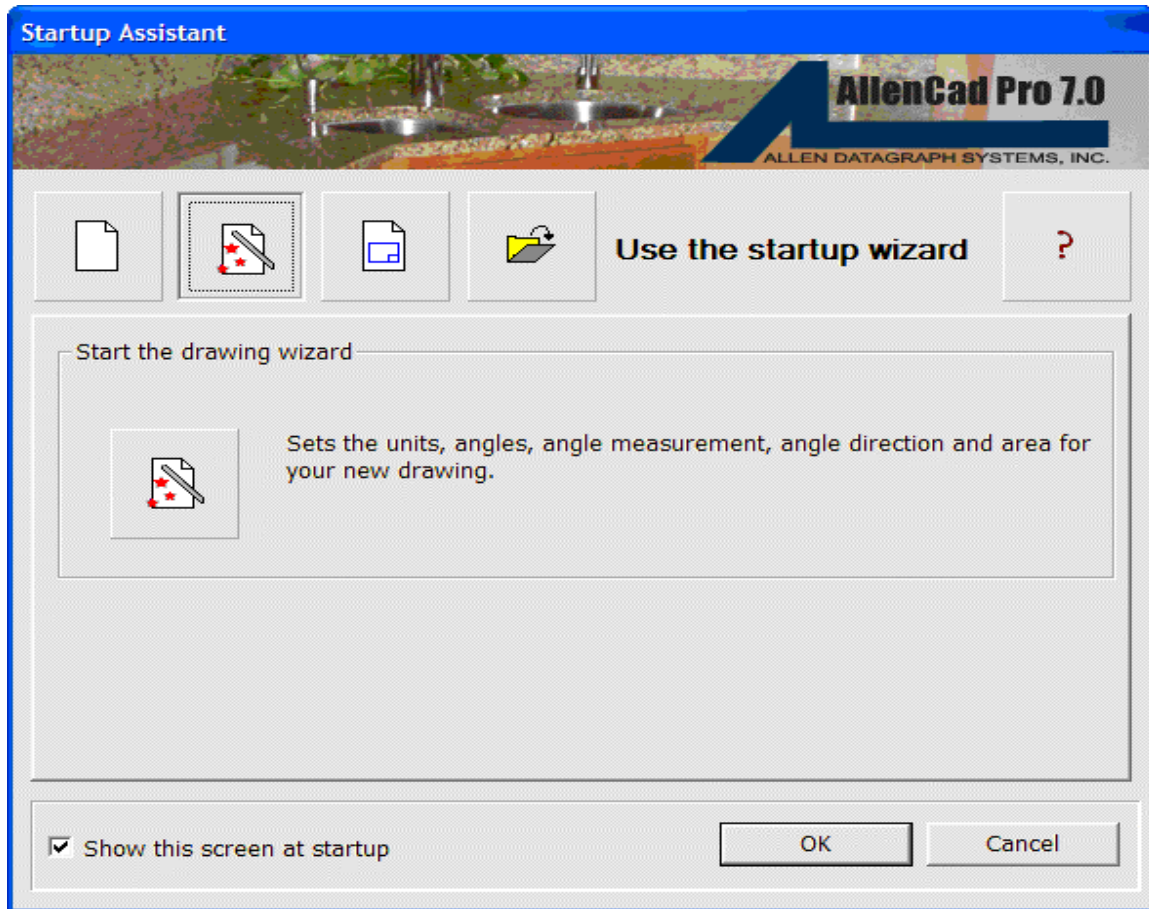
## Command ASSISTANT

The ASSISTANT command is AllenCad's one-stop drawing setup wizard. It incorporates the functions of several other configuration and program commands, including SETUP, LIMITS, TEMPLATE and OPEN. This enables you to make all of your drawing setup definitions in one location. The Assistant opens automatically during program installation, ensuring that you get up and running right away, with the settings that you need. Each time you start the program afterward, AllenCad either opens the Assistant or reminds you that the Assistant is there to help you, depending upon your preference. The command is issued by typing the command ASSISTANT (or the command alias AST) at the command prompt or selecting the desktop icon.

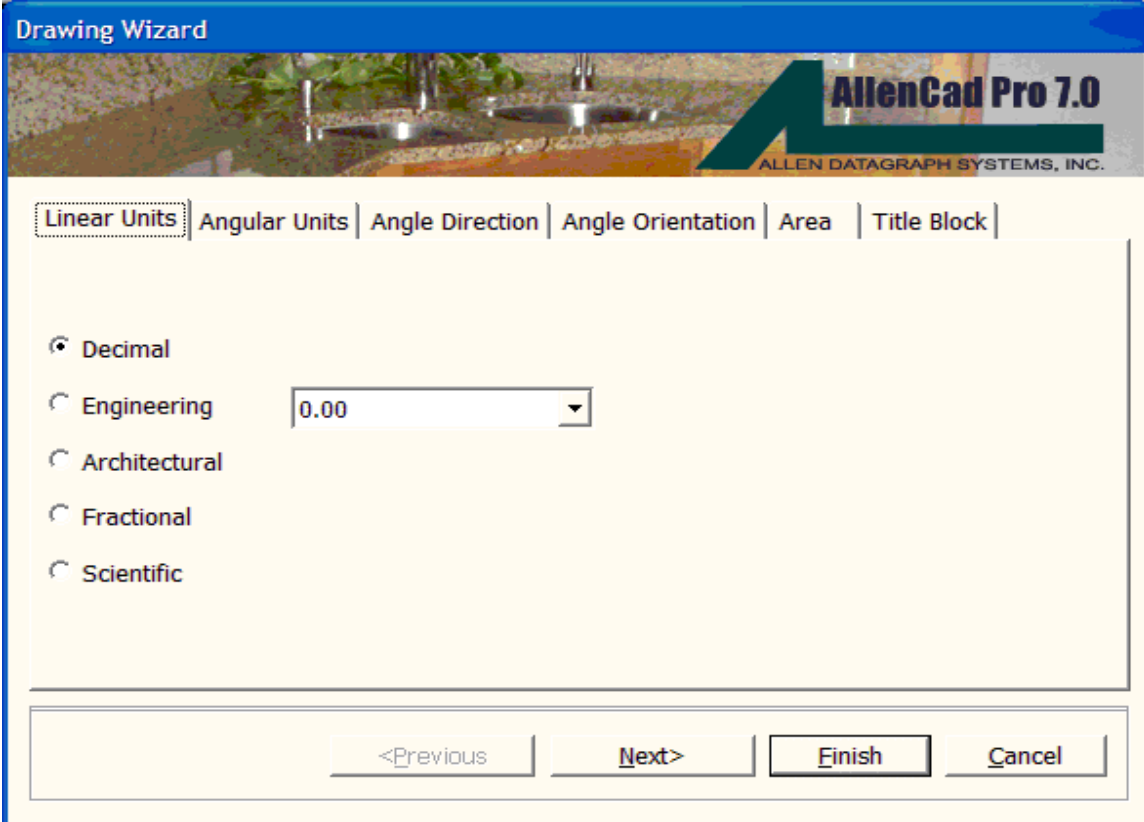
The startup Assistant screen is pictured below. On it, you will see four buttons, each described in the window below in the dialog, as you select each button. The first button enables you to select between the default Metric and Architectural unit settings.



The second button is your Drawing Setup Wizard. It leads you to dialogs in which you set such things as the unit formats and accuracies that you will require, the coordinate boundaries of the drawing that you will draw, as well as the insertion of a pre-built drawing surround and title block (if you wish).



When you select the Drawing Wizard, you are presented with a dialog containing six labeled tabs. The first of them, Linear Units, is pictured below. If you will be using feet and tenths (of feet) or metric, you would select the decimal option. If your units are feet and inches, you will select Architectural or Engineering units and if you wish to measure drawing objects strictly in inches and fractions, you would select Fractional units. Scientific units are also available, if you should need to use them. Before you start though, you should ensure that you will be using only one measurement system as you draw, or else you need to have a conversion approach in mind that will let you merge data of different formats. Once you have selected the units, you must specify the precision that the program will report dimensions in. Note that this is only the reporting or labeling accuracy and that the internal, calculations are based upon a much higher level of accuracy, altogether. The illustration to the right on the dialog, shows an example of the currently selected accuracy as it will be reported.



The image shows a 'Drawing Wizard' dialog box from AllenCad Pro 7.0. The 'Linear Units' tab is selected, showing radio buttons for Decimal, Engineering, Architectural, Fractional, and Scientific. The 'Engineering' option is selected, and a dropdown menu next to it shows '0.00'. The dialog has a blue title bar and a header area with a landscape image and the software logo. Navigation buttons at the bottom include '<Previous', 'Next>', 'Finish', and 'Cancel'.

**Drawing Wizard**

**AllenCad Pro 7.0**  
ALLEN DATAGRAPH SYSTEMS, INC.

**Linear Units** | Angular Units | Angle Direction | Angle Orientation | Area | Title Block

☒ Decimal  
☐ Engineering 0.00  
☐ Architectural  
☐ Fractional  
☐ Scientific

<Previous   Next>   Finish   Cancel

The Angular Units tab lets you select between the different angular unit formats, in much the same manner as with the linear measurement units. As with linear measurement settings, you can also specify reporting precision and see an example of the currently selected accuracy.



**Drawing Wizard**

**AllenCad Pro 7.0**  
ALLEN DATAGRAPH SYSTEMS, INC.

Linear Units | **Angular Units** | Angle Direction | Angle Orientation | Area | Title Block

Specify the angle of measurement and the precision for angles:

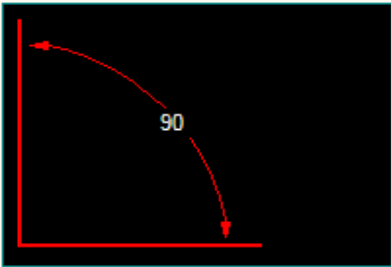
☒ Decimal      Precision:

☐ Deg/Min/Sec

☐ Grads

☐ Radians

☐ Surveyor



<Previous      Next>      Finish      Cancel

Angle direction simply sets the angle (or bearing) of "zero degrees" in your drawing. All bearings or reported drawing angles are derived from this setting. East orientation is normally selected, however North may be preferred in instances where azimuth bearings are used.

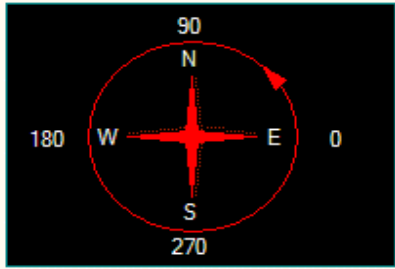
**Drawing Wizard**

AllenCad Pro 7.0  
ALLEN DATAGRAPH SYSTEMS, INC.

Linear Units | Angular Units | Angle Direction | **Angle Orientation** | Area | Title Block

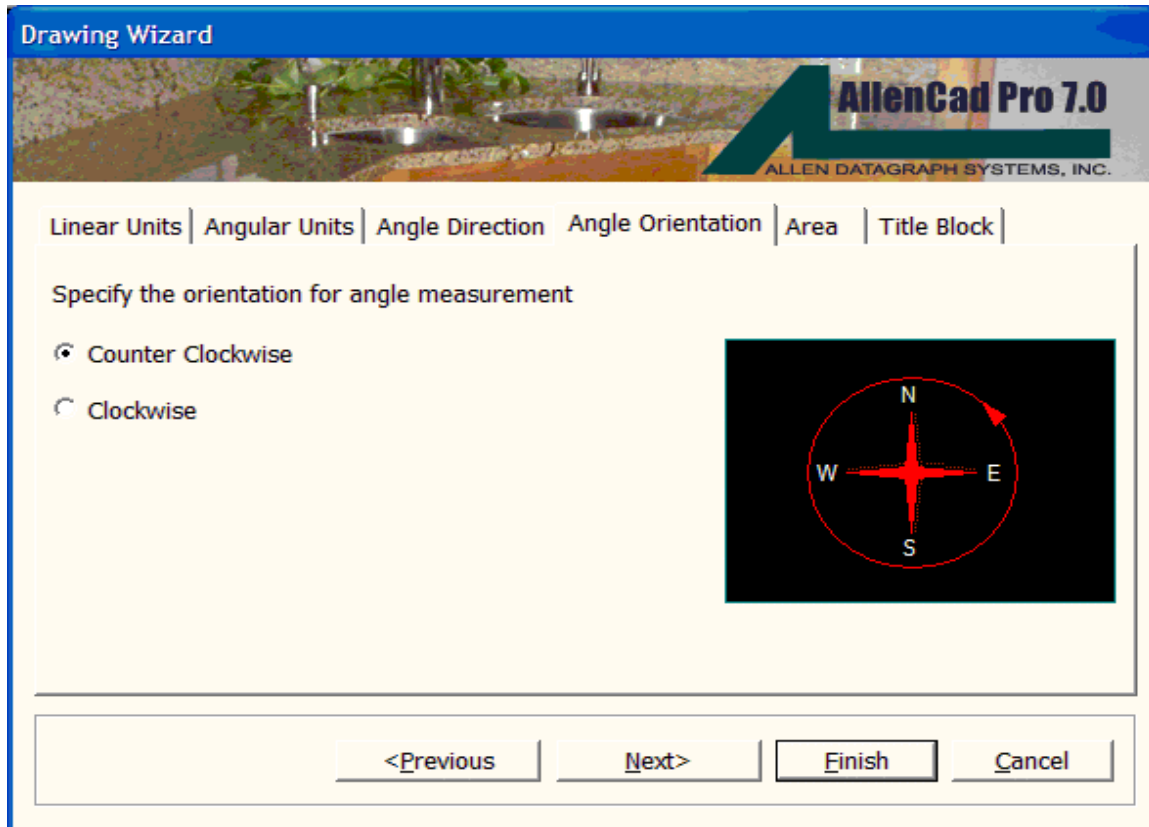
Specify the direction for angle measurement

☒ East  
☐ North  
☐ West  
☐ South  
☐ Other Angle:



<Previous   Next>   **Finish**   Cancel

Angle orientation simply specifies whether angles are measured in a clockwise or anti-clockwise direction from the specified zero alignment. By convention, anti-clockwise is most commonly used, however, clockwise measurement may be required in some instances, such as when azimuth bearings are used.



Area settings simply define the size of your drawing area. These values can be left blank if you are unsure, or if you do not wish to limit the drawing area. These values specify the "X and "Y dimensions of your drawing that correspond to the length and width of your drawing when you are looking at it in plan view (from above). There is no limit setting for elevation (the "Z coordinate values).

In the example below, these settings would be quite adequate, if the object (say a house floor plan) were to be slightly less than 35'-0x 24'-9in overall size.

Whenever you give the ZOOMPAGE command while drawing, your view of your drawing will zoom out (or in) to these limits.

**Drawing Wizard**

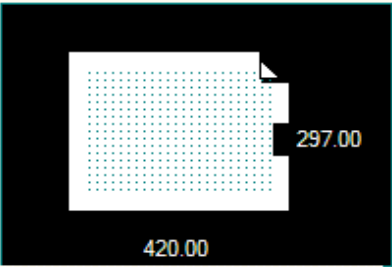
**AllenCad Pro 7.0**  
ALLEN DATAGRAPH SYSTEMS, INC.

Linear Units | Angular Units | Angle Direction | Angle Orientation | **Area** | Title Block

Specify the area you want to represent using full scale units:

Width:

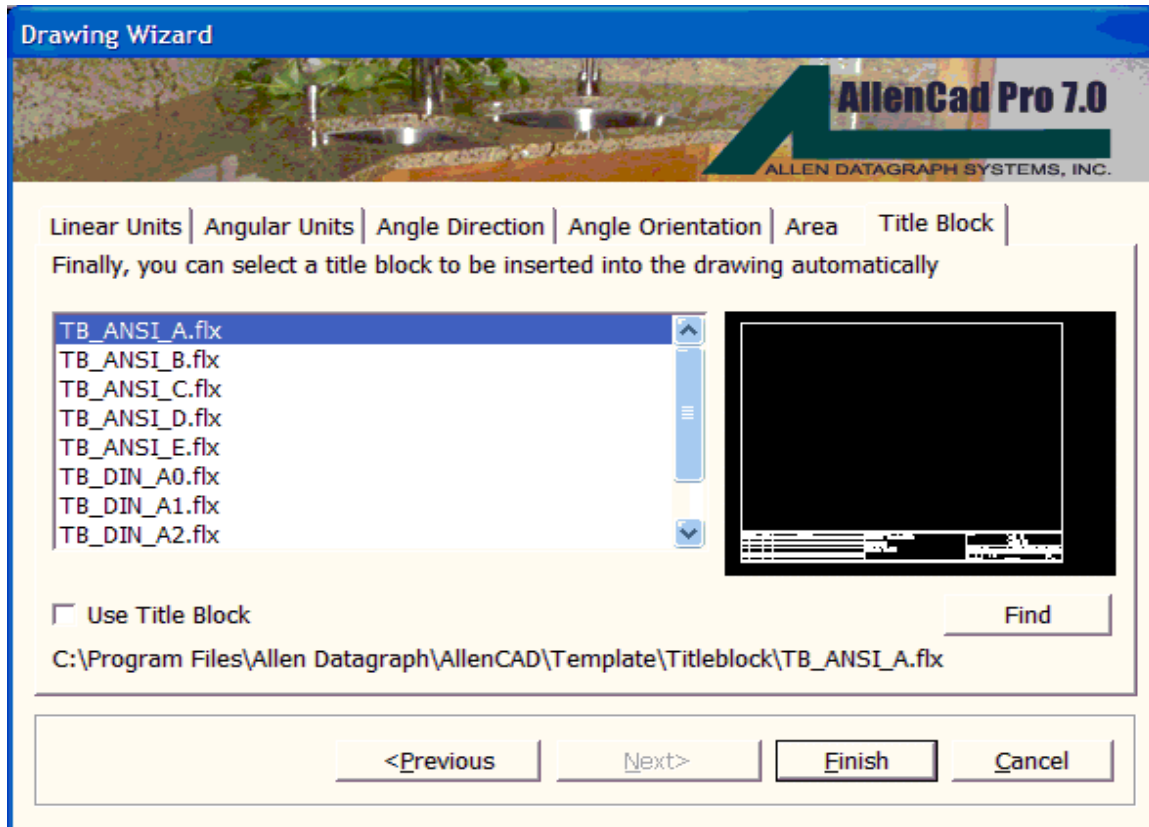
Height:



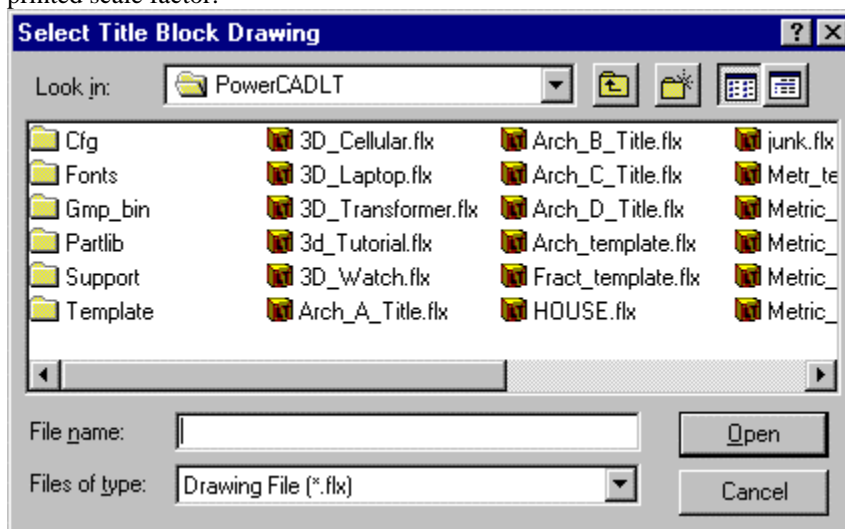
Example:  
To draw in area of 420x210 mm, enter 420 under width and 210 under height.

<Previous | Next> | Finish | Cancel

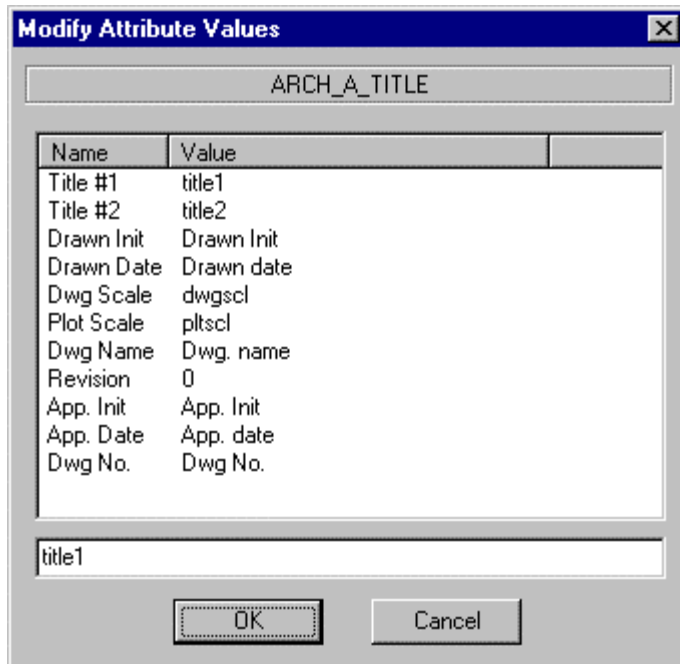
The Title Block tab helps you to insert a pre-configured title block into your drawing. This can be either one of the basic title block drawing files provided with AllenCad, or one of your own, that you have set up for this purpose.



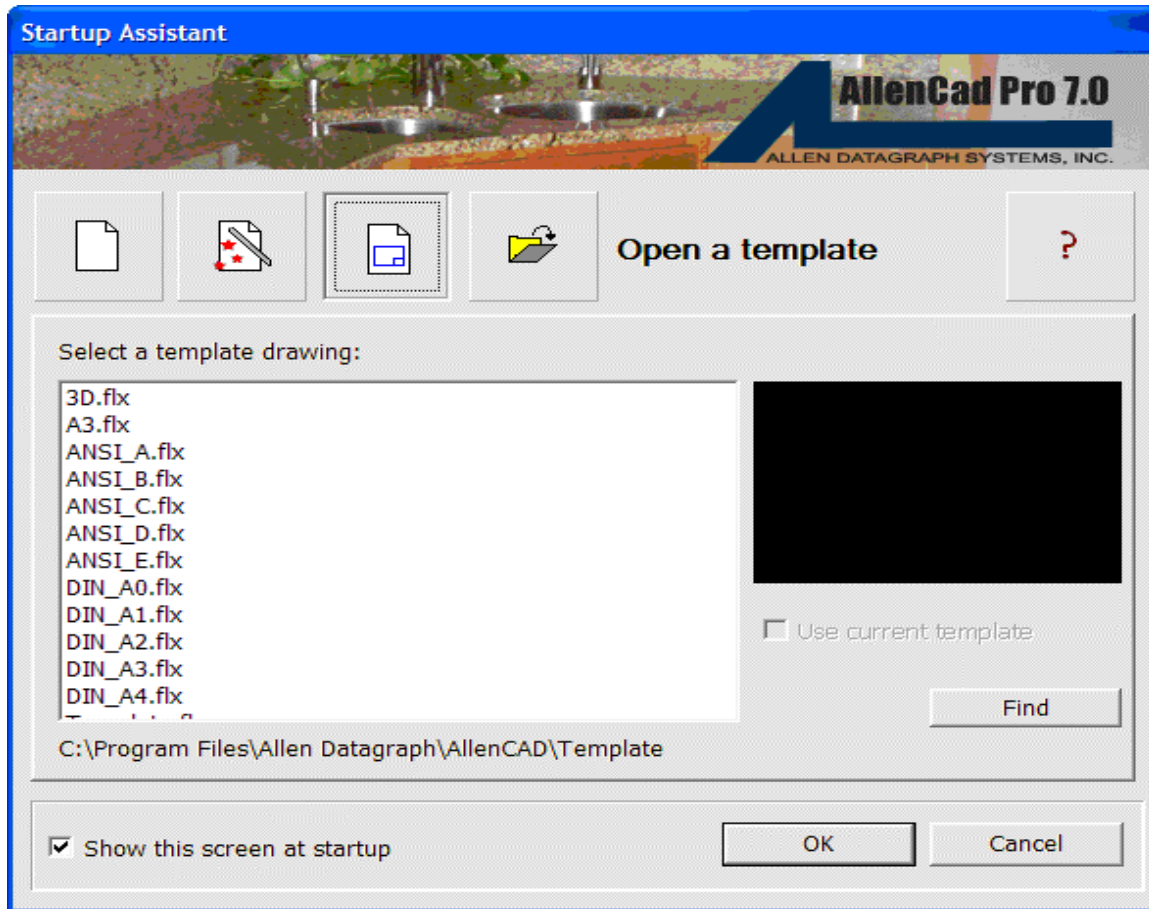
To insert one of the Architectural or Metric title blocks that are provided with your program, simply press the Find... button and navigate to the program directory. There, you will find title blocks for drawing sheet sizes A, B, C, and D as well as metric ISO size A1. Simply select the appropriate size for your drawing. This size will take into account, both the size of your drawing “model as well as the printed scale factor.



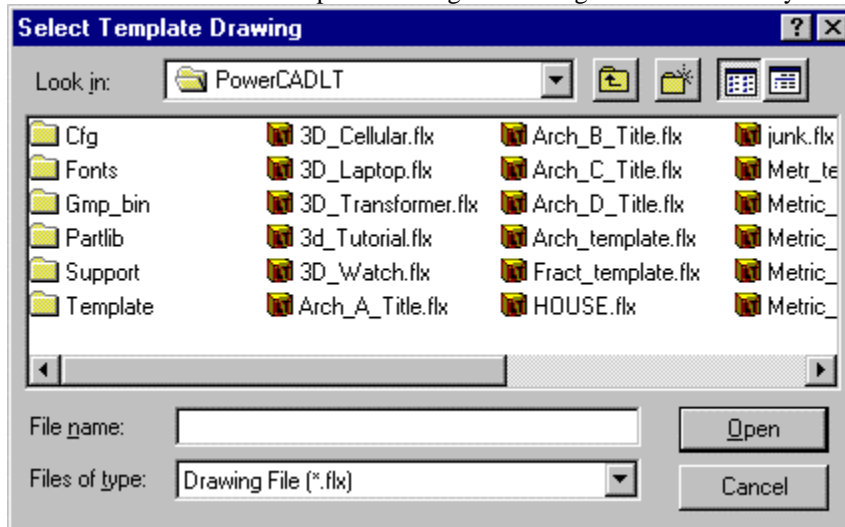
As the title block is inserted, you are presented a dialog such as pictured below that allows you to input text values that will be instead into your title block as part of the installation process. You can modify these entries afterwards, using the ATTDEDIT command, should this be necessary.



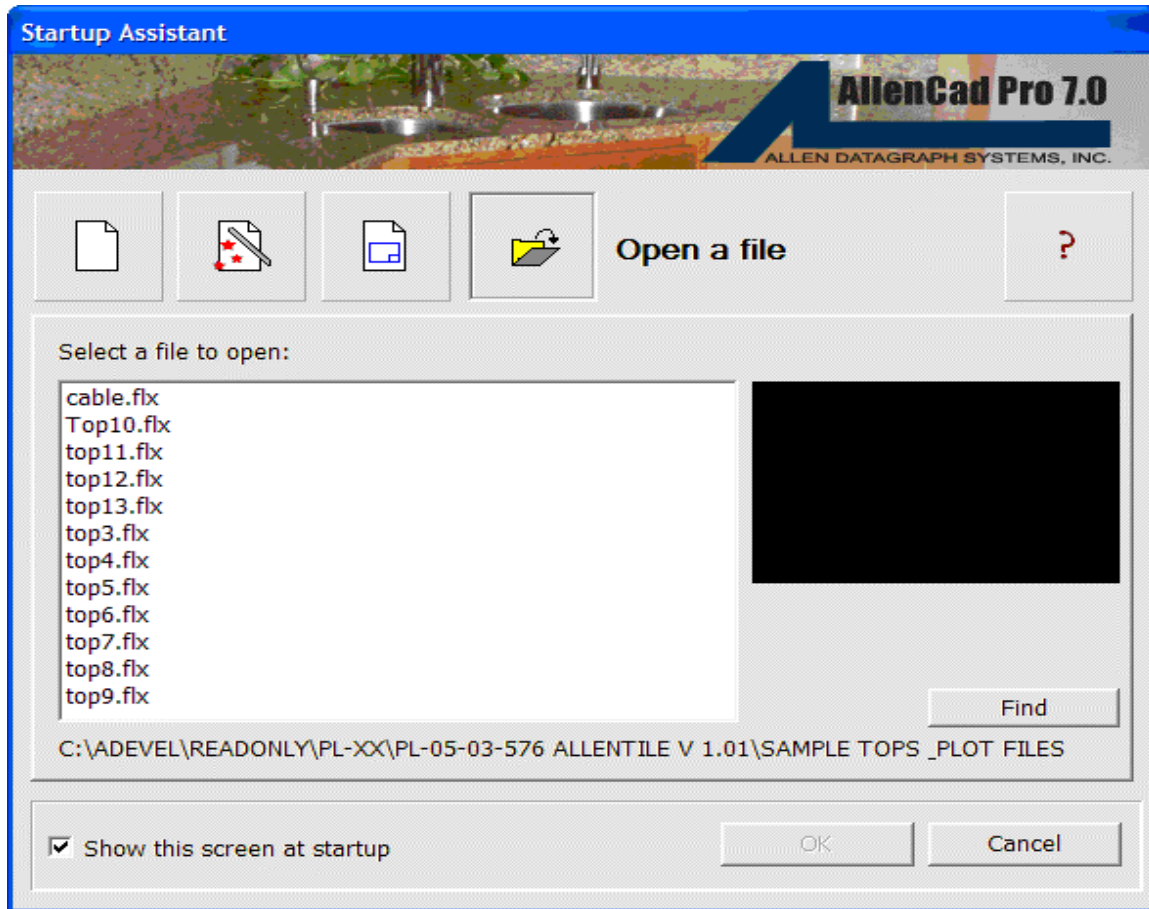
The third button on Assistant's startup dialog is the Template button. Although basic template drawings containing standard drawing setup units and parameters have been provided, you may prefer to use the Assistant to help you create new template drawing files, that contain all your own standard drawing setup information, layers, text styles, title blocks, etc. Thereafter, you can use either this Assistant utility, or set the template option to "ON" on the NEW command dialog, to automatically call your template drawing file settings into any new drawing that you create. Using drawing templates, can therefore allow you to bypass initial drawing setups when starting new drawings with standard drawing setup characteristics, making you more efficient in your drafting.



Select the Find... button to open the navigation dialog to the location of your template drawings.



The fourth and final selection button is the File Open selection button. This corresponds to the [OPEN](#) command, used to open an existing drawing. By default, AllenCad starts the ASSISTANT command by opening the dialog to this selection. If you prefer not to use the Drawing Assistant each time you start the program, simply uncheck the selection box at the bottom of this dialog.



## Command SETUP

**Command: SETUP**

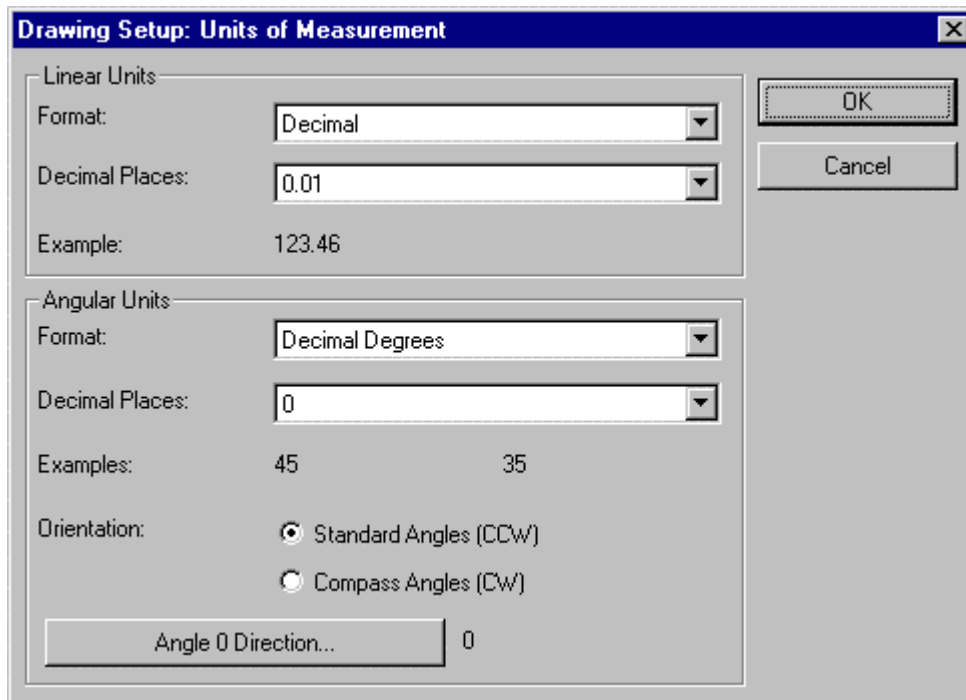
**Alias US**

**Located:**

- 1. Pull-down Menu: Tools – Drawing Setup Parameters**
- 2. Or Enter Command or Alias at the Command Line**

AllenCad allows you to set the format and display accuracy of numbers and angles. The settings are saved with each drawing, so they can be set differently in each drawing. The Setup command displays the Drawing Setup dialog box:





The command UNITSETUP is identical to this command and has been provided for the convenience of users familiar with AutoCAD commands.

### Linear Units

Linear units can be given a display accuracy ranging from zero to eight decimal places. In the case of fractional units, the display accuracy ranges from 0 (no fractions) to 1/256. You can select five kinds of units:

- Decimal units (metric) - 12.3456.
- Fractional units (unitless fractions) - 12 3/4
- Architectural Units (feet and fractional inches) - 12'-3/4" (See note below)
- Engineering units (feet and decimal inches) - 12-3.456'
- Scientific units (exponential notation) - 1.234E+02

### Angular Units

The Angular Units can be given a display accuracy ranging from zero to eight decimal places. AllenCad recognized these angular measurement systems:

- Decimal degrees - 12.3456
- Degrees - minutes - seconds - 12d34'56.
- Grads (400 grads in a circle) - 12.34g
- Radians - 12.34r
- Surveyors units (Quadrant Bearings)- N12d34'56 E

**Note:** Please refer to "Using Architectural Units" for details on the setup and use of architectural units.

### Orientation

In most drawings, angles are measured in the counterclockwise direction. This is the default in AllenCad and is called Standard Angles.

You can, however, set the orientation so that positive angles are measured in the clockwise direction, called Compass or Azimuth angles.

### Angle Zero Direction

In most forms of drafting, zero degrees is located at the positive X-axis (East, or 3 o'clock). AllenCad allows you to change the location of zero degrees. For example, surveyors normally set zero degrees as up on the page (North, or 12 o'clock).

The display of the status bar is updated depending upon the settings of the angle zero direction and the

orientation.



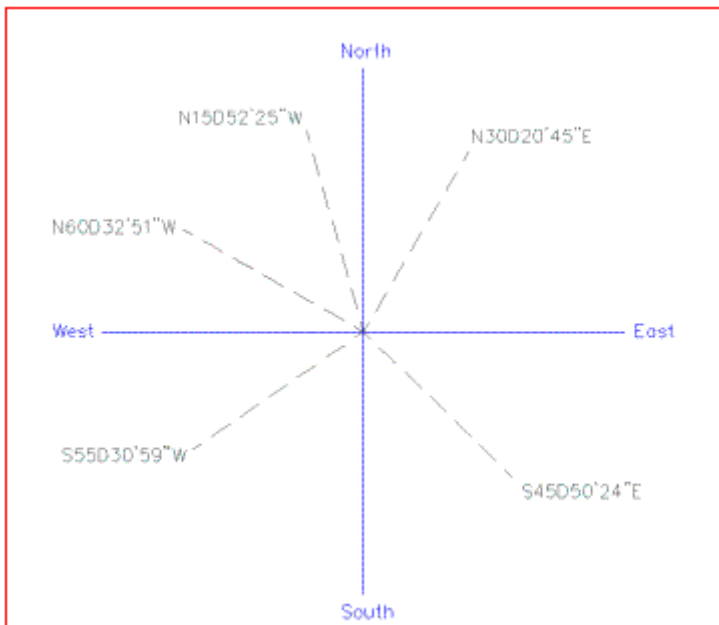
### Surveyor Units Entry

Because the surveyor units are so different from the other forms of angle entry, we have included a few examples:

Type of coordinate	Input	Example (Surveyor Units)
<b>Absolute Coordinates</b>		
Polar	distance<angle	7.5< N34D30'45"W
<b>Relative Coordinates</b>		
Polar	relative distance<angle	@7.5< N34D30'45"W

In surveyor's notation, the N stands for North. The 34D is 34 degrees from North. The 30' is an additional 30 minutes, and the 45" is an additional 45 seconds from North. The W indicates that the angle is towards the West.

There are 60 seconds in a minute, 60 minutes in a degree, and 360 degrees in a circle. Surveyors divide the circle into four parts, each with 90 degrees measured from the North or South poles in the direction of East or West.



In the figure above, we see the four points of a compass: North, South, East, and West. As examples of how the Surveyor's Units work, several lines have been drawn in various directions. We labeled these lines with the correct bearings. Note that all the letters used in the bearings are in capital letters, as required by AllenCad.

Note: When entering the N, S, E, W and D, be sure to enter them as capital letters -- lower-case letters

will not work.

## Command LIMITS

### Command LIMITS

Alias LM

#### Located:

**Entered Directly at the Command Line**

Although not strictly necessary, it can sometimes be helpful when plotting, to have the drawing area defined through the use of the LIMITS command. For greater convenience, it is wise to set your drawing area to a size that can be directly scaled to match the standard drawing sheet that you will be using. Afterwards, when plotting, you need only to specify plotting by limits to produce a plot of your full drawing, no matter what you presently are showing in your drawing edit window.

Command sequence is:

> LIMITS

Loading d:\AllenCad\SUPPORT\flx\_util.lsp... OK.

Reset the model space limits.

On/Off/<Lower-left corner><0.00,0.00>: [Accept default or enter new coordinates]

<Upper-right corner><12.00,9.00>: [Accept default or enter new coordinates]

## Settings



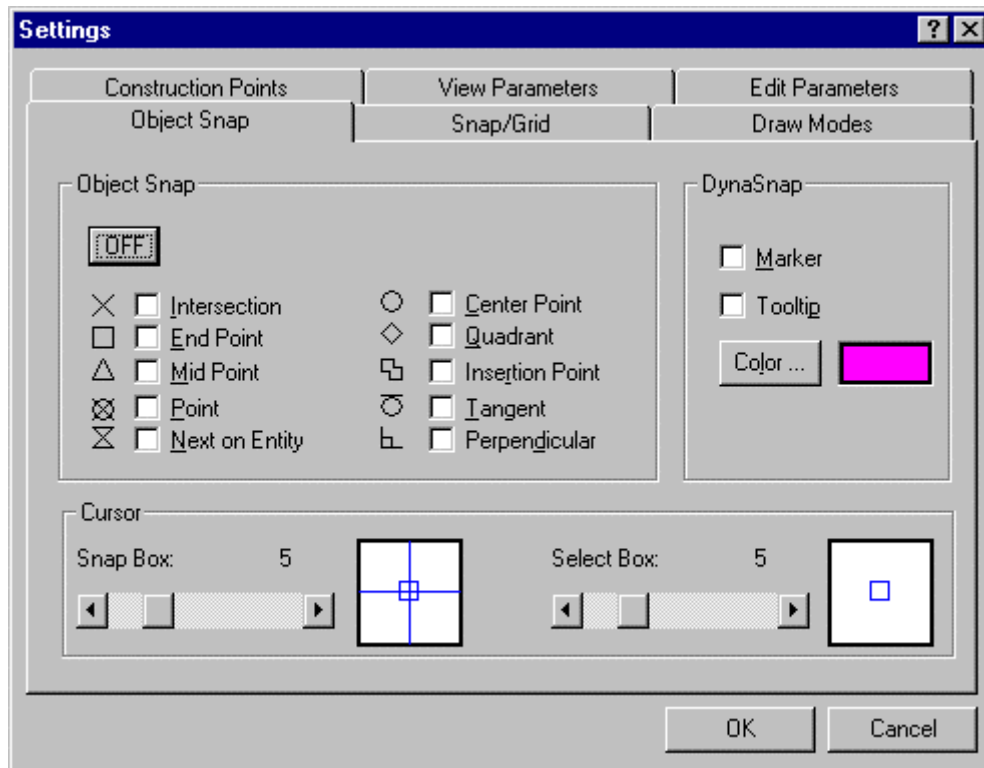
**Command: SETTINGS**

Alias ST

#### Located:

1. **Pull-down Menu: Tools – Settings**
2. **Palette: Snap**
3. **Or Enter Command or Alias Directly at the Command Line**

The command SETTINGS is the main command to set the parameters and modes of the drawing environment. Some of the tabs and settings shown below can be accessed directly, by other commands, in addition. These include: Object Snap – Command PRECPAR; Draw Modes – Command DRAWMODE; Edit Parameters – Command EDITPAR and View Parameters – Command VIEWPAR. The command calls a dialog containing a number of tabs that each have specific windows for entries that control the drawing environment.



For a more detailed explanation of each of these tabs and their entries, click on the links below:

[OBJECT SNAP](#)

[SNAP/GRID](#)

[CONSTRUCTION POINTS](#)

[DRAW MODES](#)

[EDIT PARAMETERS](#)

[VIEW PARAMETERS](#)

## **Command LOADLTYPE**

**Command:** LOADLTYPE

**Alias** LLT

**Located:**

### **1. Entered Directly at the Command Line**

The LOADLTYPE command is a command line version of the LINETYPE command, achieving the same result, but without the use of dialogs. When the command is issued, you will be prompted to supply the name of the .LIN file that is located in the program's Support subdirectory. Finally, you will be asked to name the linetypes you want loaded.

Layer Manager



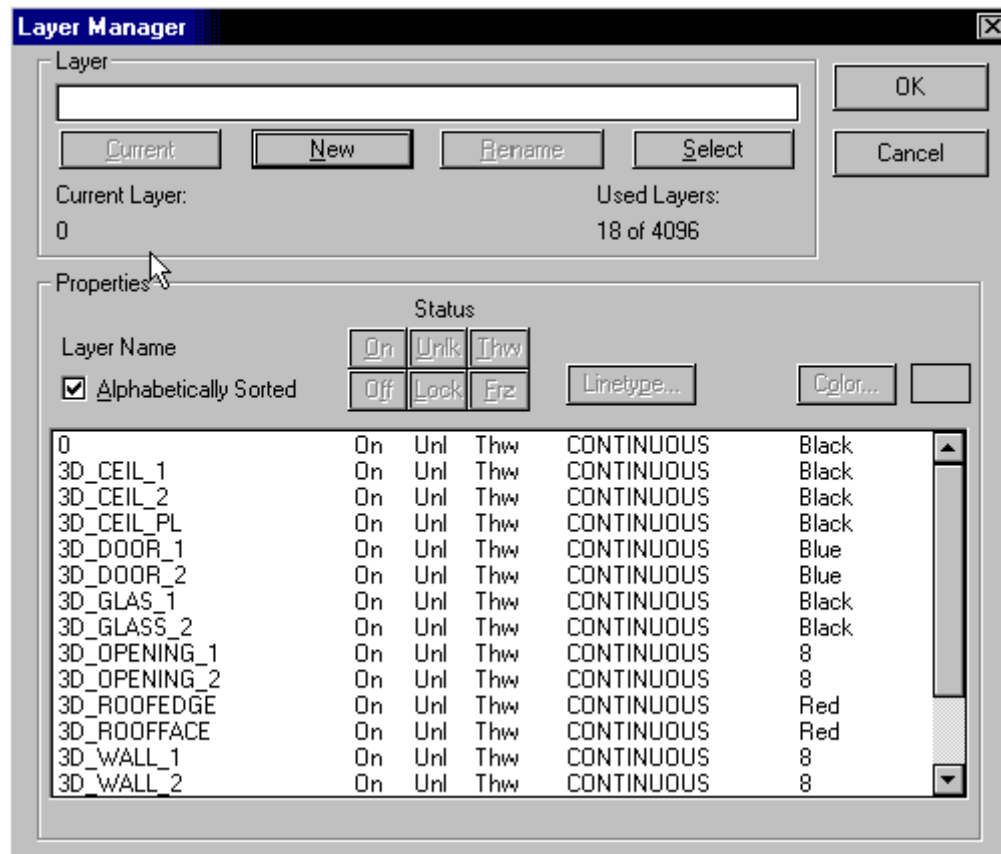
**Command:** LAYER

## Alias LA

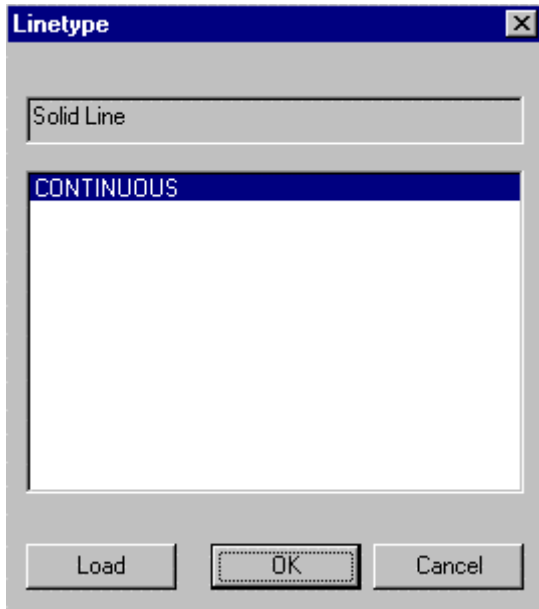
### Located:

1. Pull-down Menu: Tools – Layer
2. Properties Toolbar
3. Or Enter Command or Alias at the Command Line

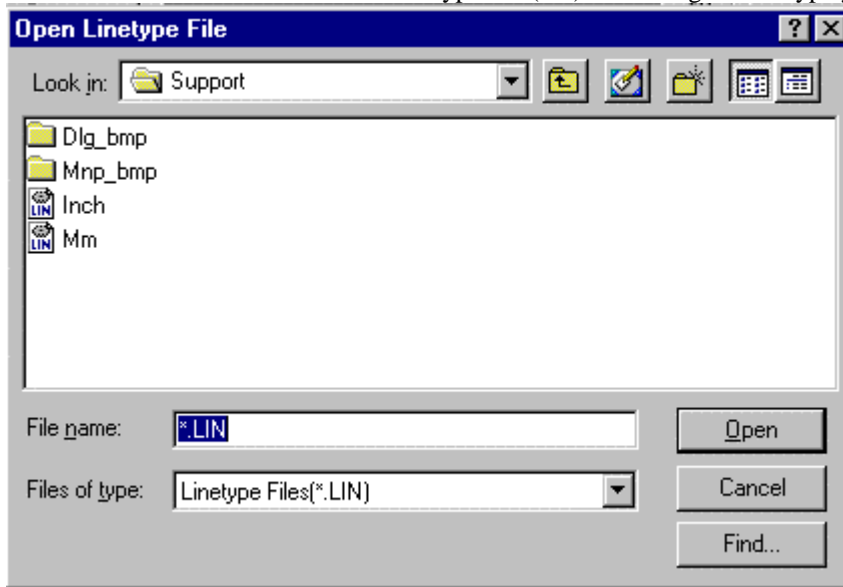
Creating new layers, choosing an active layer and controlling all previously described layer characteristics are accomplished through a dialog box. This dialog box is called *Layer Manager*.



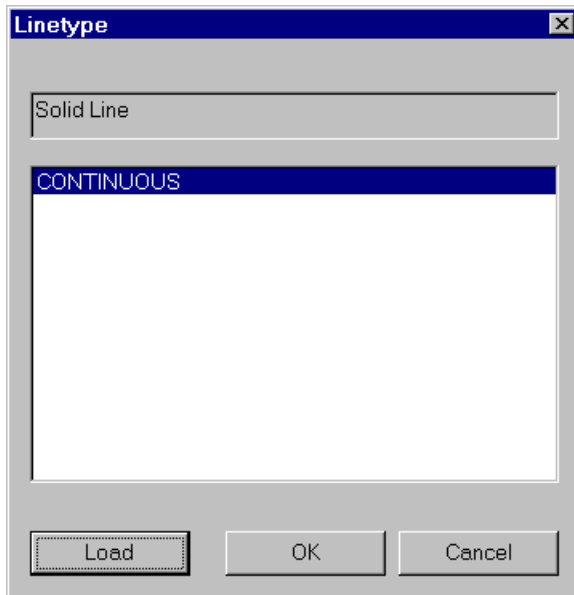
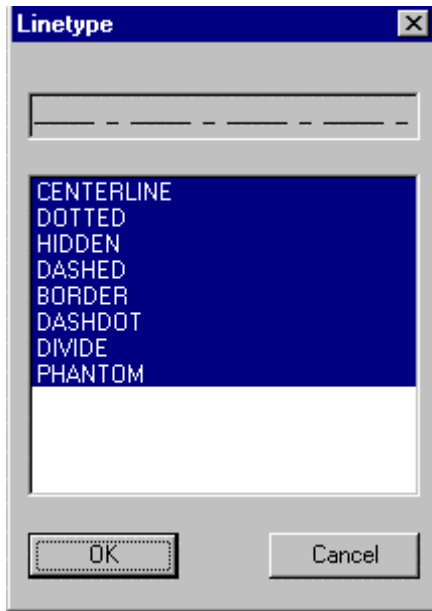
Line types may be loaded directly from the linetype dialog, in the Layer Manager dialog, by pressing the **Load** button



You will then be asked to select the linetype file (.lin) containing the linetype you want to open:



At this point, you are presented with the dialog below, showing all of the linetypes contained in the linetype file that you selected. As you select each linetype in turn, a representation of its appearance is presented in the window at the top of the dialog. Choose the linetype that you want loaded and press OK.



In the **Layer Manager** the default button is the **New** button. When you type in a layer name and press the return key, a new layer will be created and you remain in the layer manager.

### Current Layer and Used Layers

In the upper rows of the *Layer Manager* Dialog Box is a field for the name of the active (current) layer (left window) and a field for the number of defined layers in this drawing.

### Layer

Later rows contain combined view and entry fields for layers that still have to be worked on, and a selection box that allows the user to choose between creation of new layers, renaming layers and choosing an active layer.

### Properties

There are fields for the layer state and for choosing line types and layer color.

Directly beneath that, in the dialog box, all the existing layers of the drawing are shown. Any of these may be selected and activated with the cursor. The layer color and linetype are also shown there.

The meaning of the abbreviations used are shown [here](#).

Abbreviation	Meaning
On	Visibility on - The layer becomes visible
Off	Visibility off - The layer becomes invisible
Lock	Status free, drawing elements can be edited
Unlk	Status closed, drawing elements cannot be edited
Thw	Layer thawed, visible and can be edited
Frz	Layer frozen, invisible and cannot be edited, will not be regenerated

## Font... (Text Style Definition)



**Command: FONT**

**Alias FT**

### Located:

1. **Pull-down Menu: Detail – Fonts**
2. **Palette: Detail**
3. **Or Enter Command or Alias Directly at the Command Line**

The command FONT loads fonts from the font files and customizes them to the requirements of the drawing.

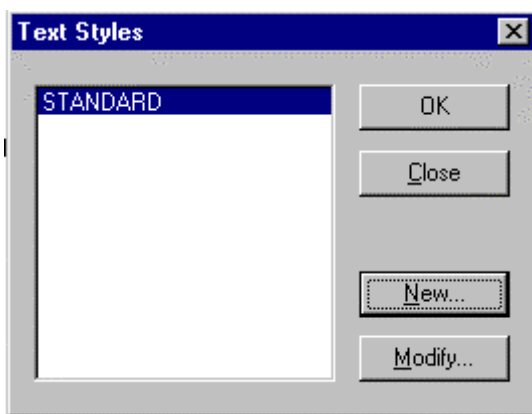
Fonts define the shapes of the text characters that make up each character set. In AllenCad, you can use:

1. AllenCad's own compiled shape (FSH) fonts,
2. AutoCAD's own compiled shape (SHX) fonts
3. True Type fonts.

All installed True -Type fonts, provided with your Windows system, are listed by font family name.

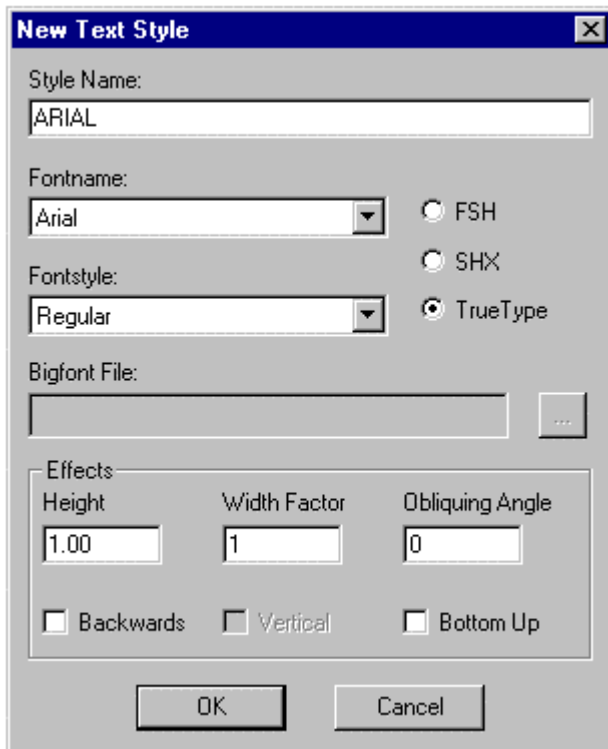
SHX fonts and FSH fonts are listed by the name of the file in which the fonts are stored.

All fonts to be used in the program must first be assigned to a text style. When you give the FONT command, you are therefore presented with the following dialog:



Here, you create a new text style or choose an existing one to modify, by assigning a different font. With the required style name highlighted, press OK and you will be presented with the font dialog:





In this case, we have chosen to create a new style called Arial and to apply the Arial true type font to this style. To do this, we click on the True Type radio button and select the down arrow on the Fontname box to display the available choices. We scroll to Arial and select it.

Next, we may need to select a fontstyle. Again we click on the down arrow in the fontstyle window and choose regular text, as we do now wish it to be bold, italic or otherwise.

If we wish to use a Bigfont file, we press the browse button to the right of the window and select our bigfont file from the available choices.

Other text effects are available at the bottom of the dialog. We can assign a text height, width factor or Obliquing (Tilting) Angle. If we want the text to run right to left in a mirror image, we can select the Backwards option. The Bottom Up option will give us text that is upside down. .

**Note:** Some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

Confirm the settings of the properties by selecting OK. Fonts added or changed in this manner can now be used for creating or editing text objects when the modified text style is current.

**Note:** Defined text styles are saved with the drawing file only with a reference to the corresponding font file. They are available any time that the drawing is opened, if the corresponding font file is found in the programs directories.

---

#### See also:

Adding Additional Fonts

Command QFONT

Command TEXT

## Dimensioning Styles



**Command: DIMTYPE**

**Alias DDR**

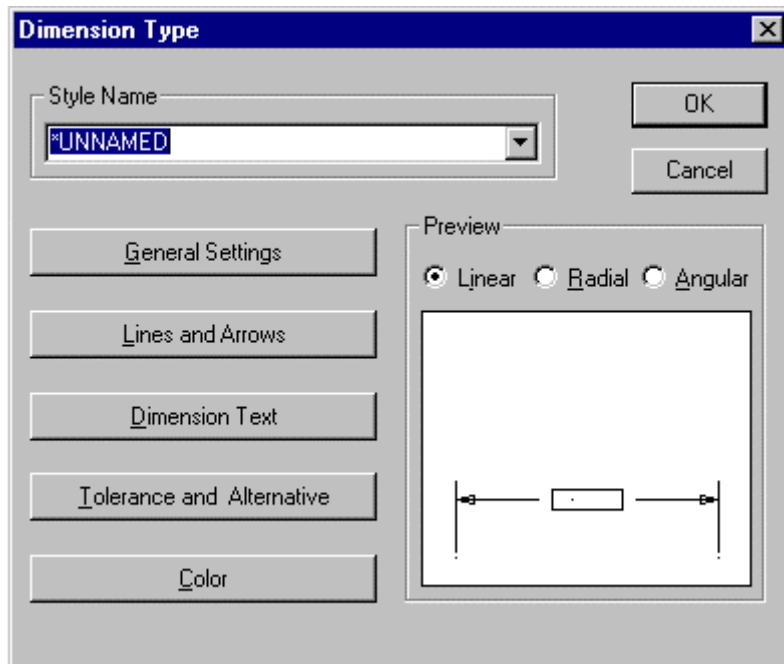
**Located:**

1. **Pull-down Menu: Dim – Dimension Type**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

Dimensioning format is controlled by the Dimensioning dialog box. Use the style name, input and selection dialog to either store the current settings as a dimensioning style, or to select a pre-defined style as the default.

The buttons described below each open additional dialog windows in which individual dimensioning parameters can be defined.

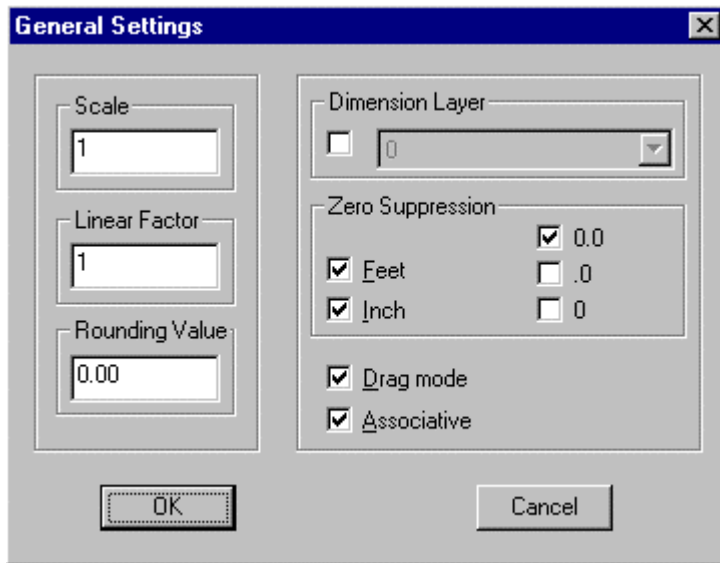
### **Dialog Box Options:**



When you issue the DIMTYPE command, you are presented with the opening dialog. Here, you see five bars on the left side that lead to specific configuration areas. On the right, you can click on the Linear, Radial and Angular radio buttons, to see a representation of current settings in the window below, for each dimension type.

Clicking on each bar will produce the dialogs detailed below:

### **General Settings**



### Scale

Scale value enlarges or reduces the size of all dimensioning elements (apart from the dimension value itself) and allows the dimensioning elements to be scaled to suit the aspect ratio of the drawing.

### Linear factor

Linear factor converts the measured drawing units into a dimension figure. The default value is 1.

### Precision

Precision defines the number of decimal places to which the measurement will be carried.

### Dim. Layer

The layer in which the dimensioning values are stored is identified by the name entered in the dimensioning layer selection field. Select the layer required, or enter a new layer name. It is prerequisite that the option has already been activated by clicking the appropriate control box. A check mark indicates the current status of the option.

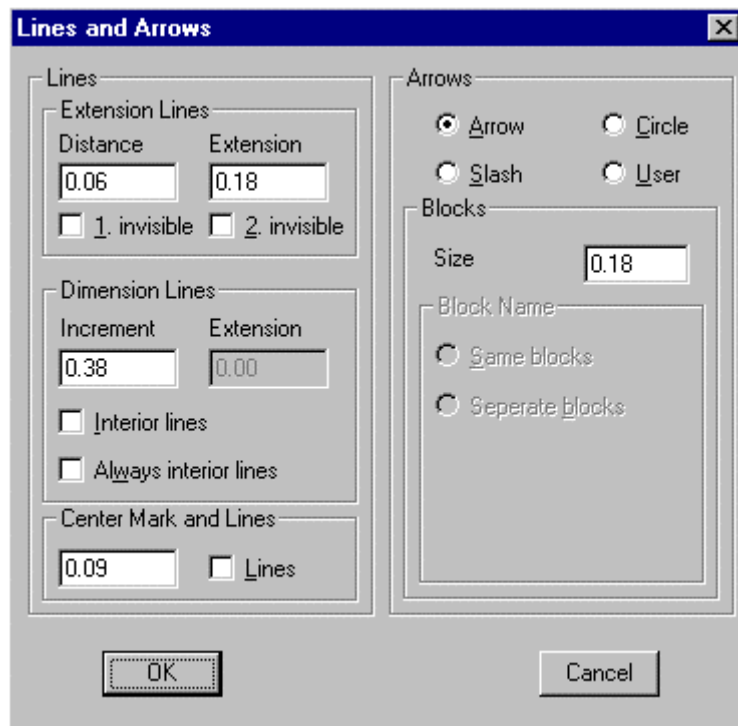
### Zero Suppression

The Zero suppression field decides the treatment of the display mode for zeroes.

### Drag mode/ Associative

The last two control boxes activate or deactivate either the "rubber band" mode and / or the associative character of the dimensioning.

### Lines and Arrows



Extension lines are the lines that extend the object dimensions outside the perimeter of the measured object, assuming that dimensioning is to take place outside of these perimeter constraints. These lines, and the nature of the arrowheads to be used, are defined in the Dialog Box *Lines and Arrows*.

## Lines

The fields in this area decide the characteristics of the extension and dimension lines.

### Extension Lines

This area defines the spacing required for the extension lines.

**Distance** allows the user to choose the space between the object and the point at that the extension lines begin. A value = 0 indicates that the lines will start directly on the object itself.

**Extension** dictates how far the extension lines should extend past the dimensioning line.

The control boxes marked **Invisible** can be used to suppress either one or both of the extension lines in the drawing.

### Dimension Lines

Dimension lines either connect the object edges, or the extension lines of the object that is to be measured. The dimensioning text is written along this line.

**Increment** sets the spacing between successive dimensioning lines that use the same extension lines.

**Extension** allows the user to enter the measurement by that the extension lines should run past the dimensioning lines. This value can only be entered when the **Slash** option has been selected as the dimension arrow type.

The field **Interior Lines** or **Always Interior Lines** decides whether dimensioning lines should be allowed to extend past the extension lines.

### Center Mark and Lines

This area defines the existence and size of a cross-mark indicating a circle center point. It also defines whether or not dashes or strokes should be added to the center point cross or not.

## Arrows

This field allows the user to select the type of arrowhead to be used.

## Blocks

This area describes the length of the arrowhead. Only the size selection of this area is normally active. Block Name is active only when the arrow style selected is User. The size of the Block to be used is defined here.

## Block Name

Active only when the arrow style selected is User, this field decides the Block Name. The name commonly in the field is ARROW.

## Separate Blocks

This field either allows or suppresses the use of divided or separate arrow blocks. When activated, it is also possible to define that elements should be used at either end of the dimensioning line.

## Dimension Text

The screenshot shows the 'Dimension Text' dialog box. It is divided into two main columns. The left column contains three groups of options: 'Text position' with 'Text always inside' and 'Text outside, lines inside' (both unchecked); 'Text always horizontal' with 'inside' and 'outside' (both checked); and 'Vertical text position' with 'above' (unchecked) and 'relative' (checked) with a value of '0.00'. The right column contains 'Dimension Text Style' set to 'STANDARD', 'Dimension Text' with 'Height' (0.18) and 'Factor' (1), 'Distance to dimension line' (0.09), an unchecked 'Frame' checkbox, and a 'Prefix <> Suffix' text field. At the bottom are 'OK' and 'Cancel' buttons.

These dialog window fields determine the position and look (size, style, spacing) of the dimensioning text.

## Text position, Text always horizontal, and Vertical text position

These fields serve to define the position of the text with respect to the dimension lines.

## Dimension text style, Dim text

These fields define the style and size of the text, and the distance separating text lines.

The prefix and suffix fields enable the user to type in a character string that will appear before or after the dimensioning text. For example, this field may be used to display the unit of measurement.

The <> character decides whether the character string will appear before or after the dimensioning text. For example, **approx. <> ft.** entered in this field would produce the following dimensioning : **approx. 12 ft.**

## Tolerance and Alternative Dimensioning

The tolerance settings control the tolerance, both the maximum and minimum values, allowed. The control fields activate or deactivate the display of these elements. A check mark in either of the control windows shows the current status of the individual options.

Values for the positive or negative tolerance are entered in the field marked Values. These fields also enable the user to type in a line of text (String).

The **Alternative mode** options control the use of a second, alternative dimensioning method. This mode, when activated, will allow dual dimensioning in metric and imperial values.

#### Factor

The conversion factor between the two units of measurement can be entered in the factor field.

#### Decimals

The number of decimal places selected will appear in the decimal display.

#### Alternative string

Alternative string should contain a description or title of the second unit of measurement.

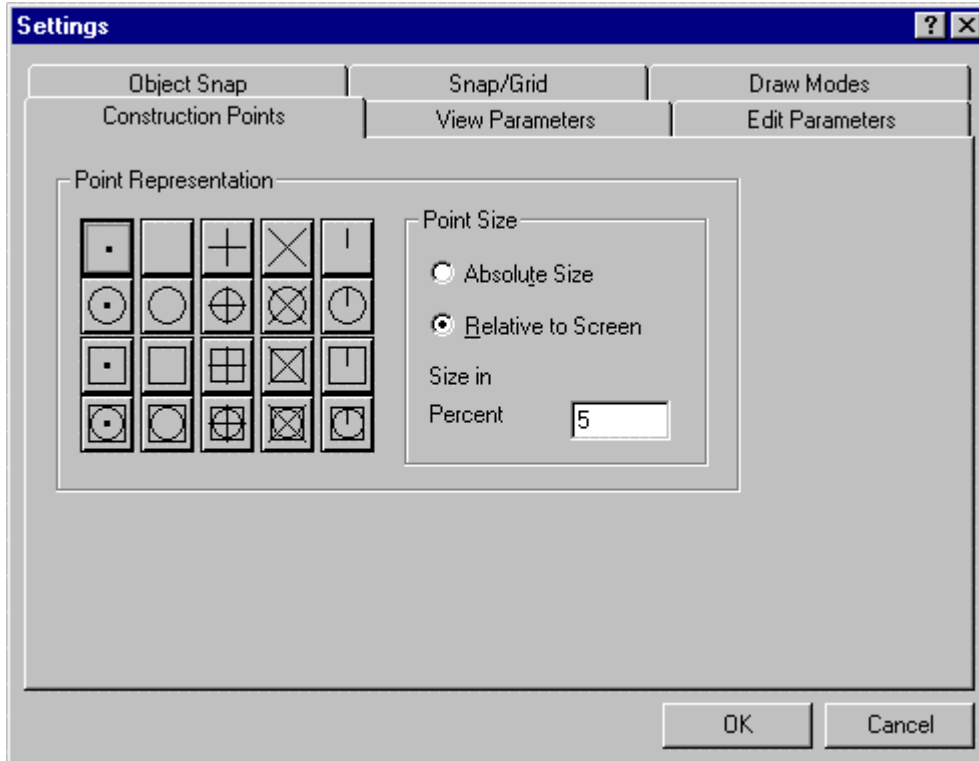
#### Colors:

The dialog window allows colors to be assigned to the dimensioning lines, extension lines and dimensioning text (dimension value, alternative dimensioning, tolerances, limit values, unit of measurement etc.)

Select the appropriate element and allocate a color from the displayed selection box. 256 standard colors are available in addition to the BYLAYER and BYBLOCK options.

### Construction Point Settings

The display of construction points can be controlled from the Construction Points tab on The SETTINGS command dialog.



As you can see from the illustration above, this tab contains a selection of different symbols that can be used to represent construction points. You simply select the symbol you want and regenerate the drawing to change all points to displaying that symbol. Note that only one symbol can be in effect at any one time and that all points will be shown with that symbol.

Apart from the symbols used, you will note that size of these symbols can be controlled, as well. While size can be specified exactly (in pixels), we recommend that you typically use the Relative to Screen option, as this will control the point display when you are zoomed in or out on your drawing.

## The Basics of Drawing

### Pointing or Identifying

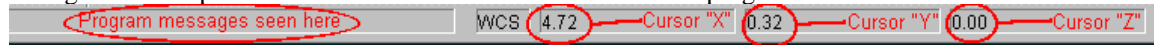
"Pointing", or "identifying", is taken to mean the input of points, straight lines, angles and other quantities by mouse or digitizer or other cursor control device.

This form of value input is recommended when the user must relate existing drawing entities. For instance, the center of a circle might be defined by the intersection of two lines. In this case, it is easier to

indicate a point resulting from the construction in order to define its coordinates.

Using the Object Snap functions assures that the accuracy of the coordinate input by pointing will be improved. The construction points like the starting or ending points, the intersection points, the center points, the perpendicular point etc. can be "pointed" with acceptable precision.

Note: Coordinate values can be previewed before point selection, by moving the cursor to the location and noting the values presented on the status bar at the bottom of the program window:

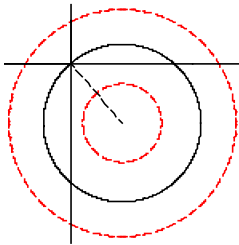


The coordinates of the current Cursor position will be interpreted directly as a pair of coordinates or, in some cases, as relative to the coordinates of a starting point. The clarity of this kind of value input is guaranteed by the dynamic preview. In doing this the user will get a kind of preview picture of the result of his or her action.

The program offers the possibility of using Coordinate Filters when point values are requested.

Coordinate Filters allow the user to relate the point value inputs to the X-, Y and Z-coordinates of existing entities.

## Dynamic Preview and Rubber banding



Dynamic Preview Mode and Rubber banding are visual construction aids that allow the user to watch the result of the pending operation before it is executed, and, if required, to make corrections. This feature also facilitates user understanding of commands and options and helps to learn the program.

### Rubberband: A Marking Line

In any command that requires selection of two or more points, a marking line (rubber band line) follows the cursor movement. This line connects the last selected point with the current cursor position.

This gives the user a visual impression of the operation and allows a better estimate of distances and angles between the last fixed point and the following point.

The rubber band line is useful both in drawing and in modifying objects.

Used during the drawing process it illustrates:

- ▶ the ending point of the next segment of a line, chain or polyline;
- ▶ the radius or diameter of a circle;
- ▶ the distance between starting- and ending point of an arc; or
- ▶ the next corner point of a 2D or 3D plane.

When used during modification of objects it helps show the distance and angle from one base point to a second point, as well as the target point for moving, copying, scaling or other modifying steps.

### Object Preview Picture

The Object Preview Picture shows in advance the contours of an object to be designed or modified. It is possible to recognize the consequences of changing of parameters like the radius of a circle or like removing or changing the angle of a copy of the original object. This option allows the user to interactively influence these changes.

## Deleting and Undeleting



Delete

Located:



1. **On the Horizontal Toolbar**
2. **Or Entered Directly at the Command Line**

During the creation of drawings, or, later while editing, changing or detailing of drawings, it may be necessary to delete certain parts of the drawing.

The program offers the command DELETE to accomplish this operation. Another command, ERASE can also be used for the same purpose. AllenCad provides both commands to assist users who may be more familiar with using one form or another of the command.

The command DELETE may be used for these drawing entities:

- ▶ Individual drawing entities, like lines, circles, rectangles, or for groups of these entities;
- ▶ Complex objects;

### **Complex Objects**

Complex objects are:

1. ▶ Parts and attributes
2. ▶ Polylines
3. ▶ Crosshatching
4. ▶ Dimensioning

To delete groups of entities, the respective group has to be converted to single entities again by the command EXPLODE Complex Objects.



### **Undelete: Withdraw Deletions**

Entities that have been deleted by mistake can be restored by the command UNDELETE. This will cause all entities previously deleted to be restored.

DELETE can be also canceled by the function Undo or the command U.

Call the commands:

1. ▶ by selecting the option in the pull-down menu *EDIT*
2. ▶ by keyboarding the command in the command line area
3. ▶ by clicking the symbol

## **Drawing Measurement**

The distance between two integer coordinates is called a drawing measurement or drawing unit. A line from point 0,1 to point 1,1 has a length of 1 drawing measurement or one unit. When drawing within a coordinate system you can create objects at a 1:1 ratio, avoiding calculation and other measurement errors. AllenCad identifies all points by coordinates (or in other words, within a coordinate system), so measurement accuracy is assured.

The drawing as a whole, or any excerpt from it, can be printed or plotted using any input scale factor. In this way, you save the trouble of creating complete drawings and detail drawings in different measurements, when you need to produce drawing prints at different scales.

### **See also:**

Coordinate Systems  
User Coordinate Systems  
UCS  
Distance

## **Choose the Drawing Scale When You Plot**

Unlike board drafting, AllenCad has no scale rules to work with. Object lines are drawn to full (1:1) scale. In other words, one drawing measurement or unit can be equal to one foot, meter, etc. When you measure a distance on the drawing, you obtain the correct numerical distance so you may need only to specify whether the units are feet, meters, centimeters, etc.

When you print or plot your drawing, you can specify any scale factor that you wish. Because the plotter utility does the scaling, everything on the drawing is scaled as it is being plotted, giving you the scale

factor on the print that you require.

This means that everything in the drawing (text, dimensions, border, and title block) is also scaled. Thus on print at a scale of 1=2 (half-scale), a 12" line will be plotted 6".

**See also:** Layout Mode

## Starting, Opening and Saving Drawings

### Starting, Opening and Saving Drawings

The program by default saves drawings as files with the file extension .flx. The default open and save format can be specified in Config – Common. Each file contains a drawing and, if necessary, up to four views of that drawing. Further on in this document, the term "drawing" may be used as a synonym with the term "file". When the document mentions "opening (or closing) a drawing", it refers to opening or closing of the corresponding file.

Apart from .flx drawings, AllenCad is also set up to open .dwg and dxf drawings and can SaveAS in these formats as well. For those who need to import/export in a raster graphic format, the commands RLOAD WMFOUT and BMPOUT are available, as is the ability to generate slides

Each drawing as well as each view of a drawing will be constructed in a separate window. All view windows of a drawing carry, in the Title Bar of the window, the same file name. They differ from each other in the continuous numbering (0 - 3), that is separated from the file name by a vertical line.



#### Starting a New Drawing

A new file has to be opened in order to create a new drawing. The command NEW in the menu *File* is used for this purpose. Alternatively, a new file can be created by clicking the corresponding system button on the Function Bar.



#### Opening an Existing Drawing

In order to continue working on an existing drawing in a later working session, the user must open the drawing. The term "Load" a drawing will also refer to "opening" a drawing. The command OPEN in the menu *File*, is used to do this. This command can also be activated by pressing the command OPEN or by clicking the standard symbol *Open* in the Function Bar



#### Closing the Current Drawing

Closing a drawing indicates that the work on this drawing is finished. The program remains open. If a drawing has been changed before closing there will be a prompt asking whether the file should be saved. This is to prevent unintended loss of work. Select the command CLOSE in the menu *File* or press the keys CTRL+F4. It is also possible to close the drawing window, as in all Windows programs, by double clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.



#### Saving Drawing Files

The command SAVE causes the drawings in the active window to be saved. At this time the file will be saved under its present name.

The command SAVEAS also saves the drawing in the current window, like the above-mentioned command SAVE. However, it allows the user to save the drawing under a different name, or in a different area.

The SAVEALL command will not only save the drawing in the current window but also all other open drawings.

#### Terminating the Program

It is very important to end every working session correctly. The only correct means of terminating the session is by executing the command EXIT in the menu *File*.

**See also:**

The Program's User Interface  
Configuration

### Setting up a drawing

To customize a drawing to your needs and requirements, you should be familiar with some basic terms and principals.

- Template drawing
- Drawing in real world units (Scale 1:1)
- Determine paper scale when printing or plotting
- Coordinate systems
- Units, Angles, Precision

When you start a new drawing, you should check the settings of the drawing environment with the SETUP command and/or the following commands:

1. Determine the linear unit system (Commands: LUNITS, LUPREC)
2. Determine the angular unit system(Commands: AUNITS, AUPREC, ANGBASE, ANGDIR)
3. Determine the sheet size (Commands: LIMMIN, LIMMAX)
4. Check precision aid parameters (Command: PRECPAR)
5. Check layer structure (Command: LAYER)
6. Check line types (Command: LINETYPE)
7. Check defined text styles(Command: FONT)

All described settings are stored with the drawing. When opening a new drawing, you can specify a Template Drawing. In such drawings, you can have already set up all parameters and modes for your specific works. New drawings based on template drawings will come up with the corresponding parameters. A template drawing is stored as an ordinary drawing file. So, it can also contain drawing entities (for example a frame or title block), a certain layer structure, pre-loaded line types, various text styles and even pre-defined views.

## Start a New Drawing



**Command: NEW**

**Alias N**

### Located:

1. **Pull-down Menu: File – New**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

A new file has to be opened in order to create a new drawing. The command NEW in the menu *File* is used for this purpose. Alternatively, a new file can be created by clicking the system button NEW on the Function Bar. Invoking the command will cause the file dialogue box to be displayed.

The program will name the drawing noname\_*n*.flx. The user may now name the drawing with the name of her or his choice.

To determine the storage location of the file select a drive, a directory, and, if required, a subdirectory in the middle part of the dialogue box. If the file name and the storage location are correct, confirm the settings. A new drawing (a new file) has now been created.

If a drawing file already exists with the same name the program will warn the user and require a confirmation so that the existing file can be replaced.

### Using Template Drawings

A template drawing may be called in order to start a new drawing. A template drawing is a model that can be used as basis for a final drawing or for the establishment of a drawing environment.

To start a new drawing using a template drawing, activate the check box Load Template in the dialog box *New file* and then double-click on the button Template Drawing.

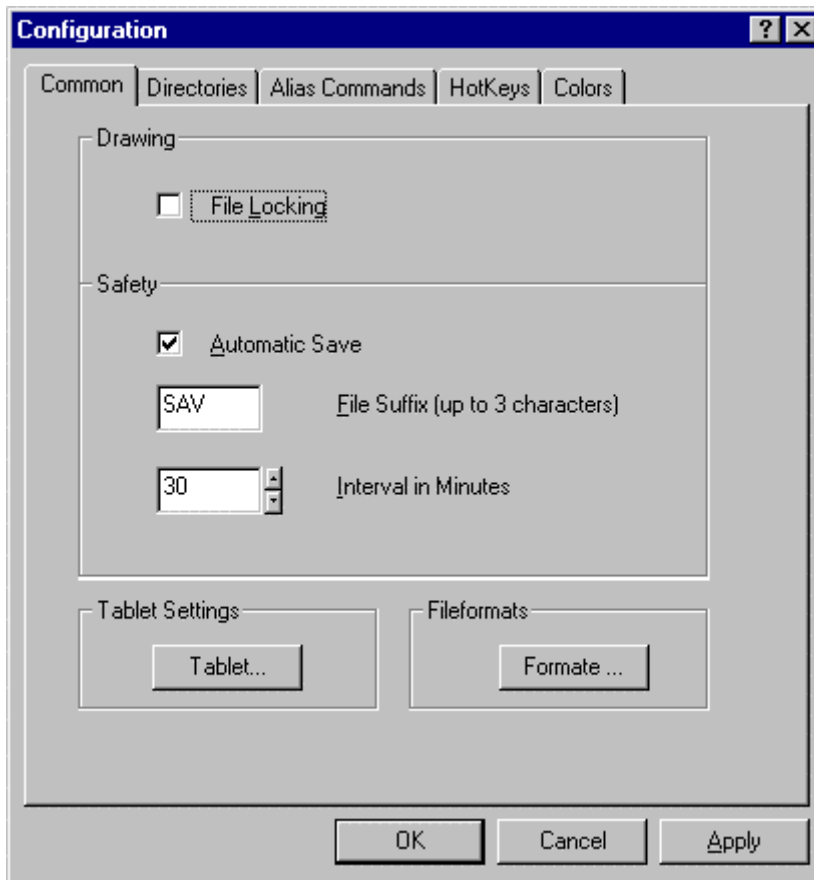
The dialog box *Template Drawing* will be displayed. Select the drawing to be to used as a template drawing and confirm the choice.

---

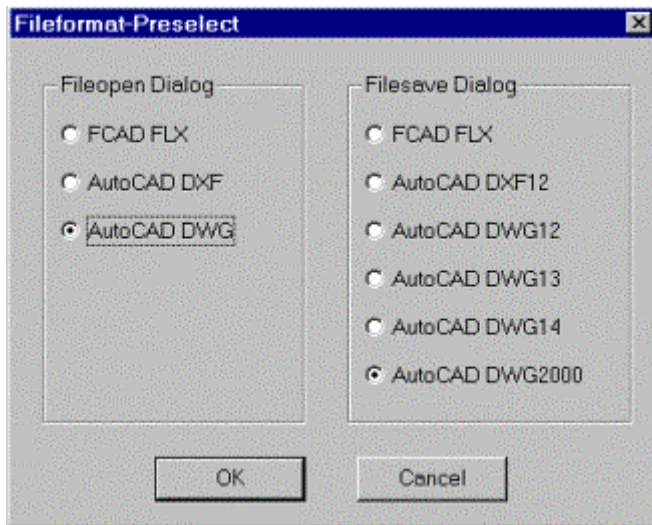
**See also:**

OPEN  
SAVE  
SAVE AS..  
SAVE ALL  
CLOSE

**Pre-defining Open and Save File Formats**



The default file format for the open and save dialog can be predefined in the config command register *Common*. If you select the button *Fileformat* , you will get the below displayed dialog. In this dialog, you can select your default file format to work within AllenCad.



## Template Drawings

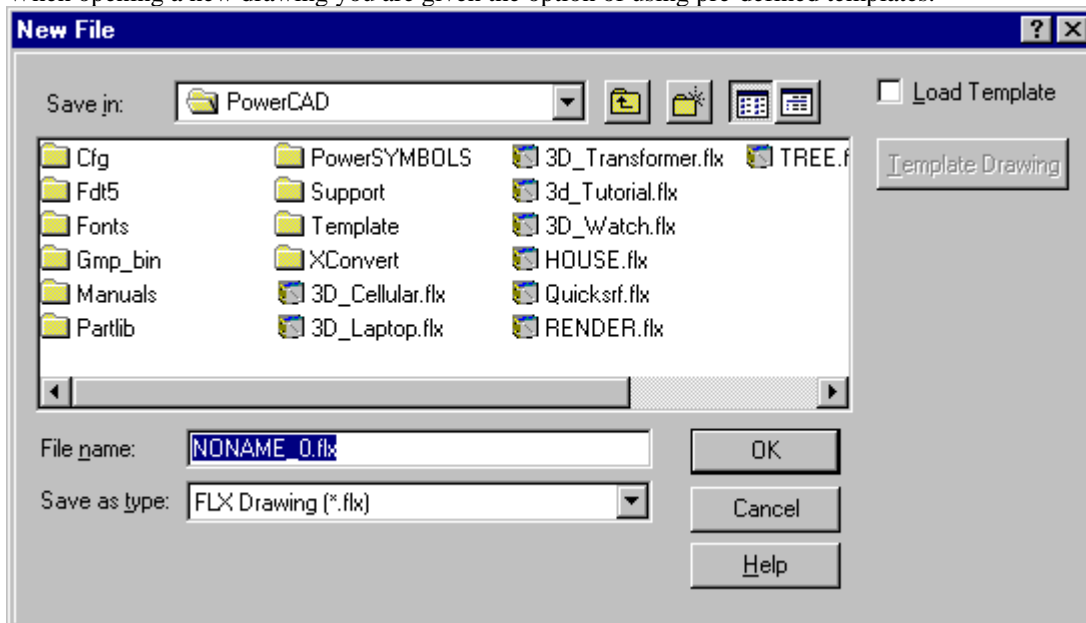
### Command TEMPLATE

#### Alias TP

#### Located:

1. Pull-down Menu: Tools – Resource Manager – Set Template Drawing
2. Or Entered Directly at the Command Line

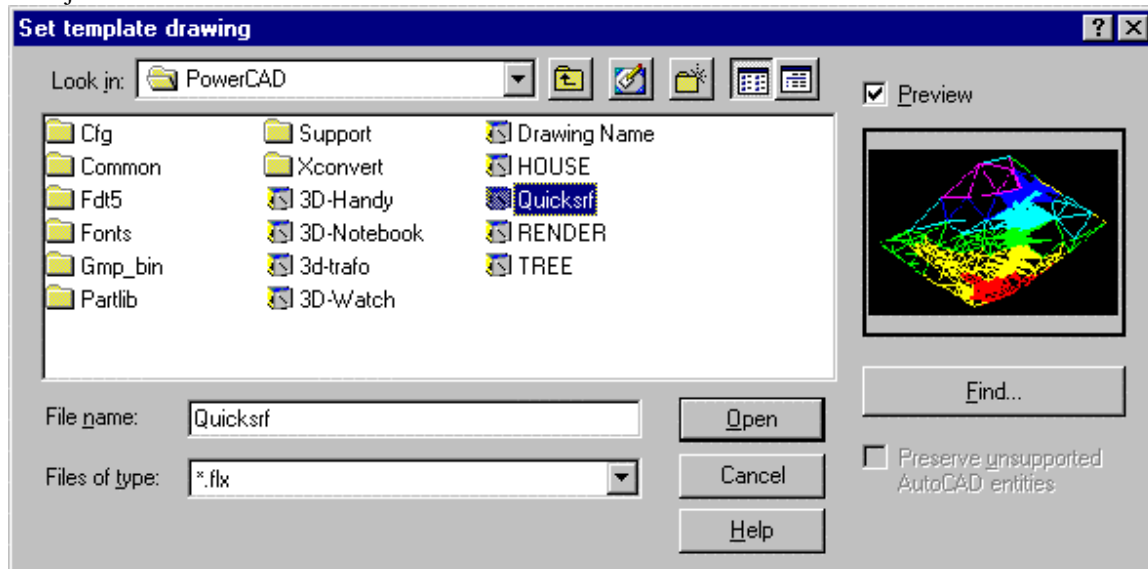
When opening a new drawing you are given the option of using pre-defined templates.



This is the standard file OPEN dialog, where you name your new drawing or choose to over-write an existing drawing with a new blank file. In the top right corner, you see the option to Load Template. If selected, this will load the default template drawing into your new file. If you wish to load another drawing as template, (with Load Template selected) select the Template Drawing button and browse to

select another drawing file as template.

To set a new default template drawing, however, you must use the TEMPLATE command from within the program. All subsequent new drawing files will have the option of using this new template that you have just made default.



When you issue the TEMPLATE command, you will be presented with the dialog shown above. You simply navigate to the drawing you want as template, select it and press the Open button. That drawing will then be the new default template drawing until changed, using this same procedure.

#### Notes On Templates:

1. Templates are a form of specimen drawing that either act as a basis for your finished work, or set up the required drawing environment.
2. The main advantage of using templates is that all elements and settings of an already existing drawing can be adopted and as such do not need to be re-entered.

This is particularly useful for parameters such as drawing boundaries, frames, guidelines, measurement settings, views etc. In order to facilitate this, a number of local variables containing stored settings are loaded along with the template. These settings are then used to display the drawing, drawing elements and objects.

For a listing of commands that can be used in template drawing setup, please refer to Drawing Setup. Alternatively, if you have a drawing file that has all your standard setup elements in it, you can simply delete any drawing elements you do not wish to have in your template drawing and SAVEAS a new drawing file, to be used as the template.

Further information on the use of template drawings can be found in Chapter 4 of the Customization and Programmer's Guide.

### Open an Existing Drawing...



**Command: OPEN**

**Alias OP**

#### Located:

1. **Pull-down Menu: File - Open**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

In order to continue working on an existing drawing in a later working session, the user must open the drawing. The term "Load" a drawing will also refer to "opening" a drawing.

The command **Open...** in the menu *File*, is used to do this. This command can also be activated by keyboarding OPEN or by clicking the standard symbol *Open* in the Function Bar.

After the entering the command, the user will see the selecting window, and may type in or select a drive, a directory, a subdirectory if it is required, and the File Name.

The file-type box offers the user the option to display AutoCAD .dwg, .dxf, AllenCad .flx drawing files, or all three.

### **Read-Only Option**

The user may activate the option Read-Only in the dialog box used to open files. To do that, click on the control box beside Read-Only. A cross inside of that box means that this function is active.

The function Read-Only allows the file to be opened in order to be read, but it may not be modified. This is a protective mechanism against inadvertently (or, perhaps, by an unauthorized user, purposefully) modifying or changing the file. After opening in Read-Only, the file can be modified in any way, but the modifications will not be saved.

If the user should try to save a Read-only file, the program will send an error message and ask, at the same time, if the file should be saved under another name. If the changes are to be saved, then the user should answer <Yes> and type another file name in the dialog box that appears.

### **Find...**

If the user does not know the exact name or the storing location of a drawing or file, she or he can look for it by using the option **Find**. The dialog box *Find File* will be opened. Select the drives to be checked. Enter the file name of the drawing to be found. The user may use the known wild cards <\*> (for any group of characters) and <?> (for any single character).

After the entries are confirmed the program will search for corresponding file names in the indicated drives. All files matching the search criteria will be listed in a result window. The size, construction date and time as well as the location will also be indicated. The user should select the desired file from this list and confirm his or her choice.

---

### **See also:**

NEW  
SAVE  
SAVE AS..  
SAVE ALL  
CLOSE

### **Close a Drawing**



**Command: CLOSE**

**Alias CL**

### **Located:**

- 1. Pull-down Menu: File - Close**
- 2. Top Function Bar**
- 3. Or Enter Command or Alias at the Command Line**

Closing a drawing indicates that the work on this drawing is finished. The program remains open. If a

drawing has been changed before closing there will be a prompt asking whether the file should be saved. This is to prevent unintended loss of work.

If the user decides to save the drawing, the program will automatically switch to the standard routine to save the drawing under a new name.

Select the command CLOSE in the menu *File* or press the keys CTRL+F4. It is also possible to close the drawing window, as in all Windows programs, by double clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

---

**See also:**

OPEN  
NEW  
SAVE  
SAVE AS...  
SAVE ALL  
Save Drawing File



**Command: SAVE**

**Alias S**

**Located:**

1. **Pull-down Menu: File - Save**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

**Save**

The command SAVE causes the drawings in the active window to be saved. At this time the file will be saved under its present name. The file is "saved" in the same drawing format and to the same storage device and area as was specified at the time of opening. The file last saved in this area will be replaced.

The command to save can also be invoked from the menu *File* or by clicking on the symbol *Save* in the Function Bar.

Local variables will be saved along with the drawing and the enclosed drawing entities current at the time the automatic save occurs. Such settings as the current coordinate system, the dimensioning mode and other adjustments will remain at the user's disposal.

**Auto-Save**

To avoid unforeseen loss of data we strongly recommend periodically saving drawings while working on them. For this purpose use the function Save Automatically that can be activated in the configuration dialog window. While an automatic save is occurring the current drawing will not be closed.

---

**See also:**

SAVE AS...  
SAVE ALL  
OPEN  
NEW



CLOSE

## **Save As...**

**Command:** SAVEAS

**Alias** SA

**Located:**

1. **Pull-down Menu:** File - SaveAs
2. **Or Enter Command or Alias at the Command Line**

The command SAVEAS saves the drawing in the current window, as the command SAVE does. However, it allows the user to save the drawing as a different type, name, or in a different area. The command SAVEAS will require that the user answer these questions, all contained in the dialog box:

Filename

File Type

Drive

Folder

The command will then save the work done as a specific file type (.dwg, .dxf or .flx), under a new filename, or in a new storage area. At the same time, a small bitmap (thumbnail) image of the drawing file is also created and stored with the file. This enables users to view the image before opening the file, thereby simplifying file identification.

---

**See also:**

SAVE

SAVE ALL

CLOSE

## **Save All**

**Command:** SAVEALL

**Alias** SAL

**Located:**

1. **Pull-down Menu:** File - SaveAll
2. **Or Enter Command or Alias at the Command Line**

The SAVEALL command will not only save the drawing in the current window but also all other open drawings.

---

**See also:**

SAVE

SAVE AS...

CLOSE

## **Import DXF-File**

**Command: DXFIN**

**Alias DXN**

**Located:**

**1. Entered Directly at the Command Line**

The command DXFIN reads a DXF file specified as argument to the command and opens the file as FLX drawing.

DXFIN is performed at command line and is provided especially for Lisp programming.

Specify the name of the file to be imported. If necessary, enter the exact path of the import file.

**Note:** DXF files can also be opened as a new drawing, using the OPEN command, or inserted as a part, using the INSERT command.

## Export DXF-File

**Command: DXFOUT**

**Alias DXO**

**Located:**

**1. Entered Directly at the Command Line**

The command DXFOUT allows you to export the current drawing as DXF file.

DXFOUT is performed at command line and is provided especially for Lisp programming.

Specify the name and the exact storing location for the export file.

## Viewing the Drawing

### Redraw



**Command: 'REDRAW**

**Alias R**

**Located:**

- 1. Pull-down Menu: Window – Redraw**
- 2. Left SideBar**
- 3. Palette: Zoom**
- 4. Or Enter Command or Alias at the Command Line**

Refresh current drawing window.

Regenerate

**Command: REGEN**

**Alias RG**

**Located:**

- 1. Pull-down Menu: Window – Regenerate**
- 2. Or Enter Command or Alias at the Command Line**

Regenerate current drawing.

## Regenerate All

**Command:** REGENALL

**Alias** RGA

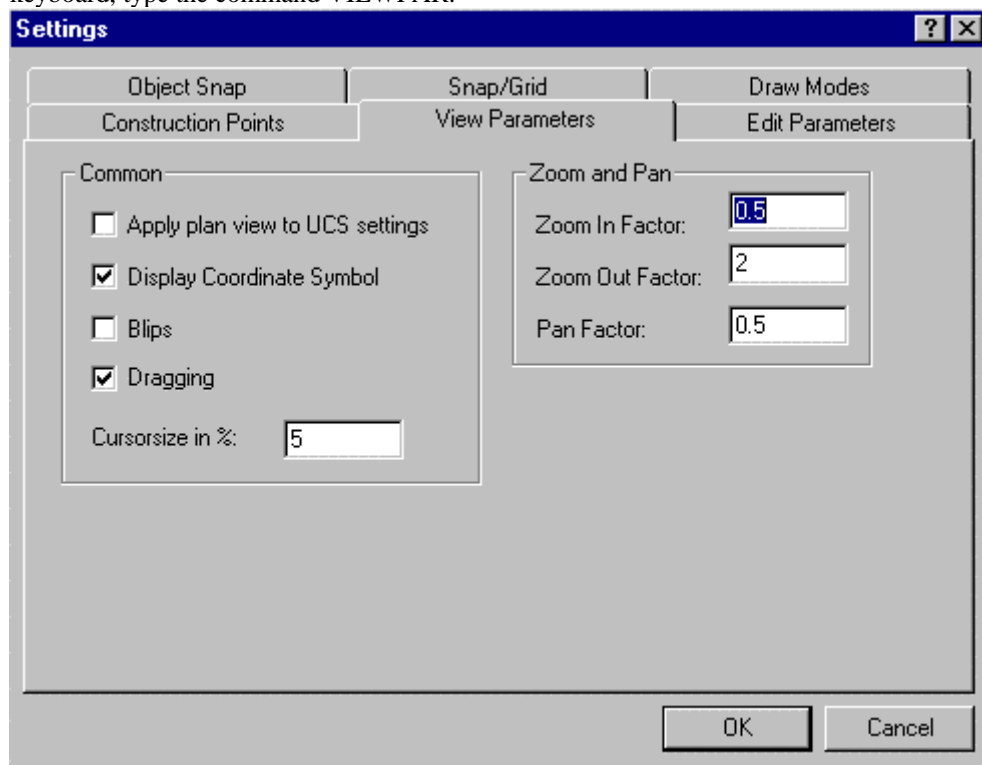
**Located:**

1. **Pull-down Menu:** Window – Regenerate All
2. **Or Enter Command or Alias at the Command Line**

Regenerate all drawings.

### Parameters for the Zoom and View Commands

To activate the function *View Parameters* choose the item of that name from the menu *View*. Or, using the keyboard, type the command VIEWPAR.



#### Zoom Factor

In the entry fields Zoom In Factor and Zoom Out Factor the user may choose the factor by that the drawing on the screen is to be enlarged or reduced. These factors affect only the visible portion of the drawing.

#### Pan Factor

In the entry field Pan Factor the user may choose the factor by that the visible portion of the drawing is moved using the command PAN. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area.

### Transparent Commands

A series of commands and functions that control your view of the drawing or that provide information

about the (about to be) selected objects are at your disposal even when another command has been called and is being executed. These commands are called transparent commands.

They include commands and functions for:

1. Changing the configuration of the drawing area (for example, 'ZOOM, 'PAN) or the window functions (for example, 'WCASCADE, 'WTILE);
2. Controlling precision ('GRID, 'SNAP, 'ORTHO, Object snap functions);
3. Switching between Command Line area and Text Window ('TSCREEN);
4. Loading or closing palettes ('PALMAN);
5. Calling the info functions ('EINFO, 'AREA, 'DIST); and
6. Calling the Help function ('HELP or '?)

After the execution of the transparent command, the program continues to execute the original command.

If transparent commands are called using the Command Line area, an apostrophe must be placed before the command for it to be treated transparently (for instance, 'ZOOMOUT).

Otherwise, transparent commands can be called as usual by clicking the proper option in the pull-down menus or by choosing the proper symbol, where the apostrophe is implied.

Transparent commands cannot be executed if:

1. The program is performing input of a sequence of characters; or
2. A dialog or selection window of another command is opened.

## Transparent Alias Commands

While a command is active, transparent alias commands can be issued by immediately prefacing the alias with an apostrophe, in a similar fashion to non-transparent alias commands.

See the example of the transparent alias command 'Z below:

Definition in the PCAD5.key file:

Z=ZOOM

> LINE

From point:

To point:

To point: 'z

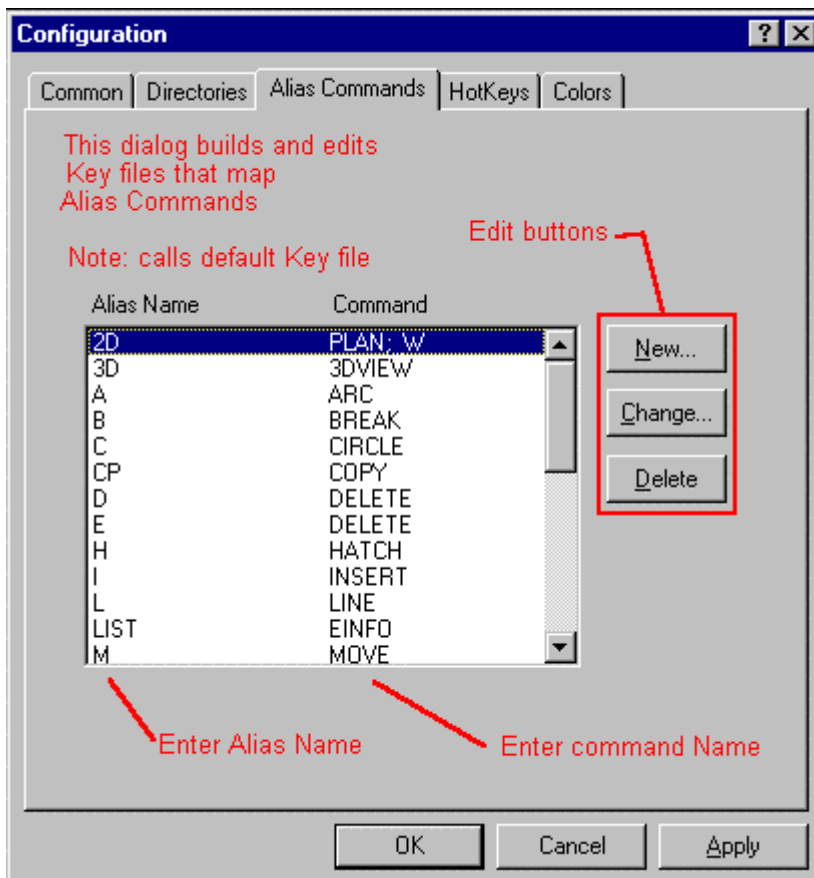
Zoom scale factor: Window

First corner:

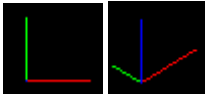
Second corner:

To point:

Note: Alias Commands are defined under the Alias Commands tab on the CONFIG dialog.



## Coordinate Symbol



Normally, a coordinate symbol is in the lower left corner of the Drawing Area. This symbol indicates the position of the axes of the coordinate system and provides orientation of the alignment of the current coordinate system. This change in orientation can be seen in the two images above, where the normal planview icon on the left changes to a representation such as on the right, when a 3-dimensional view of the drawing model is selected.

The user can classify the individual axes of the coordinate symbol by different colors that increase the clarity of this symbol.

In addition, the coordinate symbol can be faded in and out by the transparent command TICON.

## 3D View Control...



**Command: 3DVIEW**

**Alias 3V**

### Located:

1. **Pull-down Menu: View – 3D View**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the View menu allows you to define the view parameters. .



The program uses as its standard viewing angle, the top view (also called Plan View). At this angle the x-y-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views.

Along with the ability to view the drawing from any vantage point, it is also possible to rotate the view in the X – Y plane. You may want to do this, for instance, when orienting your drawing for printing.

---

**See also:**

SETVIEWDIR command

View Manager: Saving / Retrieving Views

### Command DVIEW

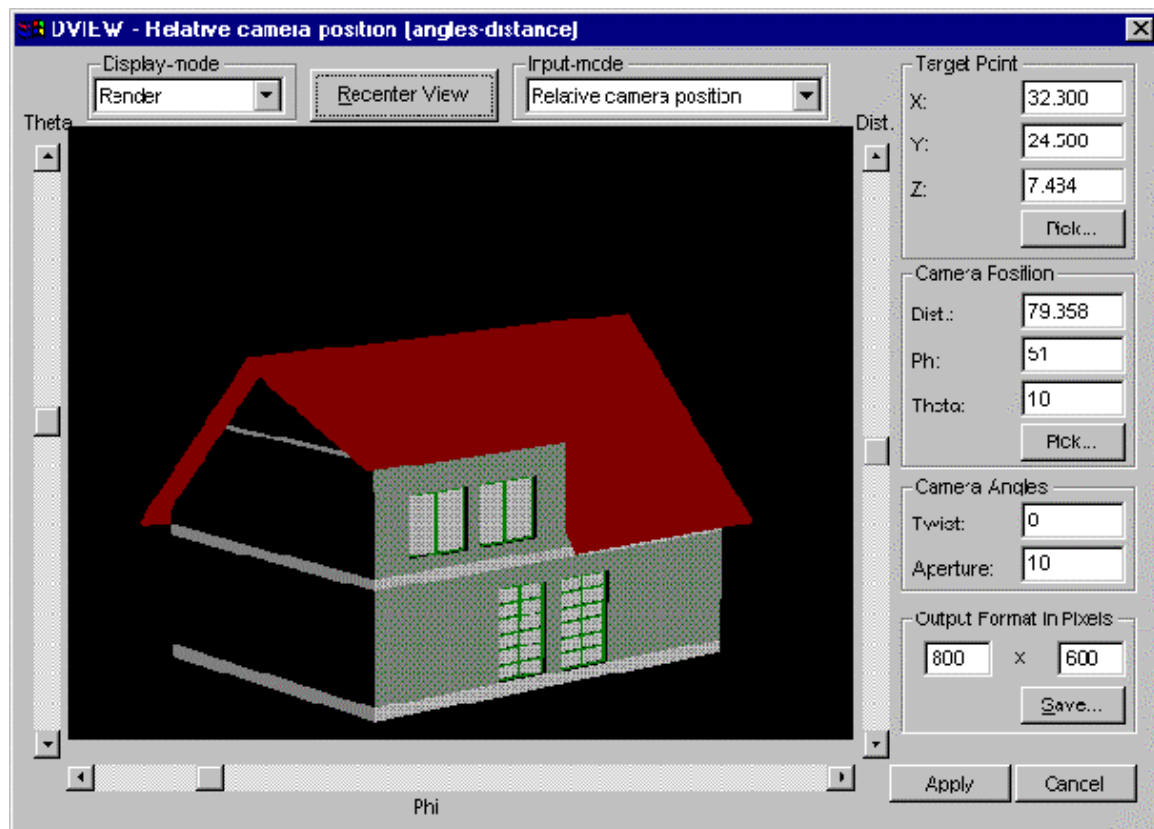
**Command: DVIEW**

**Alias DV**

#### Located:

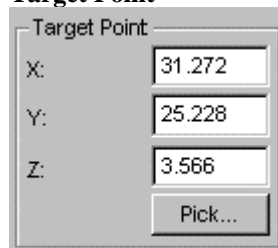
1. **Pull-down Menu: View – Dynamic View**
2. **Or Enter Command or Alias at the Command Line**

The **DVIEW** command creates a dynamic 3D-perspective view using different display modes. The command shows a dialog with a preview window. It is possible to save this view as a separate bitmap (BMP) file as well as transferring this view to the drawing.




The functionality of DVIEW in detail:

#### Target Point



This is the point the camera is aimed at. The x,y,z coordinates can be set either by entering values in the appropriate edit boxes or by picking (**pick...** button) a point in the drawing using the standard snap functions.

### Camera Position



Camera Position

Dist.: 99.783

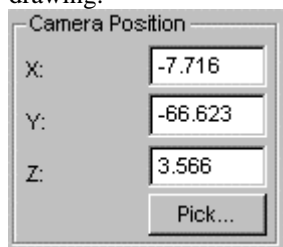
Phi: 247

Theta: 0

Pick...

This is the position of the camera relative to the target point, in other words, the position the drawing is viewed from. The distance (**Dist.**) as well as longitudinal (**Theta**) and latitudinal (**Phi**) settings can be set in this section.

The **Theta** and **Phi** values correspond to a sphere of radius **Dist.** centered around the target point. Again, these values can be adjusted by entering the values in the edit boxes or by picking a point in the drawing.



Camera Position

X: -7.716

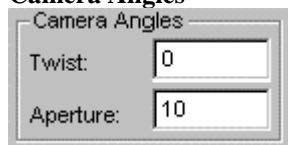
Y: -66.623

Z: 3.566

Pick...

Alternatively, the absolute camera position can be set by selecting the **Absolute camera position** entry in the **Input** drop-down box pictured above.

### Camera Angles



Camera Angles

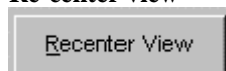
Twist: 0

Aperture: 10

**Twist:** rotates the camera while facing the target point.

**Aperture:** changes the aperture of the camera.

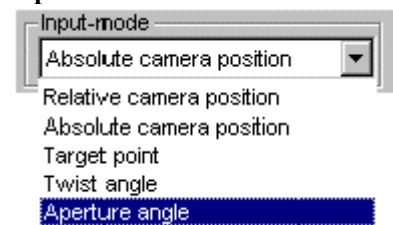
### Re-center view



Recenter View

This button re-centers the view. Pressing this button shows a good starting point for the rotation. Note: when you first issue the DVIEW command, you may need to re-center the view to bring the drawing into view.

### Input Mode



Input-mode

Absolute camera position

Relative camera position

Absolute camera position

Target point

Twist angle

Aperture angle

The input mode changes the function of the sliders on the preview window. The different modes are:



**Relative camera position:** The sliders set the relative camera position values using Phi/Theta/Dist.

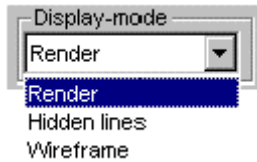
**Absolute camera position:** The sliders set the absolute camera position.

**Target point:** The sliders set the absolute target position.

**Twist angle:** A single slider changes the twist of the camera.

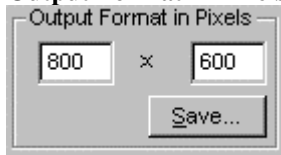
**Aperture angle:** A single slider changes the aperture of the camera.

### Display Mode



Three display modes are supported: Render, hidden line and wire frame.

### Output Format in Pixels



The 3D-view displayed can be exported as a bitmap file at the resolution entered in the edit boxes.

**Save ...** opens a standard save file dialog.



**Apply** assigns the changes to the view to the drawing coordinate system.

Please note that panning of objects (pressing the mouse in the picture and then dragging the mouse to a new location) in the preview does not result in a panning of the object in the actual drawing as only changes in the perspective view are transferred to the drawing.

## View Manager... (Saving / Retrieving Views)



**Command: VIEW**

**Alias V**

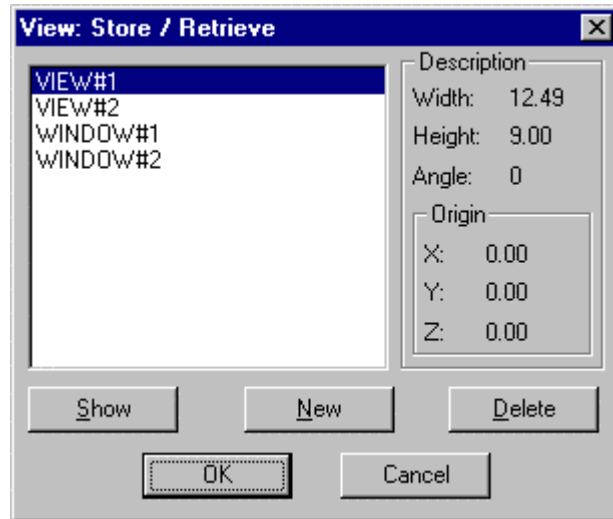
### Located:

1. **Pull-down Menu: View – View Manager**
2. **Left Side Bar**
3. **Palette: View**
4. **Or Enter Command or Alias Directly at the Command Line**

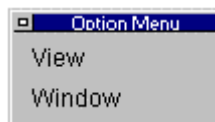
It is convenient when working in either Model mode or in Layout Mode to save different views of the drawing for later retrieval and display. The view of the drawing is essentially the viewer's 3-dimensional location and the direction of view toward the drawing. These are the elements that are stored in the named file and they correspond to the current drawing view when the view is saved.

### Save a Named View

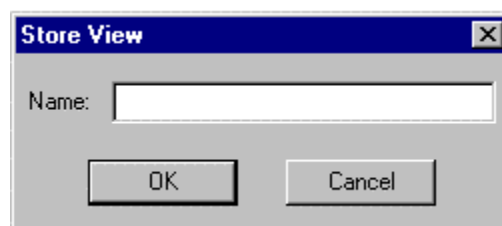
When selecting the option VIEW, the present portion will be saved as a user-defined view. Upon issuing the command, you will see the dialog:



Here we can see that we already have two views and two windows already saved. To save another, simply click on the **New** button. At this time you will be presented with the option:



If you choose view, the full screen image will be saved, whereas if you choose Window (a windowed view), you will be asked to define the window onscreen. This is done by picking upper left and lower right (or any diagonally opposite) window corners. After selection of option is complete and the window is defined, you are lastly presented with the dialog:



At this point, you specify the name for your view or window, and it will then be added to the list of other drawing views for later retrieval.

### Retrieve a Named View

A previously saved view can be retrieved through the same function, and can then become the present view. Follow these steps:

1. Choose one of the saved views in the dialog's list box;
2. Click on the **Show** button, located in the View Command dialog image above. The selected view will then become the present view;
3. Close the dialog box with OK to accept to display the chosen view. Choosing to abort will retain the previous view.

### Delete the Name of a Defined View

Views that are no longer needed, can be removed from the table of named views as follows:

1. Choose one of the names of a defined view in the list box;
2. Click onto the **Delete** button, located in the View Command dialog image above. The view will be removed from the list immediately.

### View Parameters...



Command: VIEWPAR

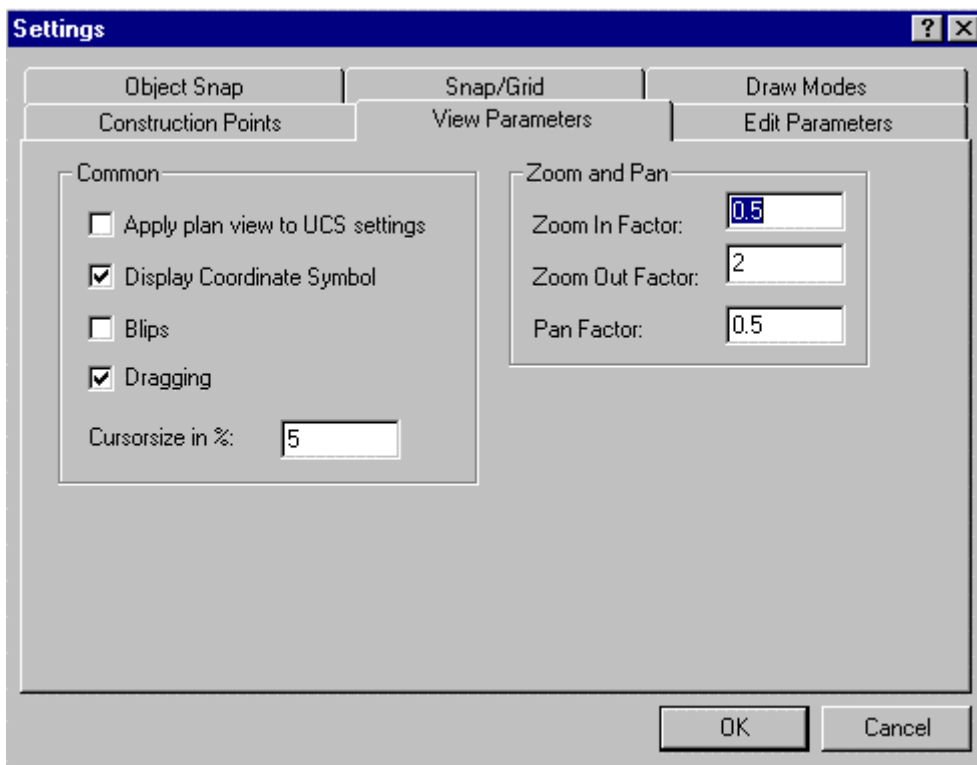
Alias VR

#### Located:

1. **Pull-down Menu:** View – View Parameters
2. **Palettes:** Zoom and View
3. **Or Enter Command or Alias at the Command Line**

Access to the View Parameters settings can be made either directly, as described above, or indirectly through the SETTINGS command. As the dialog illustrated below shows, View Parameters has its own tab on the Settings dialog, which the dialog opens to when the VIEWPAR command is issued. Various parameters may be selected to define the direction and general size and shape of a given view of the drawing.

There are also default factor values inputs for the **ZOOM**ing and **PAN**ning of the view on the screen.



### Dialog Control Elements

#### Common

Here you see selections to be made that determine visibility of the coordinate symbol, pick blips (blipmode) and onscreen dragging representation. Here too you can specify the application of plan view to UCS settings and also control the size of the cursor symbol. Note that setting this value to 100% will

extend the cursor axes across the entire visible screen area. This can be a helpful feature when aligning drawing objects visually onscreen.

### **Zoom and Pan**

In the entry field *Zoom In Factor* and *Zoom Out Factor* the user may choose the default factor by which the drawing on the screen is to be enlarged or reduced. These factors affect only the visible portion of the drawing.

In the entry field *Pan Factor* the user may choose the factor by that the visible portion of the drawing is moved using the command PAN. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area.

## **Copy a Drawing Viewport**

**Command:** QWOPEN

**Alias:** QW

### **Located:**

1. **Pull-down Menu:** Window – Copy Current Viewport 1:1
2. **Or Enter Command or Alias at the Command Line**

The option **Copy Viewport** on the *Window* menu allows you

1. to make a quick copy of the current viewport or
2. a plan view, by choosing the desired option button.

### **Fast Hide**



**Command:** FHIDE

**Alias:** FHD

### **Located:**

1. **Pull-down Menu:** View – Hidden Lines – Fast Hide
2. **Palette:** View
3. **Or Enter Command or Alias at the Command Line**

In contrast to the HIDE command, which creates a new drawing, the command FHIDE displays a hidden line view in the current drawing.

This representation is maintained until the drawing window is refreshed by a REDRAW, REGEN, ZOOM, or PAN command. This hide command offers you a visual impression of the 3D model from the current viewpoint.

Note that you can continue to draw in this display mode, even utilizing the object snap functions.

The command FHIDE is based on the OpenGL system from Microsoft that allows high performance hidden line removal and rendering.

---

### **See also:**

HIDE

Command DVIEW

DRAWORDER

## Viewing in Layout Mode

In Layout Mode, many new rules and restrictions apply to commands that you are familiar with in Model mode. This different working environment has its own viewing rules as well. Objects *drawn* in Layout Mode can be viewed much as you are familiar with in Model Mode, however any views of drawing elements produced in Model mode must be viewed through special windows called Viewports.

A special section entitled Model Mode / Layout Mode a little later on in this Help file documents a number of issues involved when working in these two modes and users are encouraged to review this section to better gain an overview of issues and viewing restrictions that apply.

## Using Viewports, Windows and Views

These three AllenCad concepts or features may be a bit difficult for the new user to grasp at first, so the following distinctive characteristics of each should be noted:

- **VIEWPORTS** are strictly Layout mode (paper space) drawing "pictures" of the Model mode drawing. These "pictures" can be 3-dimensional and can be grouped (as many as you like) within a surround and title drawn in Layout mode and plotted in layout mode as a single drawing.
- **Windows** (Commands WOPEN, WCLOSE, WTILE, WCASCADE), on the other hand, are strictly a model mode (or space) feature. The initial frame in which the drawing edit area resides when the program opens is the initial window. Other windows, offering unique working outlooks, can be opened in the same drawing. Drawing windows allow you to jump back and forth and work on separate parts of the drawing without needing to pan/zoom back and forth across your drawing. The "outlook" in each window can be 3-dimensional, as required. You can only print/plot from within a single window and you can have a maximum of four windows for any drawing at any given time.
- **Views** – (Command VIEW) These are simply - named 3 dimensional views of the drawing that are defined by vantage point and direction of view. View settings are stored in memory in this named file and can be called into drawing windows or viewports to specify the specific vantage point and drawing view as required. A single View is visible within a viewport or window at any given time.

## Drawing Windows



### Windows

AllenCad is capable of opening as many as four drawings at the same time. Each of these drawings can be viewed in its own window. Further, it is possible to open as many as four viewing windows for each drawing so that the drawing may be seen from different angles or so that portions of the drawing can be examined. A maximum of sixteen drawing windows can then (theoretically) be active at any one time.

### Opening Drawing Windows

In order to open a drawing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing. Open the menu *Window* and choose the option *Open Another Viewing Window* (command WOPEN). Before the program opens another window, you will be asked to determine the type of view in this new window. Choose a standard view from the menu, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

### Changing Drawing Windows

There are two ways to change the "active" window:

- ▶ Clicking on any part of the chosen window with the cursor will activate it.
- ▶ Open the menu *Window*, and find where all open drawing windows are listed. The active window will be marked by a check in front of its name. When a new window is opened, a number in front of the specific entry marks it.

#### **Closing Drawing Windows**

It is possible to close the viewing window, as in all Windows programs, by double clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

#### **Viewpoint / Angle of View**

The direction from that an object is viewed influences the view of the object, and may be changed in many ways inside one drawing window. Each additional view will be portrayed in another viewing window. It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the pull-down menu VIEW provides the facility to make that definition.

#### **Saving and Retrieving Views**

It is convenient when working with different drawing portions to save, and later, to retrieve changes in angles of view, different drawing portions and scales of view. This may be accomplished by using the command VIEW.

See also

Zooming and Panning

Parameter for View Commands

## Open New Drawing Window

**Command: WOPEN**

**Alias WO**

#### **Located:**

- 1. Pull-down Menu: Window – Open New Viewport**
- 2. Or Enter Command or Alias at the Command Line**

In order to open a viewing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing.

From the menu *Window* choose the option **Open Another Viewing Window....**

Before the program opens another window you will be asked to determine the type of view in this new window. Choose an individual viewpoint or a standard view from the dialog box, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

## Close Drawing Window

**Command: WCLOSE**

**Alias WC**

#### **Located:**

- 1. Pull-down Menu: Window – Close Current Viewport**
- 2. Or Enter Command or Alias at the Command Line**

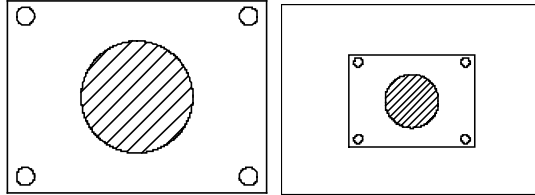
It is possible to close the current viewport, as in all Windows programs, by double-clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

## Zooming

### Zooming and Panning

The ZOOM and PAN commands are transparent ones that may be executed while other commands are being executed. It is possible, for instance, to execute the command PANDOWN (move downward) while entering points for drawing an element.

#### Zooming

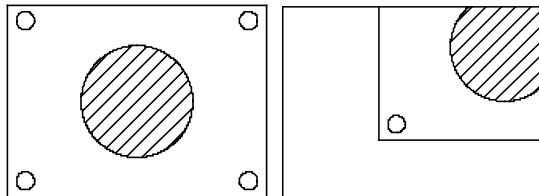


The ZOOM commands change the viewing scale of the drawing, or of the portion of the drawing chosen by the user. The absolute ratios of scale of the entire drawing to the included objects chosen remains unchanged.

These variations on the ZOOM command are possible:

- ▶ Zoom Window ZOOMWIN
- ▶ Zoom Back ZOOMBACK
- ▶ Zoom Fit ZOOMFIT
- ▶ Zoom In ZOOMIN
- ▶ Zoom Out ZOOMOUT
- ▶ Zoom Factor ZOOMFAC

#### Panning



The command PAN moves the visible portion of the drawing to the right, to the left, upward or downwards. It is now easy to look at objects that were previously not visible because they were off the viewing surface. In contrast to the ZOOM command, the drawing scale does not change.

These variations on the PAN command are possible:

- ▶ Portion to left PANLEFT
- ▶ Portion to right PANRIGHT
- ▶ Portion up PANUP
- ▶ Portion down PANDOWN

### Wheel Mouse Menu with Dynamic Panning and Zooming

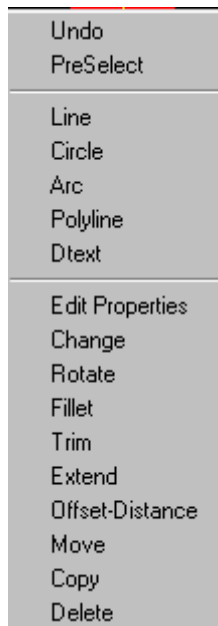
If you have a Microsoft (or compatible) wheelmouse, you have the ability to zoom in and out on the drawing simply by turning the mouse wheel. With the wheel, you can get the exact amount of zooming that you need.

Zooming is based on the current center point in your display, so you may wish to pan so that the area of interest is at the center of your screen before zooming in or out using the mouse wheel.

Using the Alt key in combination with the left mouse button, you also have access to Dynamic Panning, without the need to issue a DPAN command.

The wheelmouse also provides an additional productivity feature – the context menu. By pressing down on the mouse wheel, you call a special drawing and edit menu right onto the screen, saving you the bother of

finding the command icons or selections on the desktop or in the menus. This menu contains most of the more commonly used drawing, edit and configuration commands, as can be seen in the following graphic:



To exchange commands on the context menu, you can edit the AllenCad.key file (in the AllenCad \Cfg folder) with a text editor and change the command names listed in the Context Menu section to suit your individual needs. Before editing the key file, be sure, however. To save a backup copy of the file to assist in error recovery, should this be needed.

## Zoom

The ZOOM commands change the viewing scale of the drawing, or of the portion of the drawing chosen by the user. The absolute ratios of scale of the entire drawing to the included objects chosen remains unchanged.

These variations on the ZOOM command are possible:

Zoom Window	ZOOMWIN
Zoom Back	ZOOMBACK
Zoom Fit	ZOOMFIT
Zoom In	ZOOMIN
Zoom Out	ZOOMOUT
Zoom Factor	ZOOMFAC

The ZOOM commands, like most other commands for view control, are transparent commands that can be called up and executed at the same time another command is being executed.

For example, it is possible to execute the command ZOOMOUT while at the same time entering points when drawing an element. The screen plot will be enlarged by a pre-determined factor.

```
> LINE
From point: P1
To point: P2
To point: 'ZOOMOUT
To point: P3
To point:
```



---

**See also:**

DZOOM command

PAN command

## **Zoom Window**



**Command: ZOOMWIN**

**Alias ZW**

### **Located:**

1. **Pull-down Menu: View – Zoom Window**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMWIN command allows a chosen portion of a drawing to be shown in the largest possible scale. The drawing portion is specified by a box. Identify opposite corner points with the cursor or enter the coordinates manually.

> ZOOMWIN

First corner: **P1**

Second corner: **P2**

---

**See also:**

ZOOM command

## **Zoom Back**



**Command: ZOOMBACK**

**Alias ZX**

### **Located:**

1. **Pull-down Menu: View – Zoom Back**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The command ZOOMBACK allows you to undo the last ZOOM command and recreates the prior drawing portion. The deleted scale is saved as the last used scale and may now be called again.

Using this command allows the user to switch quickly between two definite portions in order to view an object as a whole, or to see its details more clearly.

It is still possible to recreate the last scale if other operations have been performed.

---

**See also:**

ZOOM command

Zoom to Fit



**Command: ZOOMFIT**

**Alias ZE**

**Located:**

1. **Pull-down Menu: View – Zoom Fit**
2. **Palette: Zoom**
3. **Or Enter Command or Alias at the Command Line**

The command ZOOMFIT allows the user to show the drawing with all its elements as large as possible on the viewing area.

The user should remember that points of deleted or modified elements that may still be saved in the drawing can influence the view resulting from this command. These points will be completely deleted when a drawing is regenerated. Thus, before zooming to the largest possible drawing size, the user should consider doing a REGEN.

---

**See also:**

ZOOM command

**Zoom In - Nearer to Objects**



**Command: ZOOMIN**

**Alias ZN**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom In**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMIN function enlarges the viewing window contents with each successive use by a factor determined by a given value. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

**Zoom Out - Away from Objects**



**Command: ZOOMOUT**

**Alias ZT**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom Out**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMOUT function is the opposite of the zoom-in function. It reduces the size of viewing window contents with each successive use by a factor determined by a given value. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

Zoom Factor

**Command: ZOOMFAC**

**Alias ZF**

**Located:**

1. **Pull-down Menu: View – Zoom Factor also**  
**View – Zoom In/Out/Pan Direct – Zoom 4.00 and Zoom 0.25**
2. **Or Enter Command or Alias at the Command Line**

The command ZOOMFAC allows the user to zoom with the specified scale factor. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

## Command ZOOMPAGE

**Command: ZOOMPAGE**

**Alias ZP**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom Page Limits**
2. **Or Enter Command or Alias at the Command Line**

The **ZOOMPAGE** command zooms out to the page size defined for your printer (or plotter).

## Dynamic Zoom



Command: 'DZOOM

Alias DZ

**Located:**

1. **Pull-down Menu: View – Dynamic Zoom**
2. **Left Sidebar**
3. **Or Enter Command or Alias at the Command Line**

With the command **DZOOM** you can zoom in or out to any extent within the current drawing window.

To zoom in this manner, choose *Dynamic Zoom* from the menu *View*. The cursor then changes to a magnifying glass icon with a plus sign and a minus sign. To start the dynamic zoom process, click the left mouse button and drag the mouse, either up or down. Whether you zoom in or out depends on the direction of movement of the cursor. To zoom in, drag up (in the direction of the plus sign). To zoom out, drag down (in the direction of the minus sign).

If you reach the edge of the drawing window, you can release the mouse button, move the cursor away from the edge, and left-click and drag again.

To exit the Dynamic Zoom mode, press **ESC** or **ENTER**. You can also right-click to open the context menu that contains the *Exit* option.

Alternatively, you can start another command via the pull-down menus toolbars or palettes to exit the dynamic zoom process (this works, as long as you do not call a transparent command from the menu or palette).

While executing the DZOOM command, you can also right click with the mouse and produce the following option dialog:



Beneath leaving the dynamic zoom operation, the cursor menu (that you can open by clicking the right mouse button) also allows you to switch to dynamic pan mode and to execute a Zoom Extents.

Dynamic Zoom can be used transparently (in the middle of another command).

---

**See also:**

ZOOM command

DPAN command

## Panning

Pan to the left



Command: PANLEFT

## Alias PL

### Located:

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Left**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved to the left. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

### See also:

PAN command

DZOOM command

### Pan to the Right



**Command: PANRIGHT**

## Alias PR

### Located:

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved to the right. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

### See also:

PAN command

DZOOM command

### Pan Up



**Command: PANUP**

## Alias PU

### Located:

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Up**

2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately shift upwards on the drawing. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR).

---

**See also:**

PAN command

DZOOM command

### **Pan Down**



**Command: PANDOWN**

**Alias PD**

#### **Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved down. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

**See also:**

PAN command

DZOOM command

### **Command DPAN**



**Command: 'DPAN**

**Alias DP**

#### **Located:**

1. **Pull-down Menu: View – Dynamic Pan**
2. **Left Sidebar**
3. **Palette: 3D View**
4. **Or Enter Command or Alias at the Command Line**

As described previously, panning means to move the display of the drawing in its current window without changing the magnification.

With the command **DPAN** you can simply grab the drawing and move it in any direction you want, any number of times, until you have the drawing view that you want. At this point, you terminate the command by pressing the ESC key or Enter (right mouse click) and then select the Exit option.

To pan the drawing in this manner, choose *Dynamic Pan* from the menu *View*. The cursor changes to an icon symbolizing a hand. Place the cursor anywhere in the current drawing, then left-click. Now drag in the direction you want the drawing to move, releasing the mouse button to release the "grip" on the drawing.

If you reach the edge of the drawing window, release the mouse button, move the cursor away from the edge, and left-click and drag again.

To exit the Dynamic Pan mode, press **ESC** or **ENTER**. You can also right-click to open the context menu that contains the *Exit* option. Alternatively, you can start a command via the pull-down menus or palettes to exit the dynamic pan (this works, as long as you do not call a transparent command from the menu or palette).

Beneath leaving the dynamic pan operation, the cursor menu (that you can open by clicking the right mouse button) also allows you to switch to dynamic zoom mode and to execute a Zoom Extents.

Dynamic Pan can be used transparently (in the middle of another command).

---

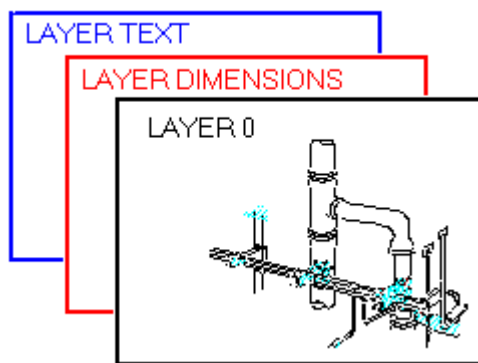
#### See also:

PAN command

DZOOM command

## Layer and Object Properties

### Layer - Basic Information



The concept of working with layers is analogous to manual drawing using layers of see-through Mylar or acetate. Parts or portions (that may be functional portions, such as construction lines) of one complete drawing are drawn on different "overlays" that, imposed on top of one another, form the entire drawing.

Layers are useful in separating objects, such as parts, sub-assemblies, or structural components such as pipe sizes or building materials. They also serve to separate elements such as construction lines, center lines, outlines, and dimension lines, or things that are to be placed in a group, such as foliage in an architectural drawing, or parking areas.

All parameters of a drawing that are active for the entire drawing, like the coordinate system, the Zoom factor, the view arrangement, fonts, attributes, and so forth, are valid for all layers and are always the same.

The advantages of creating a drawing in this fashion, whether working manually or electronically, lie in a clearer construction of the drawing and better control than drawing on a single foil.

It is possible to structure a drawing through formal or content characteristics.

The standard layer is always called Layer 0.

One layer in each drawing will be defined as the active layer. All new objects will automatically be placed on this layer. As long as Layer 0 is the only one existing, it will also be the active layer.

Each layer has a color and a linetype associated with it when it is created. Anything drawn on this layer will be drawn with this linetype and color.

Aside from the color and the linetype, there are other attributes of the layer that allow the user switch between "states". The states are visibility (on/off), freeze/thaw and lock/unlock.

The layers with their status and their attributes are part of the drawing. They are saved together with the drawing and can be called when re-opening the drawing.

See also:

Layer Attributes

Layer Manager

### Layer Manager



**Command: LAYER**

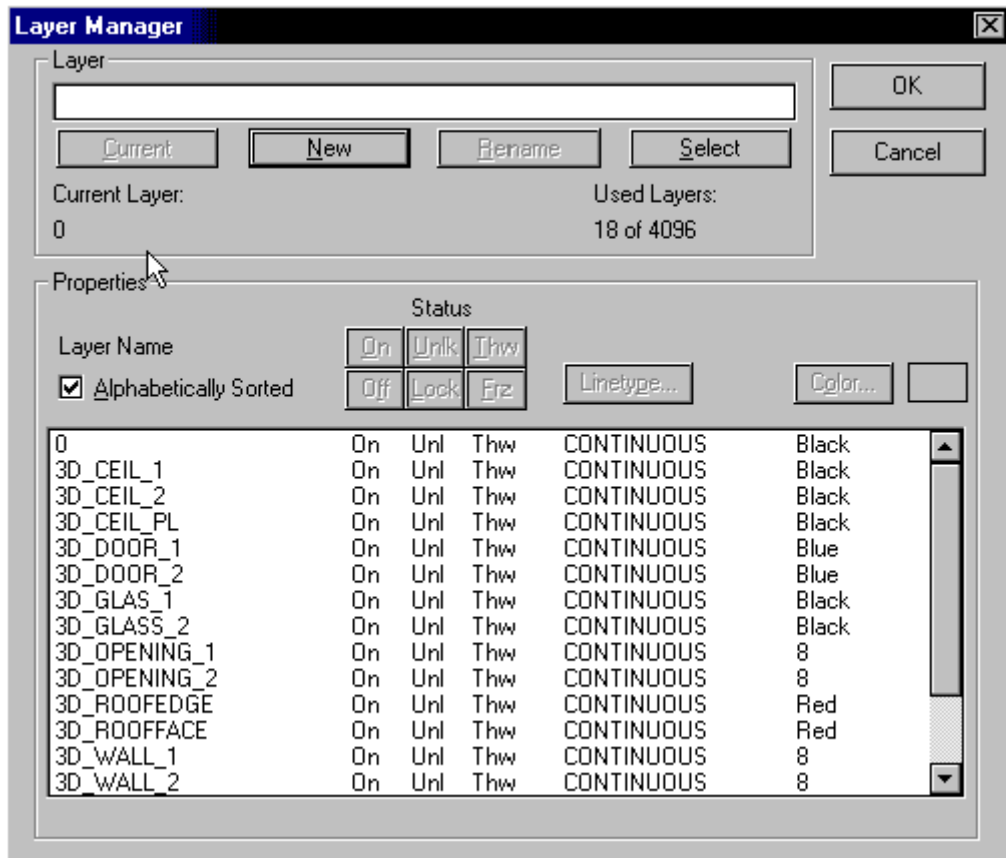
**Alias LA**

#### **Located:**

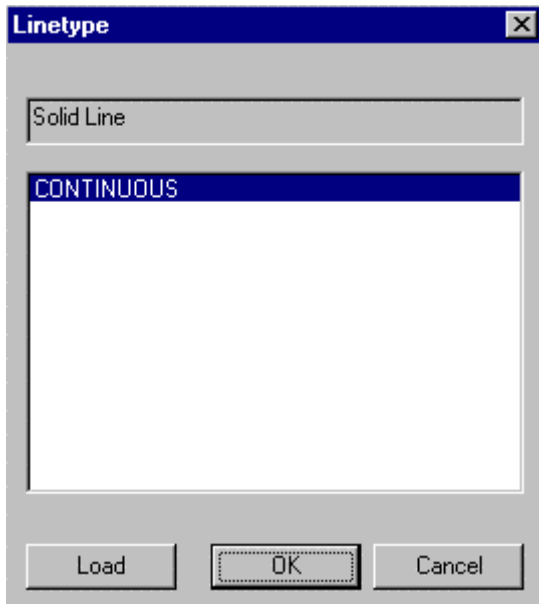
- 1. Pull-down Menu: Tools – Layer**
- 2. Properties Toolbar**
- 3. Or Enter Command or Alias at the Command Line**

Creating new layers, choosing an active layer and controlling all previously described layer characteristics are accomplished through a dialog box. This dialog box is called *Layer Manager*.

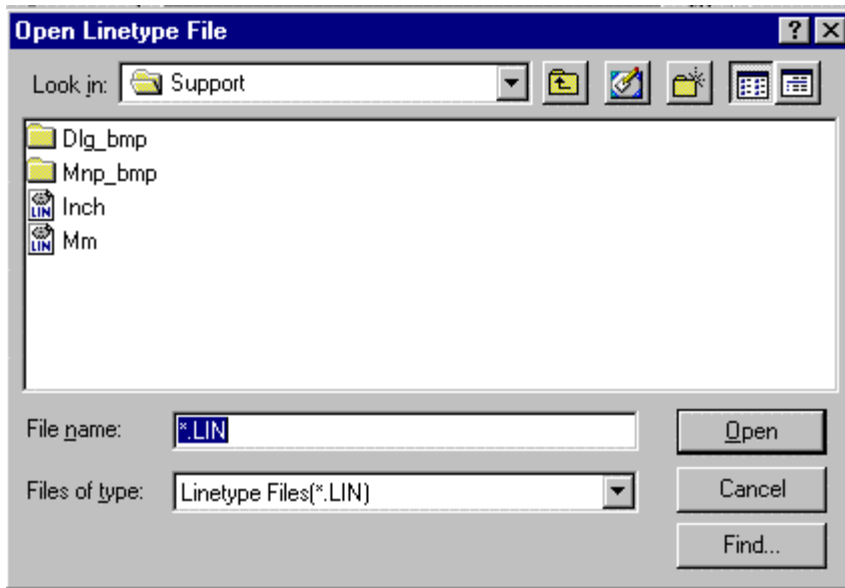




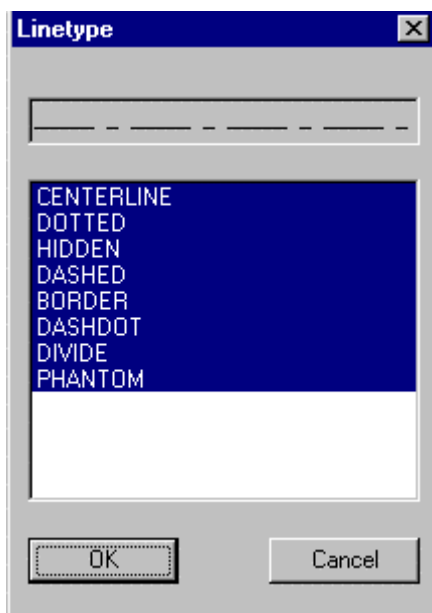
Line types may be loaded directly from the linetype dialog, in the Layer Manager dialog, by pressing the **Load** button

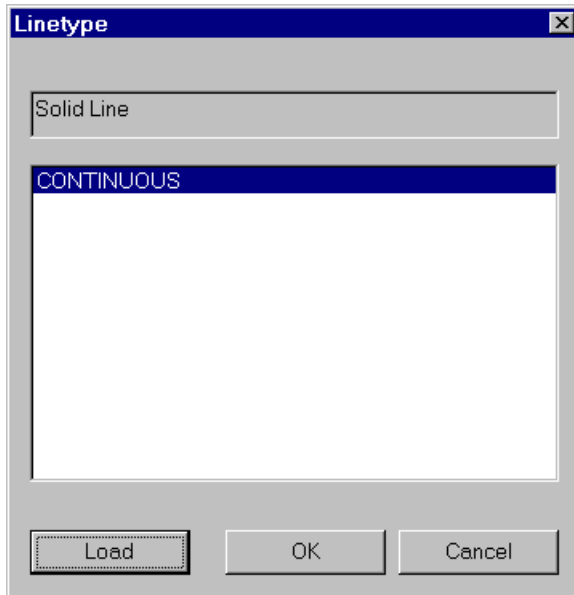


You will then be asked to select the linetype file (.lin) containing the linetype you want to open:



At this point, you are presented with the dialog below, showing all of the linetypes contained in the linetype file that you selected. As you select each linetype in turn, a representation of its appearance is presented in the window at the top of the dialog. Choose the linetype that you want loaded and press OK.





In the **Layer Manager** the default button is the **New** button. When you type in a layer name and press the return key, a new layer will be created and you remain in the layer manager.

### Current Layer and Used Layers

In the upper rows of the *Layer Manager* Dialog Box is a field for the name of the active (current) layer (left window) and a field for the number of defined layers in this drawing.

### Layer

Later rows contain combined view and entry fields for layers that still have to be worked on, and a selection box that allows the user to choose between creation of new layers, renaming layers and choosing an active layer.

### Properties

There are fields for the layer state and for choosing line types and layer color.

Directly beneath that, in the dialog box, all the existing layers of the drawing are shown. Any of these may be selected and activated with the cursor. The layer color and linetype are also shown there.

The meaning of the abbreviations used are shown here.

Abbreviation	Meaning
On	Visibility on - The layer becomes visible
Off	Visibility off - The layer becomes invisible
Lock	Status free, drawing elements can be edited
Unlk	Status closed, drawing elements cannot be edited
Thw	Layer thawed, visible and can be edited
Frz	Layer frozen, invisible and cannot be edited, will not be regenerated

### Changing the Active Layer

Choose the layer that you want to become active from the list in the Dialog Box. The name of the chosen layer will appear in the view *Current Layer* window;

Verify the choice of the new active layer by clicking on **ACTIVE**.

Check the name of the now active layer in the window. If the layer change has been performed correctly, the name of the chosen layer should appear there.

### Quick Layer Setting...

**Command: SETLAYER**  
**Alias STL**

**Located:**

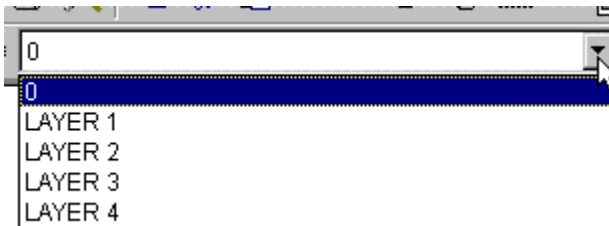
1. **Pull-down Menu: Tools – Set Layer**
2. **Properties Bar**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to switch the active layer. It is not possible to create a new layer or to change any characteristics of the existing layer through the selection of this button. These changes must be made in the fashion described previously in Layer Control.

The letters in front of the layer names show the status of the characteristics visibility, access, and thawing/freezing. They have the following meaning:

- ON = On (Visibility)  
OFF = OFF (Visibility)
- U = Unlocked (Access)  
L = Locked (Access)  
T = Thawed (Thawing/Freezing)

When changing layers on the Property Bar, click on the down arrow to the right of the layer window, to produce a drop-down window showing all drawing layers.



You can then select the layer you want to be current from this list. After selection, the drop-down window closes and the layer selected should appear in the collapsed window. This indicates that it is now the current layer.

**Note:** A frozen layer cannot become the active layer, so that choice will not appear for the current layer. Highlight a layer in the list with the cursor and verify your choice by clicking on OK.

---

**See also:**

Layer Manager

**Layer Attributes**

**Layer Name**

To identify individual layers and differentiate between them, each must have a distinctive name. The name can consist of up to 31 characters. They can be letters, numbers, or special characters like \_ (Underline) and \$ (Dollar). All characters are automatically capitalized.

**Color**

Every layer has a color that is user-defined. If a new color is not chosen, the layer will be colored black. The standard color of the layer can be marked in the color box as BYLAYER.

It is possible to choose the standard color of a layer from among 255 color shades. Every color has its own number.

Since it is possible to draw on the active layer only, objects still to be drawn will be in the standard color

of the active layer, unless a new color was chosen with the option Object Color. If the standard color of the layer is changed using the dialog box for layer control then all drawing elements of this layer that are marked with the color BYLAYER will assume the new color.

## Linetype

Similarly, there is a standard linetype that is assigned to the line. If you do not choose another linetype, the Linetype CONTINUOUS will automatically be assigned to it.

The chosen linetype is marked in the dialog box BYLAYER. All elements are drawn in the linetype of the layer unless another linetype is chosen for the element.

## Visibility

One of the major advantages of working with layers is that it aids the user in visualizing the part, or drawing. The visual clarity of a drawing can be improved greatly through choosing layers properly and "hiding" the drawing elements. Concentration on meaningful drawing elements makes the drawing and construction much easier.

The visibility of a layer can either be turned "on" or "off". Every layer can be turned "on" or "off" independent of any other layer(s). Changing the visibility is done through the Layer Manager.(command LAYER).

## Locking / Unlocking Layers

It is possible to "lock" layers so that the objects or elements contained within the layer may not be changed.

Normally, new layers will be created with the status *Lock*. There are no limits inside the normal program functions when working with an unlocked layer.

When a layer's status is locked it is not possible to change, add, or delete objects from this layer. The status must be changed to *Unlock* if the user desires to work with the objects of this layers. This function makes it difficult to make an unauthorized drawing change.

It is possible to draw new elements onto locked layers but it is not possible to work with them further.

Other options of layer control can also be affected when working with a locked layer. For example, the standard color of the layer may be changed. A locked layer may be designated active, visible or invisible, frozen or thawed.

## Thaw / Freeze Layers

The options Thaw and Freeze are functions that control the visibility as well as the influence of the objects within the layer.

A "frozen" layer is both invisible and locked, and is thus impossible to change. A frozen layer will not be included in drawing regeneration. When working with very complex drawings, freezing some unneeded layers can increase the speed of drawing considerably.

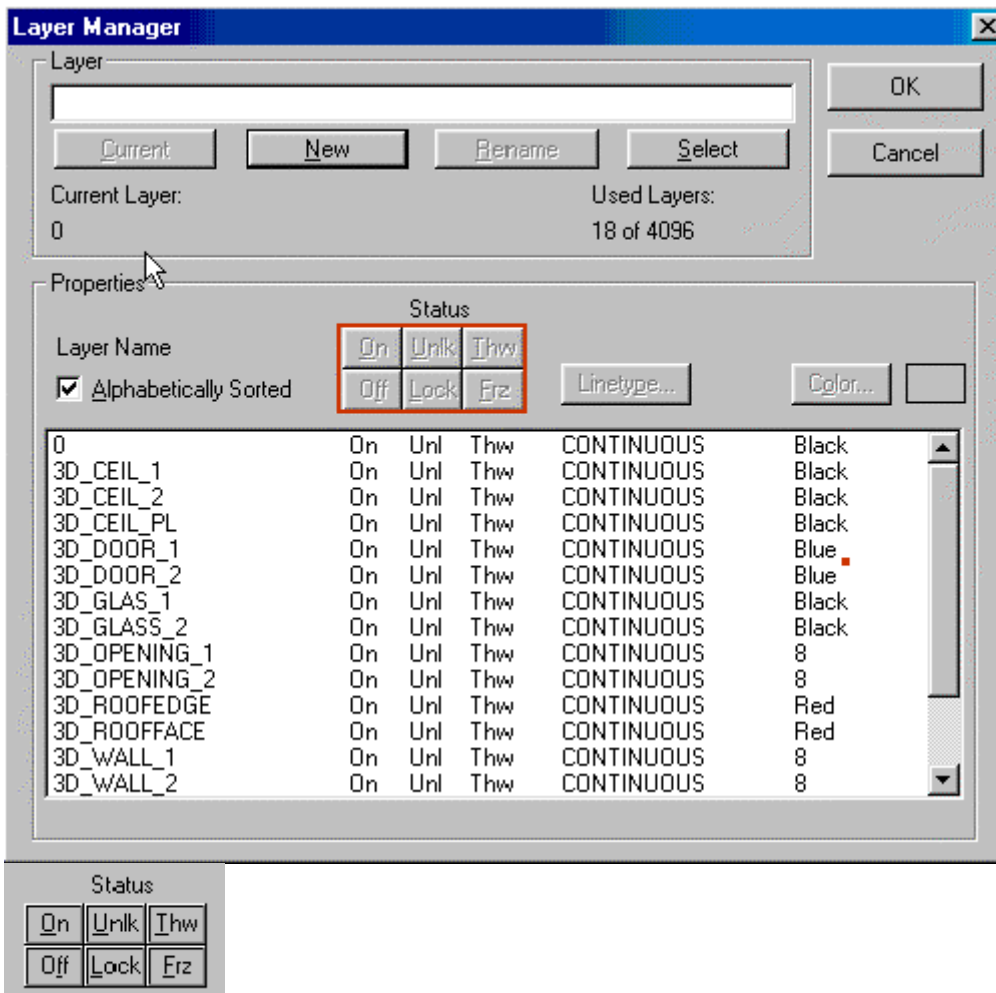
The option Thaw undoes the freezing of a layer. The layer will become visible and changes may be made once more. The active layer of a drawing cannot be frozen.

See also:

Loading Linetypes

### Controlling the Layer Status

Changing the status of a layer is accomplished by a clicking on the proper field in the Layer Manager dialog. Characteristics of each layer are displayed in the Dialog Box. When layers are selected in the window below, these buttons become available for use in modifying the status of the selected layers.



The layer status "ON - OFF" simply controls the display visibility of the layer. "UNLK-LOCK" controls access to the layer, where "UNLK" allows editing and "LOCK" prohibits editing. "THW-FRZ" controls visibility and the ability to edit or change the layer, where "THW" unlocks and makes the layer visible and "FRZ" locks and makes the layer invisible. Changes may be made in two steps:

Mark the layer to be changed in the Dialog Box by selection with then cursor. The status changes will be executed for the marked layer.

Select the field required for the status change(s).

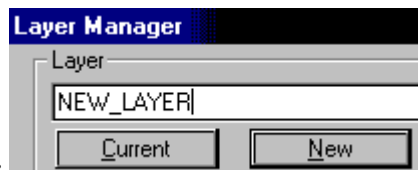
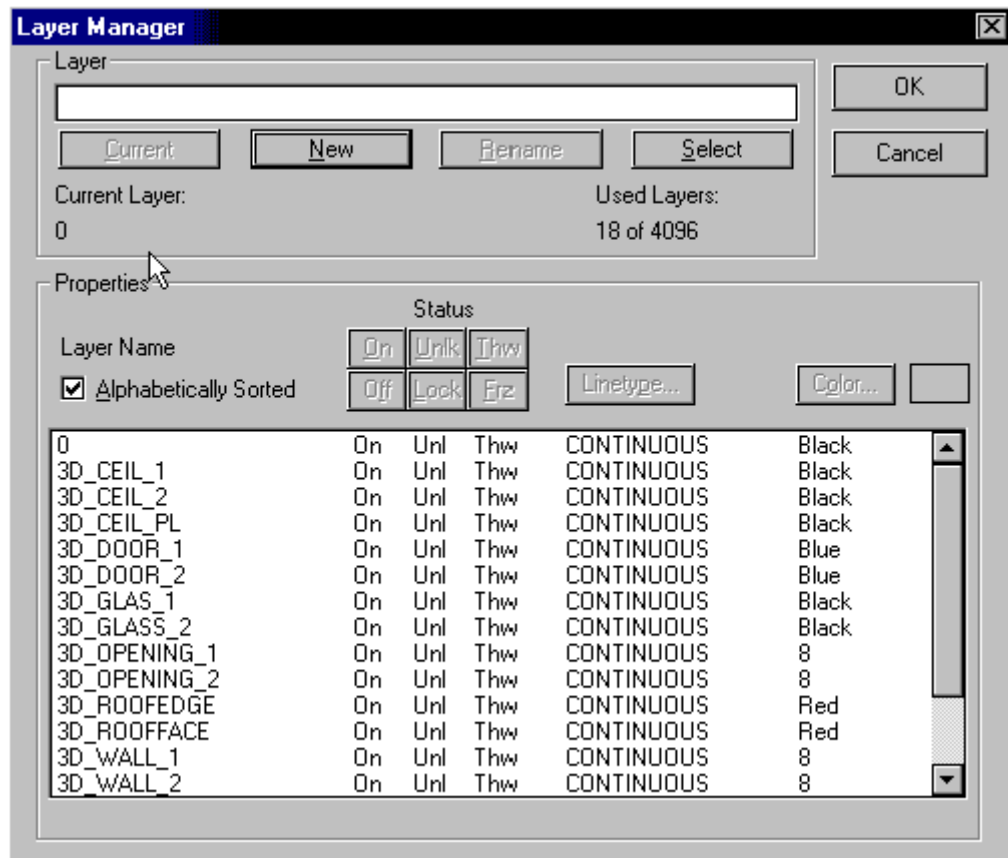
Remember that the current or active layer cannot be frozen.

### Creating a new Layer

From within Layer Manager, a new layer can be created at any time. Newly created layers do not have to have drawing elements right away.

The layers of a drawing are not listed alphabetically or by content, but according to their order of creation. To work effectively, it makes sense to plan the layers to be used in advance, and create them in a systematic manner.

If you use the same layer system in all of you drawings, you may wish to consider using a Template drawing, if you are not already doing so.



#### To create a new layer:

Enter the name for the new layer in the entry- and view field for layer adaptation (second row).

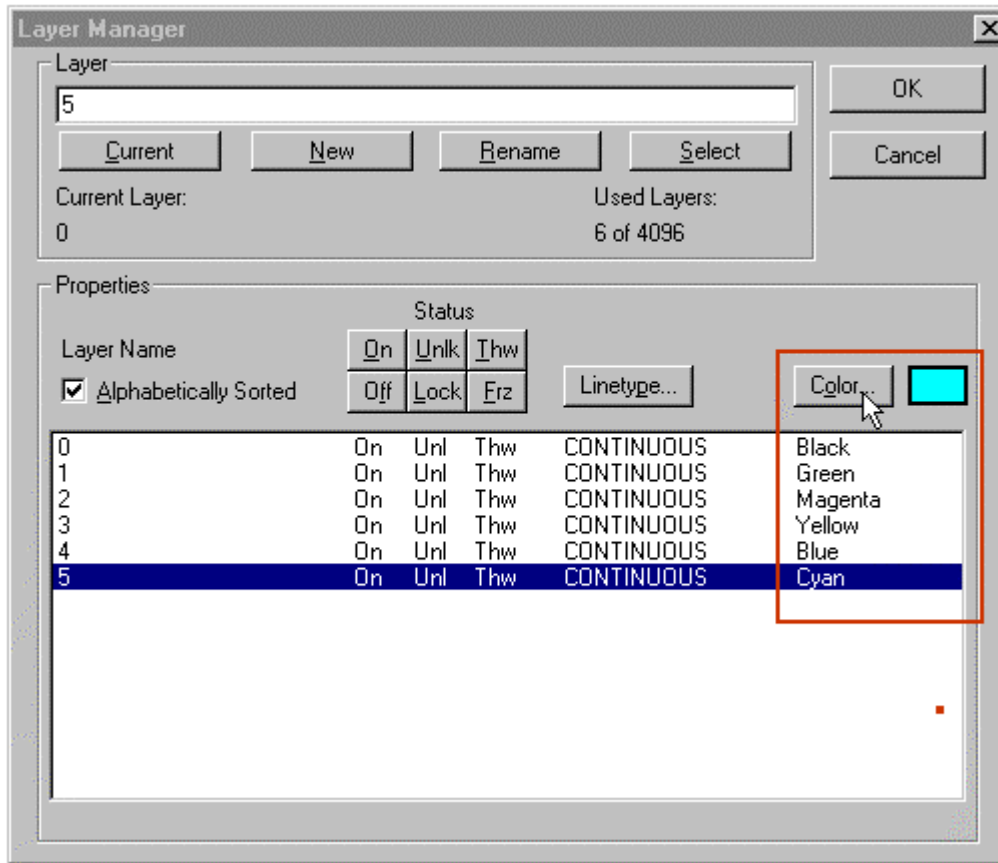
Remember the limits of naming.

After that, select the button *New*. The layer will appear following the layer last created.

Every new layer has a color 7 (black, if no other is chosen) and the linetype (continuous, if no other is chosen) assigned to. Other linetypes and colors are assigned through selection. More information regarding that point will follow.

#### Define Layer Color

When a new layer is created it will automatically be assigned the layer color 7 (black). A new color can be assigned to the new layer via the click-on field COLOR in Layer Manager. The section of the dialog that deals with color assignments is outlined in red for clarity.



Click on *Color...* The Colors dialog box will open;

Choose a new standard color from among the color boxes or enter a color number (1-255) into the edit box. The chosen color will appear in the top left preview area;

Verify your choice by clicking OK.

This makes the chosen color the "standard" color of this layer. All elements of this layer will be drawn with this color that is also called color BYLAYER.

When the standard color is changed, all elements of this layer that were drawn with the standard color will be shown with the new color now designated as the BYLAYER color .

The elements of the layer that were drawn with a color different than the standard color will not be affected by the change.

## Command TABSERVICE



### Command TABSERVICE

Alias TB

#### Located:

1. Pull-down Menu: Edit – Drawing Database Tables – Tables Maintenance
2. Top Function Bar
3. Palette Edit
4. Or Enter Command or Alias at the Command Line

The TABSERVICE command offers the user the opportunity to simply alter and maintain parameters used in the current drawings drawing database. The command is located on the Edit Pull-down menu as the "Drawing Database Tables – Tables Maintenance" selection. It can also be typed in at the command



line, if preferred.

When the command is issued, you will be presented with the dialog pictured below. Here you can select the parameter that you wish to modify:



If you wish to modify Layers, for example, you can eliminate all empty (containing no entities) or unused table entries, by clicking on the Purge Table button. You can also rename or delete items, such as layers, by clicking on the other two buttons.



## Command LAYCONF

**Command:** LAYCONF

**Alias** LC

### **Located:**

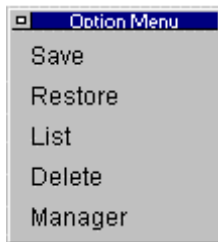
- 1. Pull-down Menu: Tools – Layer Configurations**
- 2. Or Enter Command or Alias at the Command Line**

The LAYCONF command is used to save and restore layer settings in and from a named status file.

As you work in the program, you will probably add new layers and change layer colors or other parameters using the Layer Manager. The LAYCONF command allows you to save layer settings at any time, in a named setting configuration file, and also lets you return to those saved settings at any later point by specifying the saved configuration file.

Note that the status of newer layers, that are not specified in the named configuration file are not changed in any way by a return to a previous layer status – only layer parameters for layers specified in the saved configuration are changed.

The command LAYCONF is typed in at the command line, at which time you see the option dialog:



You also see a corresponding prompt at the command line:

```
> LAYCONF
```

```
Select option: Save/Restore/List/Delete/Manager:
```

At this point, you can make your selection by picking the option on the dialog with the mouse, or by typing in the first letter of the option you want at the command line.

The **Save** option will save current layer status in the filename that you specify. The default is "LC2" for the first saved file, "LC3" for the second, and so on. You can accept these default filenames by pressing the Enter key, or else you can change the name to another and confirm your choice by pressing Enter.

The **Restore** option is used when you want to revert to a previously saved configuration. You will be prompted for the filename, after which the layer status is changed back. Remember that if you think you will want to return at any time to the status you have prior to the reversion, that you should save that status first in a different configuration file.

The **List** option simply returns a list of all stored configuration files. This can be useful if you have forgotten the filename that you specified earlier when you saved.

The **Delete** option is used to delete a configuration file that you no longer need.

The **Manager** option simply calls the Layer Manager, which allows you to change current layer settings and add new layers.

## Object Properties

These characteristics are said to be "properties" of an object:

- ▶ objects may be placed on a different layer;
- ▶ objects may have a different color; and
- ▶ objects may be drawn with a different linetype.

These characteristics are associated with a drawing automatically when it is created. This association is made through the parameters called BYLAYER that are determined for the layer when it is defined. Alternatively, the user may choose (in the manner just ascribed) a special color or linetype for an object.

## Setting Current Object Properties

To call on these functions, select the indicated entries on the Options pull-down menu.

### Pull-down Menu: Tools – Set Layer

**Set Current Layer - Command SETLAYER**

**Located:**

1. Entered Directly at the Command Line

**Set Object Color - Command SETCOLOR**

**Located:**

1. Pull-down Menu: Tools – Set Color - Set Explicit Color
2. Or Entered Directly at the Command Line

### **Set Linetype - Command SETLINETYPE**

#### **Located:**

1. **Pull-down Menu: Tools – Set Linetype - Set Explicit Linetype**
2. **Or Entered Directly at the Command Line**

## **Changing Object Properties**

It may be desirable in certain circumstances to change these parameters once into the drawing or layer. In order to do this, the proper command is PROPEDIT.

The command PROPEDIT allows the user to make these changes:

- ▶ objects may be placed on a different layer;
- ▶ objects may be given a different color;
- ▶ objects may be given a different linetype.

### **Modify Object Properties...**



**Command: PROPEDIT**

**Alias PED**

#### **Located:**

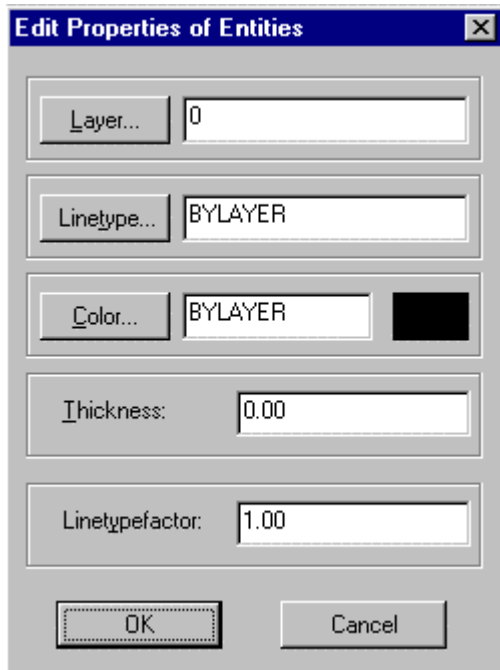
1. **Pull-down Menu: Edit – Modify Object Properties**
2. **Left Side Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

The function **Modify Properties...** invokes the dialog boxes that allow you to set the properties of entities, and make changes in **Color**, **Layer**, and **Linetype**.



As with the other commands in this set, first select one or more objects, using the selection dialog shown above, if required.

Then press RETURN, or use the right-hand cursor button.



The dialog box that appears displays the current properties Layer, Linetype, Color, Thickness and Linetype Factor (also called Linetype Scale Factor ) of the entities selected. You may then edit these properties.

#### Command TABSERVICE



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Alias TB

#### Located:

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4. Or Enter Command or Alias at the Command Line

The TABSERVICE command offers the user the opportunity to simply alter and maintain parameters used in the current drawings drawing database. The command is located on the Edit Pull-down menu as the "Drawing Database Tables – Tables Maintenance" selection. It can also be typed in at the command line, if preferred.

When the command is issued, you will be presented with the dialog pictured below. Here you can select the parameter that you wish to modify:



If you wish to modify Layers, for example, you can eliminate all empty (containing no entities) or unused table entries, by clicking on the Purge Table button. You can also rename or delete items, such as layers, by clicking on the other two buttons.



## Entering Commands and Options

### Issuing Commands and Options

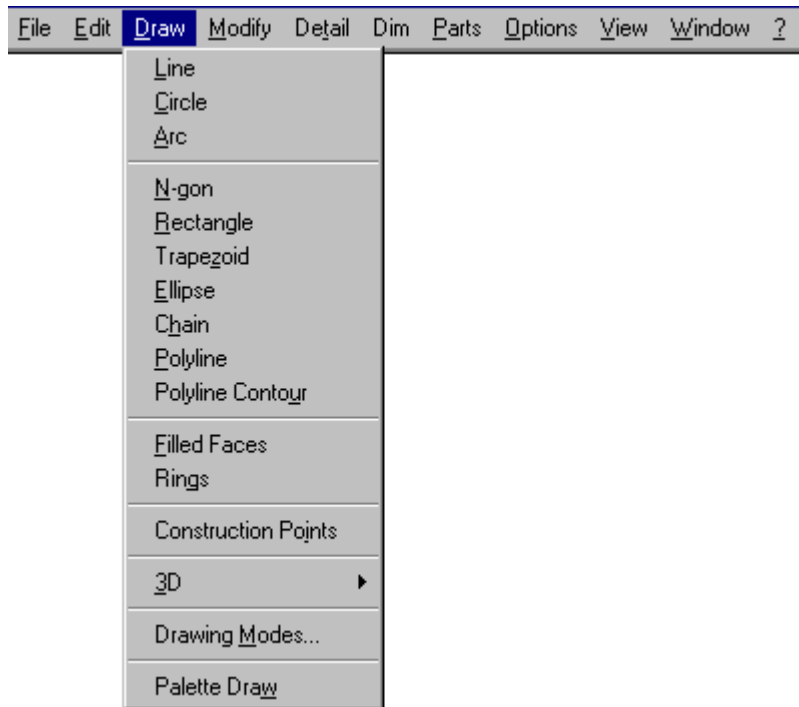
#### Command Entry

For most commands, AllenCad offers several methods of command entry. Commands may be initiated by selecting an option on a pull-down menu, selecting an icon on a toolbar or palette or directly entering the command name at the command prompt.

A series of commands can also be recorded in Macros (also called Script files) or programmed in LISP routines for automated execution.

See Also: Using Relative Coordinates

#### Pull-down Menus



Commands (and their options) can be selected using pull-down menus. The menus and commands may be requested by key sequence (for example, ALT + F, N for new file) or by clicking the proper option field within its menu.

For some commands, submenus are available. These submenus offer further commands or variations of the command.

### Horizontal Toolbar



When you install *AllenCad* out of the box, by default, below the menu bar there is a button lines divided into a left and a right area. This component of the desktop layout is called Toolbar.

Depending upon your selected preference for command options, there may also be another bar below, called the Options Bar.

#### System Buttons



A group of six buttons is located on the left side of the toolbar. These buttons cover commands from the *File* menu that refer mainly to the opening, saving and output of drawings, and to general program settings:



**Starting a New Drawing**



**Opening an Existing Drawing**



**Closing the Current Drawing**



**Saving the Current Drawing File**



**Printing the Current Drawing File**



**Configuration: General Settings for the Program**



Another group of symbols is arranged on the toolbar, to the right of the systems buttons. These icons represent commonly used drawing management commands.

Among the symbols you find, are those for customization of the program's environment and the arrangement of the drawing windows.



On the right side of the toolbar is a group of icons that call the palettes onscreen. These palettes are like toolbars in that they contain groups of associated command icons, but unlike the toolbar itself, these palettes are called onto and can be moved around on the drawing window, to be available where and when you need them. When finished with them, palettes can be closed until called again.

## Control Panel - Left Sidebar



The Control Panel – Left Sidebar is situated immediately left of the Drawing Edit Window and it contains a relatively permanent series of command or function buttons. Please refer to the Desktop and Menu Editor utility if you wish to re-organize or replace the icons on this panel. Please note also that this control panel can be switched off, using the available option in the DESKCONF command. This panel has been designed to provide the most commonly used drawing and editing commands, however, and the user should assure himself/herself that they have a suitable alternative command entry method in mind, before turning the panel off.

The symbols arranged there can be subdivided into groups as follows:

**Drawing tool symbols** - Explanations for the adjustment of these functions and their use in drawing may be found in the Help section *Users Guide – The Basics of Drawing – Precision Aids*

**Symbols to control the drawing views** - The explanation of these commands are given in *Users Guide – The Basics of Drawing – Viewing the Drawing*.

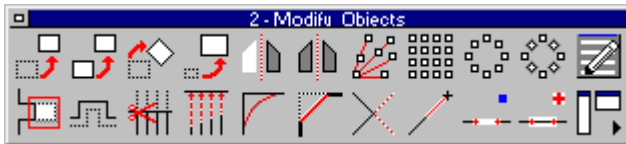
**Commands to control User Coordinate Systems and Layout Mode** - See the section *Users Guide – The Basics of Drawing – Coordinate Systems* for information on the usage of User Coordinate Systems and *Users Guide – Layout Mode / Paper Space* for information on the Layout Mode features.



**Detailing Functions: dimensioning, crosshatching, text** - Dimensioning is discussed in the section *Users Guide – Detailing the Drawing – Dimensioning Variable Summary*, hatching in *Users Guide – Detailing the Drawing – Crosshatching*, and text editing in *Users Guide – Detailing the Drawing – Text*.

**Inquiry Commands** - The explanation of these commands is given in *Users Guide – Obtaining Drawing Information*

## Palettes (Floating Toolbars)



A series of frequently used commands can be called using symbol fields. Symbols in palettes, arranged in any way, that can contain any command for that the palette is designed.

Commands and functions are called from a palette by clicking on the symbol.



Loading palettes

Palettes are files with the file extension <.mnp>. The command PALETTE is used to load a palette. Individual palettes can also be loaded by clicking on the palette icons on the toolbar or making the selections available on the pull-down menus.

See also:

PALCLOSE

PALOPEN

## Input by keyboard



With the availability of alias commands, one advantage of keyboard entry of command instructions is speed and efficiency. With practice and familiarity, users can develop drawings in a fraction of the time it would take other users to enter the same commands using toolbar icons or pull-down menus.

Input by keyboard has the additional advantage of precision. Using this method, the user can construct drawing entities with exact dimensions or with exact placement in the coordinate system.

Values for distances, straight lines and other linear quantities are specified in drawing units, angle values in degrees.

Coordinates are given, in sequence, as the values for the X-, Y- and Z-axes with a comma between.

Please note that a point is used only to designate decimal values. A comma serves as a separator between the x, y and z axis values for any given point.

The program also provides Coordinate Filters that can be used when point values are requested.

Coordinate Filters allow the user to relate the point value inputs to the X, Y and Z coordinates of existing entities.

When specifying an input value for an angle the sign < (smaller than) comes before the value.

An input sequence on the keyboard might look like this:

LINE

From point: 2.5,0

To point: 5.75,0

To point: 0,3.25

<ENTER>

Moving an object might look like this:

Select objects: Selecting

Basis point: 5,2  
Target point: @3,0

## Command Aliases, Accelerator Keys and Scripts

AllenCad is compatible with most of AutoCAD's shortcuts, such as command aliases and accelerator keys. These aliases and key assignments are simply modified to meet your needs through the CONFIG command. A full listing of all pre-set command aliases is included on the Command Reference Chart.

### Alias Command Chart

<b>AllenCad Command</b>	<b>Alias</b>	<b>Command Function</b>
3DBOX	3B	Creates a three-dimensional solid box
3DCONE	3DC	Creates a three-dimensional solid cone
(APPLICATION)		
3DELLIPS	3L	Creates a 3d ellipsoid
3DFACE	3F	Creates a three-dimensional face
3DMESH	3M	Creates a three-dimensional polygon mesh
3DMIR	3MR	Mirrors a 3d Object
3DMIR (Application)	M3	Creates a mirror image of objects about a plane
3DOBJECTS	3O	Creates three-dimensional polygon mesh objects
(Application)		
3DROT	3R	Rotates a 3D Object
3DROT (Application)	R3	Moves objects about a three-dimensional axis
3DSPHER (Application)	3DS	Creates three-dimensional solid sphere
3DSPHERE	3S	Creates a 3D sphere
3DTORUS	3T	Creates a donut-shaped solid
3DVIEW	3V	Sets the three-dimensional viewing direction
3DWEDGE (Application)	3W	Creates 3D solid with a sloped face tapering along the X axis
'ABOUT	AB	Displays information about the Program
'APPLOAD	AP	Loads and unloads applications and defines which applications to load at startup
ARC	A	Creates an arc
AREA	AR	Calculates the area and perimeter of objects or of defined areas
ARENDER	AE	Animated rendering
ARRAY	RA	Creates multiple copies of objects in a pattern
ATTDEDIT	ATD	Modify existing attribute definition
ATTDEF	AD	Creates an attribute definition
ATTEDIT	ATT	Modify part attributes, one by one
ATTEXP	ATX	Extracts attribute data
'ATTMOVE (Application)	ATM	Move attribute text
ATTROT (Application)	ATR	Rotate attribute text
ATTVALUE	AV	Edit attribute text value
ATTXEDIT	AX	Edits the variable attributes of a block
BATPROC (Application)	BT	Patch processing - Plots and conversion
BHATCH	BH	Fills an enclosed area with a hatch pattern
BMAKE	BK	Defines a block using a dialog box
BMPEDIT	BPE	Calls external BMP into Windows Paintbrush program for editing
BMPOUT	BO	Saves selected objects to a file in device-independent bitmap format
BOUNDARY	BD	Creates a boundary polyline (command line version of BPOLY)
BPOLY	BP	Creates a region or a polyline from an enclosed area
BREAK	B	Erases parts of objects or splits an object in two

CHAIN	CN	Draws single entity continuous lines
CHAMFER	CH	Bevels the edges of objects
CHANGE	CV	Changes the properties of existing objects
CHPROP	CHP	Changes the color, layer, linetype, linetype scale factor, and thickness of an object
CIRCLE	CI	Creates a circle
CLOSE	CL	Exit the drawing but not close the program
CONFIG	CFG	Configure program operating parameters
COPY	C	Duplicates objects
COPYPROP	CP	Change object properties to that of selected entity
COPYSELECT	CST	Copies objects to the Clipboard
CUTSELECT	CT	Copies objects to the Clipboard and erases the objects from the drawing
DELETE	DEL	Removes objects from a drawing
DELFLK	DLK	Deletes AllenCad lock files (*.flk)
DELPARTIAL	DPL	Delete part of an entity
DESKCONF	DSK	Configures the desktop options
DGNIN	DGI	Imports a Microstation .dgn drawing file
DGNOUT	DGO	Exports a Microstation .dgn drawing file
DIGICONFIG	DC	Digitizer configuration
DIGIOFF	DGF	Switch digitizer mode off
DIGION	DGN	Switch digitizer mode on
DIMA3P	D3P	Creates an angular dimension
DIMA4P	D4P	Creates an angular dimension out of four definition points
DIMALI	DAL	Creates an aligned linear dimension
DIMBAS	DBA	Continues a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension
DIMCENT	CM	Creates the center mark or the center lines of circles and arcs
DIMCON	DCN	Continues a linear, angular, or ordinate dimension from the second extension line of the previous or a selected dimension
DIMDIA	DDA	Creates diameter dimensions for circles and arcs
DIMDRAW (Application)	DDR	Creates and modifies dimension styles
DIMEDIT (Application)	DED	Edits dimensions
DIMHOR	DH	Horizontal dimensioning
DIMLIN	DL	Creates linear dimensions
DIMOBL	DOB	Modifies the orientation of existing linear dimensioning objects.
DIMORD	DOD	Creates ordinate point dimensions
DIMRAD	DRA	Creates radial dimensions for circles and arcs
DIMREST	DRE	Reinstate a stored style file so that it can be reinstated as the current style
DIMROT	DRO	Rotate existing dimensions
DIMSAVE	DS	Allows the user to combine the individual parameters and store them in a named dimensioning style
DIMSER	DSR	Adds multiple additional dimension labels in the same orientation
DIMTHOME	DTH	Undoes the movement of dimension text
DIMTMOVE	DTM	Moves dimension text
DIMTNEW	DN	Modify Dimension text value
DIMTROT	DTR	Rotate dimension text
DIMTYPE	DDR	Creates and modifies dimension styles
DIMTYPE	DTP	Accesses dimensioning dialog
DIMUPD	DU	Updates dimension
DIMVER	DVR	Labels dimensions along the Y (vertical dimension) axis
DINSERT	DIN	Insert a part without registering it with the drawing
'DIST	D	Measures the distance and angle between two points
DIVIDE (Application)	DI	Places evenly spaced point objects or blocks along the length or

DLGEDIT	DM	perimeter of an object
DOFFSET	DOF	Desktop and Menu editor
DOFFSET	O	Creates concentric circles, parallel lines, and parallel curves
DPAN	DP	Distance offsetting of entities
DRAWMODE	DR	Pan dynamically
DRAWORDER	DO	Specify drawing modes for subsequent entities
DTEXT	DT	Changes the display order of images and other objects
DVIEW	DV	Displays text on screen as it is entered
DXFIN	DXN	Defines parallel projection or perspective views
DXFOUT	DXO	Imports a drawing interchange file
DZOOM	DZ	Creates a drawing interchange file of the current drawing
EDGE (Application)	EG	Dynamic zooming
EDITPAR	EP	Changes the visibility of three-dimensional face edges
EINFO (Application)	LI	Dialog to change edit parameters
'ELEVATION	EL	Displays database information for selected objects
ELLIPSE	ELL	Sets elevation properties of new objects
ERASE	E	Creates an ellipse or an elliptical arc
EXIT	X	Removes objects from a drawing
EXPAND	EXP	Exits AllenCad
EXPLODE	EX	Extends an object to meet another object
EXTEND,	ET	Breaks a compound object into its component objects
FCOMPILE	FC	Extends an object to meet another object
FHIDE	FHD	Compile a shape file to a binary font file
FILLET	F	Creates a second drawing file with a hidden line view
FLIP	FL	Rounds and fillets the edges of objects
FLXRECOVER	REC	Mirror and delete original entity
FONT	FT	Repairs a damaged drawing
		Creates or modifies named styles and sets the current style for text in your drawing
GROUP	G	Creates a named selection set of objects
HATCH	H	Fills a specified boundary with a pattern
HELP (F1)	?	Displays online help
HIDE	HD	Creates a second drawing file with a hidden line view
HPEDIT	HED	Modifies an existing hatch object
'ID	ID	Displays the coordinate values of a location
INSERT	I	Inserts a part or another drawing
INTERSECT	IN	LIGHTEDIT Creates composite solids or regions from the intersection of two or more solids or regions
LAYER	LA	Manages layers and layer properties
LAYCONF	LC	Save and restore layer settings
LAYREF	LRF	Change object layer properties to match reference object
LENGTHEN	LEN	Lengthens an object
LIBRARY	LB	Manages collections of symbols and parts
LIGHT	LIT	Manages lights and lighting effects
LIGHTEDIT	LE	Modify light source settings
'LIMITS (Application)	LM	Sets and controls the drawing boundaries
LINE	L	Creates straight line segments
LINETYPE,	LT	Creates, loads, and sets linetypes
LISTFILE	LF	View contents of an external file
LOADLISP	LL	Loads a LISP file for use
LOADLTYPE	LLT	Loads linetype definitions from the command line
LOGFILEOFF	LOF	Closes the log file opened by LOGFILEON
LOGFILEON	LON	Writes the text window contents to a file
'MACRO	MA	Executes a sequence of commands from a script
MEASURE (Application)	MR	Places point objects or blocks at measured intervals on an object
MENU	MU	Loads a menu file

MERGE	MG	Import an external .flx drawing file
MIRROR	MI	Creates a mirror image copy of objects
MOVE	M	Displaces objects a specified distance in a specified direction
MTEXT	MT	Creates multi-line text
MTEXTEDIT	MTE	Edit multiline text / Import text files
NEW	N	Creates a new drawing file
NGON	NG	Creates an equilateral closed polyline
OFFSET	O	Offset linear objects in parallel orientation
OPEN	OP	Opens an existing drawing file
OSNAP	OS	Calls for manual object snap assignment
PALCLOSE	PLC	Removes a palette from the desktop
PALETTE	PLT	Places a palette on the desktop
PALETTE, PALMAN (Application)	PA,PM	Displays, hides, and customizes toolbars
PALMAN	PM	Opens the palette manager
PALOPEN	PLO	Opens a new palette
'PAN	P	Moves the drawing display in the current viewport
'PANDOWN	PD	Pans down
'PANLEFT	PL	Pans left
'PANRIGHT	PR	Pans right
'PANUP	PU	Pans up
PARTDEF	PDF	Define a part (block)
PARTEXP	PX	Writes objects to a new drawing file
PARTLIB	LIB	Calls the part library manager
PASTE	PST	Inserts data from the Clipboard
PCONTOUR	PC	Creates a boundary polyline on top of other entities
PFACE	PF	Creates a three-dimensional polyface mesh vertex by vertex
PLAN	PLN	Displays the plan view of a user coordinate system
PLANVIEW	PV	Returns a 3d view to being planar to current UCS
PLBSETUP	PLB	Sets up or modifies existing part (block) libraries
POINT	POI	Creates a point object
POLY3D	3P	Creates a polyline of straight line segments in three-dimensional space
POLYEDIT,	PE	Edits polylines and three-dimensional polygon meshes (VTXEDIT edits polyline vertices)
POLYGON,	PG	Creates an equilateral closed polyline
POLYLINE	PO	Creates two-dimensional polylines
PRECPAR,	PCP	Customizes the AllenCad settings
PRINT	PT	Plots a drawing to a plotter, printer, or file
PROPEdit	PED	Changes the properties of existing objects
PROPLAYER	PP	Moves objects to selected layer
PSETUP	PS	Sets up print devices
PULLDOWN	PN	Loads partial menu files
PURGE	PRG	Removes unused named objects, such as blocks or layers, from the database
QARRAY	QR	Command line version of ARRAY command
QATTEDIT	QAE	Changes attribute information independent of its block definition (Command line version)
QBMPOUT	QB	Command line version of BMPOUT command
QFONT	QF	Command line version of FONT command
QHATCH	QH	Command line version of HATCH command
QHPEDIT	QHE	Command line version of HPEDIT command
QINSERT	QI	Command line version of INSERT command
QLAYER	QL	Command line version of LAYER command
QMTEXT	QM	Command line version of MTEXT
QOPEN	QO	Command line version of OPEN command

QPRINT	QP	Command line version of PRINT command
QPROPEDIT	QPR	Command line version of PROPEDIT command
QSAVE	QS	Saves the current drawing
QSLDOUT	QSL	Command line version of SLDOUT command
QTEXT	QT	Controls the display and plotting of text and attribute objects
QVIEW	QV	Command line version of VIEW command
QVIEWPORT	QVP	Command line version of VIEWPORT command
QWMFOUT	QWO	Command line version of WMFOUT command
QWOPEN	QW	Command line version of WOPEN command
QXLINK	QX	Controls external references to drawing files (Command line version)
RCONFIG	RC	Sets rendering preferences
RECTANGLE, RECTANG	RT	Draws a rectangular polyline
REDEFINE (Application)	RE	Restores AllenCad internal commands overridden by UNDEFINE
REDO	RD	Reverses the effects of the previous UNDO or U command
REDRAW	R	Refreshes the display of the current viewport
REGEN	RG	Regenerates the drawing and refreshes the current viewport
REGENALL	RGA	Regenerates the drawing and refreshes all viewports
REJOIN	RJ	Creates single entity from co-linear objects
RENAME	RNM	Changes the names of objects
RENDER	REN	Creates a realistically shaded image of a three-dimensional wireframe or solid model
REVSURF	RS	Creates a rotated surface about a selected axis
RING	RI	Draws filled circles and rings
ROTATE	RO	Moves objects about a base point
RTEXT	RX	Writes new text with same properties as a selected text entity
RULESURF	RSF	Creates a ruled surface between two curves
SAVE	S	Saves the drawing under the current file name or a specified name
SAVEALL	SAL	Saves all open drawings
SAVEAS	SA	Saves an unnamed drawing with a file name or renames the current drawing
SCALE	SC	Enlarges or reduces selected objects equally in the X, Y, and Z directions
SELECT	SL	Places selected objects in the Previous selection set
SENDALL	SDA	Sends all drawings via Microsoft Exchange
SENDFLX	SD	Sends the current drawing via Microsoft Exchange
'SETCOLOR	STC	Sets the color for new objects
'SETLAYER	STL	Change current layer to selection
SETLIB	SLB	Displays current part library dialog
SETLINETYPE	SLT	Creates, loads, and sets linetypes
'SETTINGS	ST	Customizes the AllenCad settings
SETUP or UNITSETUP	US	Sets up drawing units
'SETVAR	SV	Lists or changes the values of system variables
SETVIEWDIR, 3DVIEW	STV	Sets the viewing direction for a three-dimensional visualization of the drawing
SLDOUT	SO	Creates a slide file of the current viewport
SPLINE (Application)	SP	Creates quadratic or cubic spline (NURBS) curve
SPLINE;D	SPD	Edits a pline object
STRETCH	SR	Moves or stretches objects
TABCONFIG	TC	Configure tablet menu areas
TABLES	TBL	Documents and controls database information and drawing parameters
TABLET	TA	Calibrates, configures, and turns on and off an attached digitizing

TABSECTION	SEC	tablet
TABSERVICE	TB	Replaces a section of the tablet menu
TABSURF	TF	Changes the names of named objects
		Creates a tabulated surface from a path curve and a direction vector
TCORRECT	TR	Corrects text entities and attribute definitions
TEMPLATE	TP	Defines drawing file to be used as example in initial drawing setup
TEXT	TX	Creates a single line of text
TEXTEDIT	ED	Edits text and attribute definitions
TEXTEDIT	TE	Modifies a line of text
'TGRID	TG	Toggles grid display on/off
'THICKNESS	TH	Sets extrusion thickness properties of new objects
TICON,	TI	Controls the visibility and placement of the UCS icon
'TLMODE	TM	Switches from paper space to a model space viewport
'TORTHO	TO	Constrains cursor movement
TRAPEZOID	TZ	Creates a 3D trapezoid
TRIM	T	Trims objects at a cutting edge defined by other objects
'TSCREEN	TN	Opens the text window
'TSCREENFULL	TSC	Toggles command line window on/off (full screen)
'TSNAP	TS	Restricts cursor movement to specified intervals. Toggles snap modes on/off
U	U	Reverses the most recent operation
UCS	UC	Manages coordinate systems
UCSFOLLOW	UF	
UCSICON	UI	Controls the visibility and placement of the UCS icon
UNDEFINE (Application)	UF	Allows an application-defined command to override an internal AllenCad command
UNDELETE	UD	Restores erased objects
UNDO	U	Reverses the effect of commands
'UNITSETUP	US	Sets coordinate and angle display formats and precision
VIEW	V	Saves and restores named views
VIEWPAR	VR	Selects view parameters
VIEWPORT	VP	Creates floating viewports and turns on existing floating viewports
VIEWPORT and option	VPL	Sets layer visibility within viewports
LAYER		
VSLIDE	VS	Displays a raster image slide file in the current viewport
VTXEDIT	VX	Edits polylines and three-dimensional polygon meshes (VTXEDIT edits polyline vertices)
WBLOCK,	WB	Writes objects to a new drawing file
WCASCADE	WD	Arrange windows in cascade format
WCLOSE	WC	Closes the current viewport
WICONARR	WI	Arranges all drawing window icons
WMFOUT	WM	Saves objects to a Windows metafile
WOPEN	WO	Opens new viewport
WTILE	WT	Arranges Windows in a tiled format
WTILEHOR	WH	Arranges Windows in a horizontally tiled format
WTILEVER	WV	Arranges Windows in a vertically tiled format
XINSERT	XI	Attaches an external reference (Xref)
XLINK	XL	Attaches an external reference to the current drawing
XPLODE (Application)	XPL	Breaks a compound object into its component objects
XPLODER	XP	Accesses and allows alteration of the current drawings properties
XREF	XR	Controls external references to drawing files (Command line version)
'ZOOM	Z	Increase or decreases the apparent size of objects in the current

		viewport
'ZOOMBACK	ZX	Zoom to previous view
'ZOOMFAC	ZF	Zoom by a factor
'ZOOMFIT	ZE	Zoom to extents
'ZOOMIN	ZN	Zoom into the drawing
'ZOOMOUT	ZT	Zoom out from the drawing
ZOOMPAGE	ZP	Zoom to drawing page limits
'ZOOMWIN	ZW	Zoom to a window

## Transparent Alias Commands

While a command is active, transparent alias commands can be issued by immediately prefacing the alias with an apostrophe, in a similar fashion to non-transparent alias commands.

See the example of the transparent alias command 'Z below:

Definition in the PCAD5.key file:

Z=ZOOM

> LINE

From point:

To point:

To point: 'z

Zoom scale factor: Window

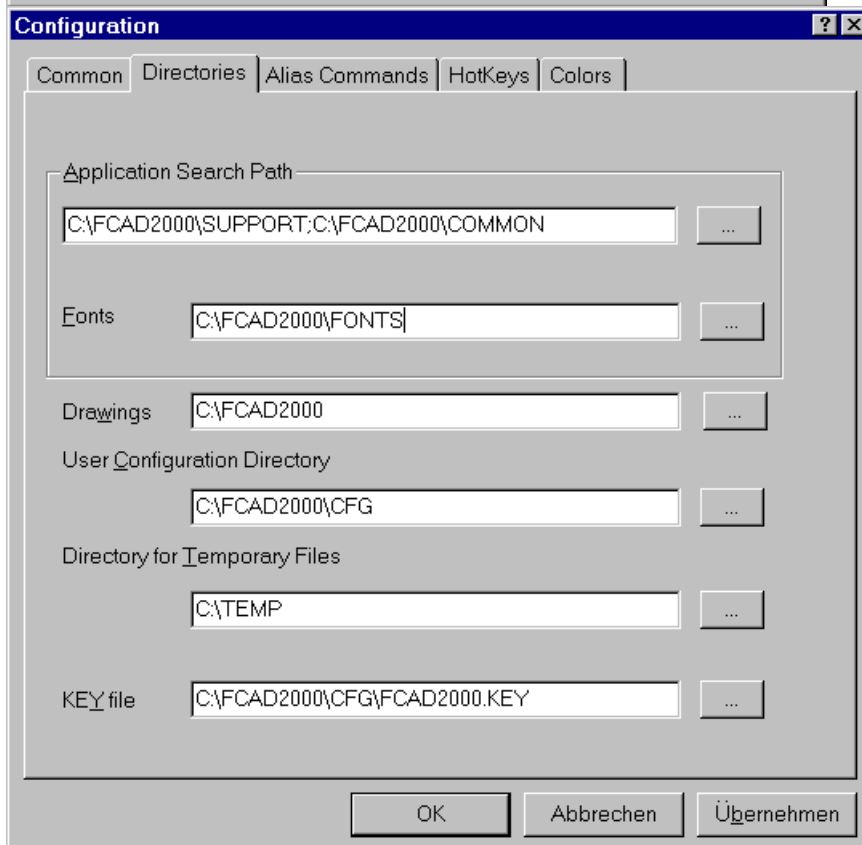
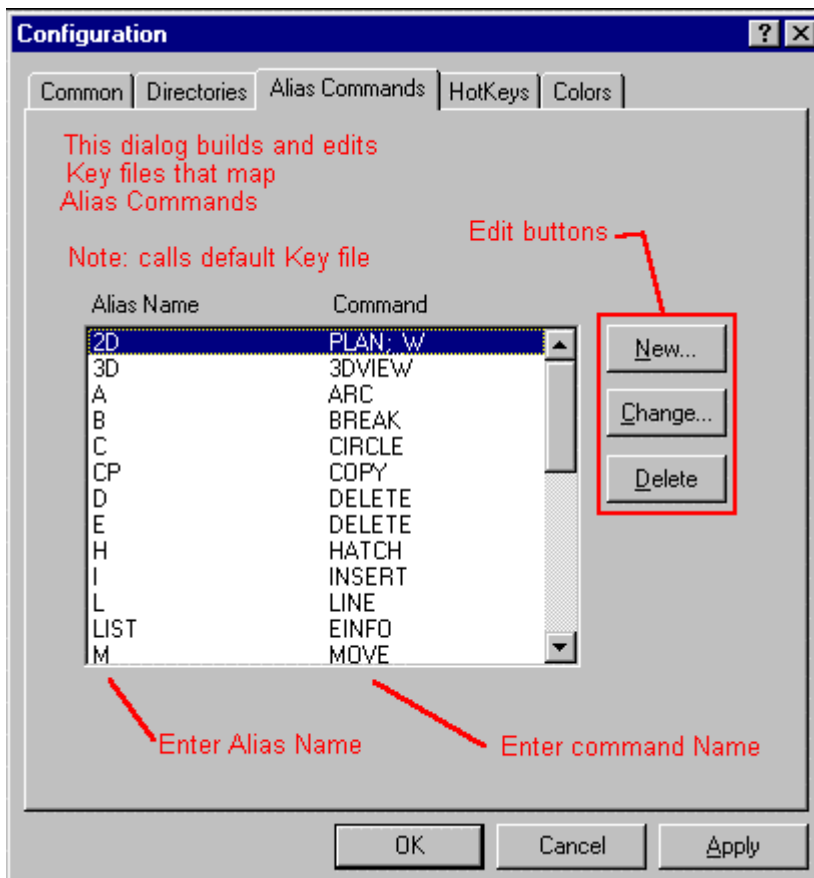
First corner:

Second corner:

To point:

Note: Alias Commands are defined under the Alias Commands tab on the CONFIG dialog.





## Repeating Commands

During practical experience with the program the user will often need to repeat a command. To repeat the same command either by pull-down menu or keyboard entry would be annoying and inefficient. The user can repeat the last executed command by pressing the ENTER key or the right mouse button. The command repeat can be performed as often as is required. Only the standard variant of the corresponding command will be repeated. Other options that were selected will not be used automatically. However, these options are presented and are at the disposal of the user, and may be selected again.

For example when drawing a line:

> LINE

From point: *P1*

To point: *P2*

To point: *P3*

To point: <ENTER>

<ENTER> (this second ENTER or right mouse click reactivates the previous LINE command)

> LINE

From point: *P4*

To point: *P5*

To point: *P6*

To point: *Close*

## Interrupting the Execution of a Command

It is possible to interrupt the performance of a command before the command has been completed. This can be useful if a command was activated by accident.

To interrupt the execution of the current command:

Press ESC; or

Click the Cancel button



That will interrupt the execution of the current command, and in the Text Window, the user will see the message \*\*\* Cancel \*\*\*.

## Withdrawing a Command (UNDO)

If the user recognizes that he or she has input an erroneous command, it is possible to withdraw the command. This command can also be issued repetitively and can be used to undo as many as the previous twenty commands.

As it is running, the program stores the command history in a list called the Undo List. That list contains a registry of the previous twenty commands and their options. These can be withdrawn in reverse input order by using the UNDO function. You simply

Back the program up to the point where you want to continue issuing commands again. The command can be repeated until all working steps stored in the UNDO LIST have been canceled. The program will then send out the message "UNDO LIST empty

Note however, that certain commands are not registered in the UNDO LIST. These are typically viewing commands and include Zoom, Pan, Window, etc., as well as other transparent functions. Issuing the UNDO command after these commands will result in the command previous to the view-altering command being undone.

The Undo List is kept only with the current session of the program. It will not be saved with the drawing. Calling the Command

► Click the Undo symbol in the toolbar or palette;



, or

- Enter U by keyboard and confirm this input by <ENTER>.


See also:

Restoring (REDO)

### Restoring (REDO)

Just as UNDO backs up and erases previously issued commands, REDO reverses the erase operation, sequentially replacing commands that have been "undone". This only works however, if no other command executed between the commands UNDO and REDO.

REDO is used in the following fashion:

- Call the command REDO by clicking the symbol , or
- Enter the command name REDO at the command line.

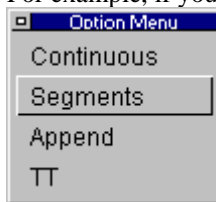
Only the last relevant working step will be restored with each issued REDO command

Another form of restoration is to use the command UNDELETE. This command cancels the last relevant UNDO operation completely. Note that it is possible to UNDELETE even if there have been other drawing or working operations executed between the UNDO and the UNDELETE. Undelete simply reverses the last deletion command.

### Options Menu (Options Menu) / Options Bar

Commands that are performed at command line often provide several options. These context sensitive options determine how the command processes further. Please notice that in many cases these options also appear in the command line.

For example, if you have started the command LINE, the following Options Menu appears:



Command options appear in an Options Menu

#### Note:

After installation out of the box, the Options Menu is initially displayed in the upper right corner of the screen. You can move the Options Menu by dragging it. From now on, the Options Menu will pop up where you last placed it.

#### Hint:

As an alternative to the Options Menu, you can configure to utilize an Options Bar instead.

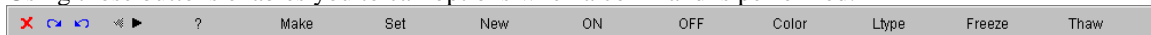
## The Options Bar

As an alternative to the Options Menu, you can configure to utilize an Options Bar, or Options Bar, instead. This bar will not be visible unless this option is selected.

The Options Bar is a special area of the desktop located underneath the command line. (where command line is located at the top of the screen).

It contains context-sensitive option buttons.

Using those buttons enables you to call options when a command is performed.



The Options Bar is arranged as follows:

A group of three initial buttons on the Options Bar pertain to the execution of commands.

These are (from the left):

1. ► Cancel the execution of the current command (Escape)
2. ► Nullify the withdrawing of the last work step (Redo)
3. ► Withdraw the last work step (Undo)

Two arrows in the Options Bar serve to scroll the available options.

A number of command prompts come up with such a number of choices that cannot be displayed all over the available space of the Options Bar. In such situation, one (or both) of the arrows will be available.

After a command has been invoked, the options currently available are displayed in the option bar. Clicking on the desired option will select that option. Hotkeys may also be used, if they are assigned to the options.

The option fields will be changed to correspond with the current status of the drawing. Doing this keeps the number of options shown manageable. If there are no options to the command, these option fields are empty. The option field may change as often as after each step in the drawing.

### **FLISP (FLEXIBLE LISP FORMAT)...**

AllenCad Pro provides the Flexible LISP Interpreter called "FLISP". FLISP is a program language variant of common LISP that provides the ability to develop powerful routines that extend the capabilities of your CAD program. With programmed FLISP routines, the ability to issue long sequences of commands automatically with a single run command becomes available. In this way, repetitive command sequences that will be used over, and over again can be saved for use whenever needed.

Unlike Macros, which simply save command steps in a file, FLISP program commands, in the hands of an experienced LISP programmer, have the ability to develop entirely new program commands. Many of AllenCad Pro's extended commands are in fact, FLISP programs.

### **Macros**

#### **Command: MACRO**

#### **Alias MA**

#### **Located:**

##### **1. Entered Directly at the Command Line**

Macros (sometimes referred to as script files) are designed to automate the execution of a series of specific instructions or program commands one step at a time. This eliminates the need to enter the commands or entries each time they are required in the program.

#### **Creating Macros**

Macros can be written using any text editor or word processing program that is able to store pure (ASCII) text format i.e. without the addition of any formatting information.

Enter the desired commands and instructions and their corresponding input values into this file in the same order in which the instructions are to be executed. A description of the individual commands and their necessary inputs can be found in the corresponding chapters of the *User's Guide*.

Command execution takes place line-by-line or instruction-by-instruction when separated by a blank space. Either an end of line or a space character is interpreted as confirmation of the previous entry.

This means that either of the following macros could be used to draw a circle with a mid point at 0,0 and a radius of 10.

**Sample**

#### **Using the Clipboard**

A simple and effective method of creating a macro file is made available by using the text window History Function and the Windows clipboard utility.

The text window contains the complete list of program commands and data entries that have been made during the course of a work sitting. This enables the user to reference and reconstruct every procedure

carried out so far. By either using the scroll bar located on the right hand side of the text window, or alternatively pressing the upward / downward directional keys, the user can "scroll" through the listed entries. The Windows clipboard utility can then be used to copy entries into an editor program where they can be stored as a macro file.

Use the following procedure:

1. Carry out the command steps that you wish to store.
2. Open the text window and use the mouse cursor to mark the command sequence.
3. Copy the highlighted section onto the clipboard using the CTRL + C key combination.
4. Call up your editor program, for example Windows Notepad.
5. Insert the contents of the clipboard into the editor using CTRL + V
6. Store the file in the support directory using any file name followed by the **.mcr** file extension identifier.

### Utilizing Macros (Command MACRO)

The macro utility can be called by entering the MACRO command. The program will then request the file name. It utilizes the pre-defined **.mcr** file if not specified otherwise.

Further information on the use of Macros can be found in Chapter 3 of the **Customization and Programmer's Guide**.

## Data Input

During the drawing process, it is often necessary to modify the value of data that has been input during the various steps of design. This occurs when the program needs input values for solutions. In those cases, a prompt will appear in the Text Window asking for the entry of a value.

Examples of these required values are:

Coordinates for defining points of the drawing (starting and ending points, center points, points of contact, inserting points, base and target points);

Distances;

Angles;

Width values (for example, line width); or

Number.

Number

Input by keyboard

Pointing or Identifying

Coordinates

The different coordinate types can always be entered as absolute or as relative coordinate values.

► Absolute Coordinates and

► Relative Coordinates.

## Menus

### Menu Bar and Pull-down Menus



The separate pull-down menus are arranged in the menu bar. A description of each pull-down menu and

the commands that they contain, can be viewed by selecting each menu heading listed below in the left window (Help Contents). Each pull-down menu contains groups of thematically related commands.

Menu items can have these states:

Dimmed, or grayed-out commands are not available in the particular working situation and can therefore not be selected. The command Save for example, cannot be executed if no drawings have been created or opened.

Three points (...) behind the command - for instance, Open... - show that no further requests will be displayed at the command line and all command parameters are set in a dialog box.

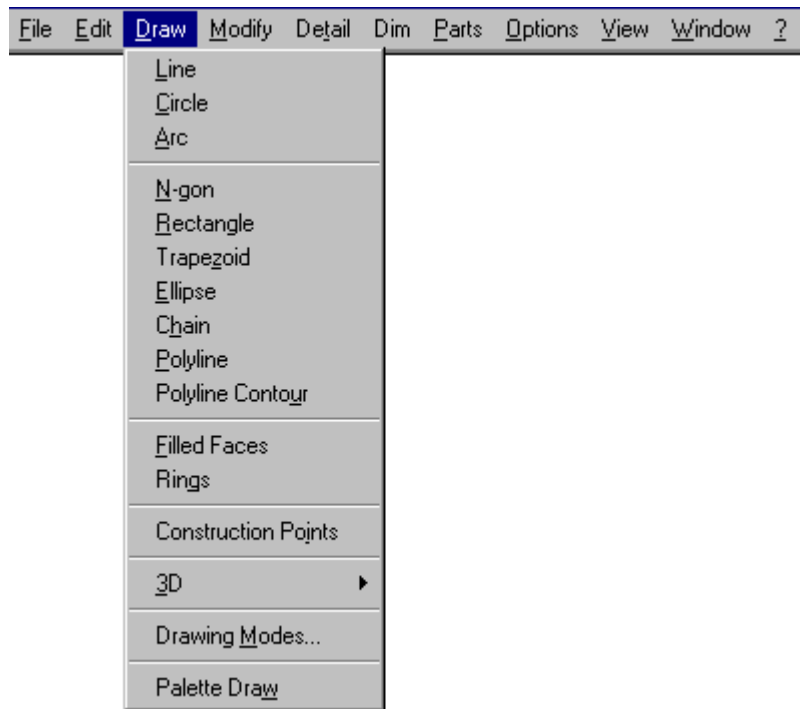
An arrow (triangle) behind a command indicates that a cascading menu will be opened and a variant of this command must be selected.

The content of the Menu Bar varies depending on the loaded menu file. Different menu files may be loaded, adapting the menu bar to the specific necessities of the user. Menu files can be identified by the extension *.mnu*. The program contains a standard menu file located in the sub-directory *APPLIC*.

The program offers you the opportunity to load different menu files. These files can be individualized menu files that have been created and adapted by help of the Desktop and Menu Editor (see explanation in the *Programmer's Guide*), or any of a number of standard menu files supplied with the program.

The menu file can be changed by using the Desktop/Tool Manager from the *File* menu or by typing the command *MENU* into the command line or the text window. For further information see the paragraph Desktop and Tool Manager in Chapter 11 of this document.

## Pull-down Menus



Commands (and their options) can be selected using pull-down menus. The menus and commands may be requested by key sequence (for example, ALT + F, N for new file) or by clicking the proper option field within its menu.

For some commands, submenus are available. These submenus offer further commands or variations of the command.

## Desktop and Menu Editor

### Desktop and Menu Editor – Command DLGEDIT

## Alias DM

### Located:

1. **Pull-down Menu: Tools – Resource Manager – Run Desktop and Menu Editor**
2. **Or Entered Directly at the Command Line**

The Desktop and Menu Editor (DME) design and lay out your menus and palettes in a visual environment. Knowledge of programming is not required.

Please refer to the Command Button Chart for input information when modifying palettes.

For more information on the DME click here: DME Help 

## Menu Files

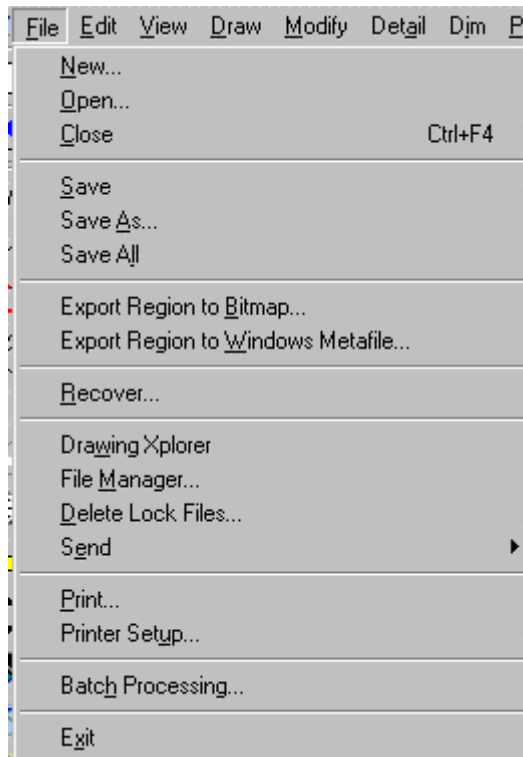
Menu files are saved using the .mnu file extension. They can be called into the program for use, using the MENU command. Once loaded, the menu file replaces the previous default. It is then identified as the new default menu in the PDMENUNAME variable. This new menu will now load automatically each time you start AllenCad. To create or alter pull-down menus, please refer to The Desktop and Menu Editor section in the documentation.

### Notes:

1. For those users who may be more familiar with, and prefer, AllenCad comes with .mnu files that emulate AutoCAD version 14 and 2000 pull-down menus (*R14\_emu.mnu* and *R2000\_emu.mnu*). These may be loaded for use in preference to the default *AllenCad5.mnu* file.
2. References to pull-down menu layout and selections in this and other AllenCad documentation refer to the default *AllenCad5.mnu* file, except where noted otherwise.
3. Further information on creating and editing menus can be found in Chapter 8 of the Customization and Programmer's Guide

## File

### "File" Pull-down Menu



The File pull-down menu contains command instructions that deal with drawing file configuration and external operations such as file saving, printing etc. For a description of each command, please refer to the specific topics.

## Open an Existing Drawing...



**Command: OPEN**

**Alias OP**

### Located:

1. **Pull-down Menu: File - Open**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

In order to continue working on an existing drawing in a later working session, the user must open the drawing. The term "Load" a drawing will also refer to "opening" a drawing.

The command **Open...** in the menu *File*, is used to do this. This command can also be activated by keyboarding OPEN or by clicking the standard symbol *Open* in the Function Bar.

After the entering the command, the user will see the selecting window, and may type in or select a drive, a directory, a subdirectory if it is required, and the File Name.

The file-type box offers the user the option to display AutoCAD .dwg, .dxf, AllenCad .flx drawing files, or all three.

### Read-Only Option

The user may activate the option Read-Only in the dialog box used to open files. To do that, click on the control box beside Read-Only. A cross inside of that box means that this function is active.



The function Read-Only allows the file to be opened in order to be read, but it may not be modified. This is a protective mechanism against inadvertently (or, perhaps, by an unauthorized user, purposefully) modifying or changing the file. After opening in Read-Only, the file can be modified in any way, but the modifications will not be saved.

If the user should try to save a Read-only file, the program will send an error message and ask, at the same time, if the file should be saved under another name. If the changes are to be saved, then the user should answer <Yes> and type another file name in the dialog box that appears.

### **Find...**

If the user does not know the exact name or the storing location of a drawing or file, she or he can look for it by using the option **Find**. The dialog box *Find File* will be opened. Select the drives to be checked. Enter the file name of the drawing to be found. The user may use the known wild cards <\*> (for any group of characters) and <?> (for any single character).

After the entries are confirmed the program will search for corresponding file names in the indicated drives. All files matching the search criteria will be listed in a result window. The size, construction date and time as well as the location will also be indicated. The user should select the desired file from this list and confirm his or her choice.

---

### **See also:**

NEW  
SAVE  
SAVE AS..  
SAVE ALL  
CLOSE

### **Close a Drawing**



**Command: CLOSE**

**Alias CL**

**Located:**

- 1. Pull-down Menu: File - Close**
- 2. Top Function Bar**
- 3. Or Enter Command or Alias at the Command Line**

Closing a drawing indicates that the work on this drawing is finished. The program remains open. If a drawing has been changed before closing there will be a prompt asking whether the file should be saved. This is to prevent unintended loss of work.

If the user decides to save the drawing, the program will automatically switch to the standard routine to save the drawing under a new name.

Select the command CLOSE in the menu *File* or press the keys CTRL+F4. It is also possible to close the drawing window, as in all Windows programs, by double clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

---

### **See also:**

OPEN

NEW  
SAVE  
SAVE AS...  
SAVE ALL  
Save Drawing File



**Command: SAVE**

**Alias S**

**Located:**

1. **Pull-down Menu: File - Save**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

**Save**

The command SAVE causes the drawings in the active window to be saved. At this time the file will be saved under its present name. The file is "saved" in the same drawing format and to the same storage device and area as was specified at the time of opening. The file last saved in this area will be replaced.

The command to save can also be invoked from the menu *File* or by clicking on the symbol *Save* in the Function Bar.

Local variables will be saved along with the drawing and the enclosed drawing entities current at the time the automatic save occurs. Such settings as the current coordinate system, the dimensioning mode and other adjustments will remain at the user's disposal.

**Auto-Save**

To avoid unforeseen loss of data we strongly recommend periodically saving drawings while working on them. For this purpose use the function Save Automatically that can be activated in the configuration dialog window. While an automatic save is occurring the current drawing will not be closed.

---

**See also:**

SAVE AS...  
SAVE ALL  
OPEN  
NEW  
CLOSE

**Save As...**

**Command: SAVEAS**

**Alias SA**

**Located:**

1. **Pull-down Menu: File - SaveAs**
2. **Or Enter Command or Alias at the Command Line**

The command SAVEAS saves the drawing in the current window, as the command SAVE does. However, it allows the user to save the drawing as a different type, name, or in a different area. The command SAVEAS will require that the user answer these questions, all contained in the dialog box:

Filename  
File Type  
Drive  
Folder

The command will then save the work done as a specific file type (.dwg, .dxf or .flx), under a new filename, or in a new storage area. At the same time, a small bitmap (thumbnail) image of the drawing file is also created and stored with the file. This enables users to view the image before opening the file, thereby simplifying file identification.

---

**See also:**

SAVE  
SAVE ALL  
CLOSE

## Save All

**Command: SAVEALL**

**Alias SAL**

**Located:**

1. **Pull-down Menu: File - SaveAll**
2. **Or Enter Command or Alias at the Command Line**

The SAVEALL command will not only save the drawing in the current window but also all other open drawings.

---

**See also:**

SAVE  
SAVE AS...  
CLOSE

## Copy Region to Bitmap



**Command: BMPOUT**

**Alias BO**

**Located:**

1. **Pull-down Menu: File – Export Region To Bitmap**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

Call the command either by keyboarding BMPOUT or by selecting **Copy Region to Bitmap...** from the menu *Edit*.

The following options allow all **entities** in the region to be copied into a bitmap:

**Window**

Define a window by setting two points.

**View**

The complete current view will be exported into a bitmap.

**Extents**

The region to be copied is defined by the extent of the drawing.

Next, specify the **Width** and **Height** of the bitmap in Pixel Units (10...2000 pixels).

Finally, you may choose whether the bitmap should be written into a **File** or copied to the Windows **Clipboard**.

**Notes:**

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The bitmap remains in the clipboard until you clear it. Place new objects on it with the RENDER, BMPOUT, WMFOUT or other command, or Cut or Copy in other Windows applications.

## Copy Region to Windows Metafile



**Command: WMFOUT**

**Alias WM**

**Located:**

1. **Pull-down Menu: File – Export Region to Metafile**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

The following options allow the region to be copied into a WMF vector format file:

**Window**

Define a window by setting two points.

**View**

The complete current view will be exported into a bitmap.

**Extents**

The region to be copied is defined by the extent of the drawing.

Finally, you may choose whether the picture should be written into a **File** or copied to the Windows **Clipboard**.

**Notes:**

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The WMF vectors remain in the clipboard until you clear it.

## Command FLXRECOVER

**Command: FLXRECOVER**

**Alias REC**

**Located:**

1. **Pull-down Menu: File - Recover**
2. **Or Enter Command or Alias at the Command Line**

Beginning with the version 4.01 (Service Pack2) release, the FLX drawing format modifications permit the partial recovery of damaged drawing files. This utility, called by the FLXRECOVER command, is therefore available to FLX files saved in this (or later) release format. For damaged drawing files saved in AutoCAD DWG or DXF format, applicable AutoCAD recovery utilities should be used.

Usually, the need to use this utility becomes apparent when you experience difficulties opening an existing drawing file. File damage may apply to only a portion of the drawing database and the recovery utility will help you to salvage undamaged drawing elements from the file.

To use the recovery routine, simply issue the FLXRECOVER command from the pull-down menu selection or at the command prompt and select the damaged filename in the dialog presented.

Drawing Navigator...



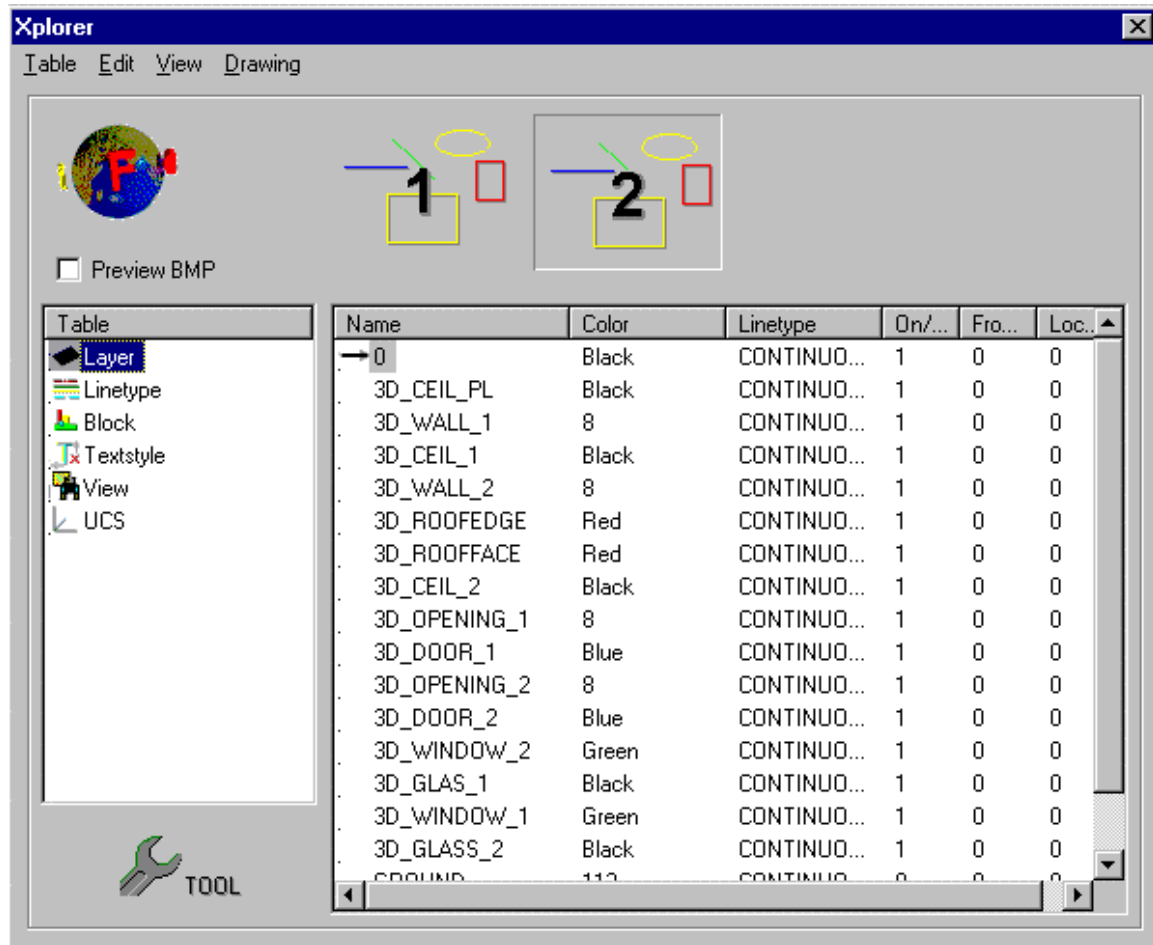
**Command: XPLORE**

**Alias XPL**

**Located:**

1. **Pull-down Menu: File– Drawing Xplorer**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The Drawing Xplorer offers you fast and uncomplicated access to important information on the currently open drawing(s). Furthermore, you can use the Xplorer to alter various properties of the current drawings.



More precisely, the Drawing Xplorer is a tool to manage the named objects in the drawing database tables. With it, you can manage the tables and table entries of layers, linetypes, textstyles, views, user coordinate systems, and blocks.

Type in the command `XPLORER` to activate this tool or select *Drawing Xplorer* from the *File* menu.

The *Xplorer* window displays the preview bitmap of the currently open drawings in the area underneath the menu bar. You switch to another drawing table, click the appropriate bitmap. A ToolTip over the corresponding bitmap displays the drawing name including drive and path.

The left-hand list box titled *Table* serves to select and access a specific drawing database table. The table entries with all its named objects and characteristics will then be displayed in the right-hand list box.

In detail, the drawing Xplorer permits the following:

- Delete unused, not referenced table entries (allowed only for the active drawing)
- Rename table entries (allowed only for the active drawing)
- Set the current layer, current linetype, current textstyle, current user coordinate system, or current named view (allowed only for the active drawing) by double clicking to one of the table entries
- In case the *Block* table is the current one, you can insert a block by double clicking to its name.

Clicking the *Tool* button will invoke the main corresponding function associated to the active table.

For example, if *Layer* is the Drawing Navigators current table, you will call the Layer Manager dialog window by clicking the *Tool* button. This way, you might create new layer entries.

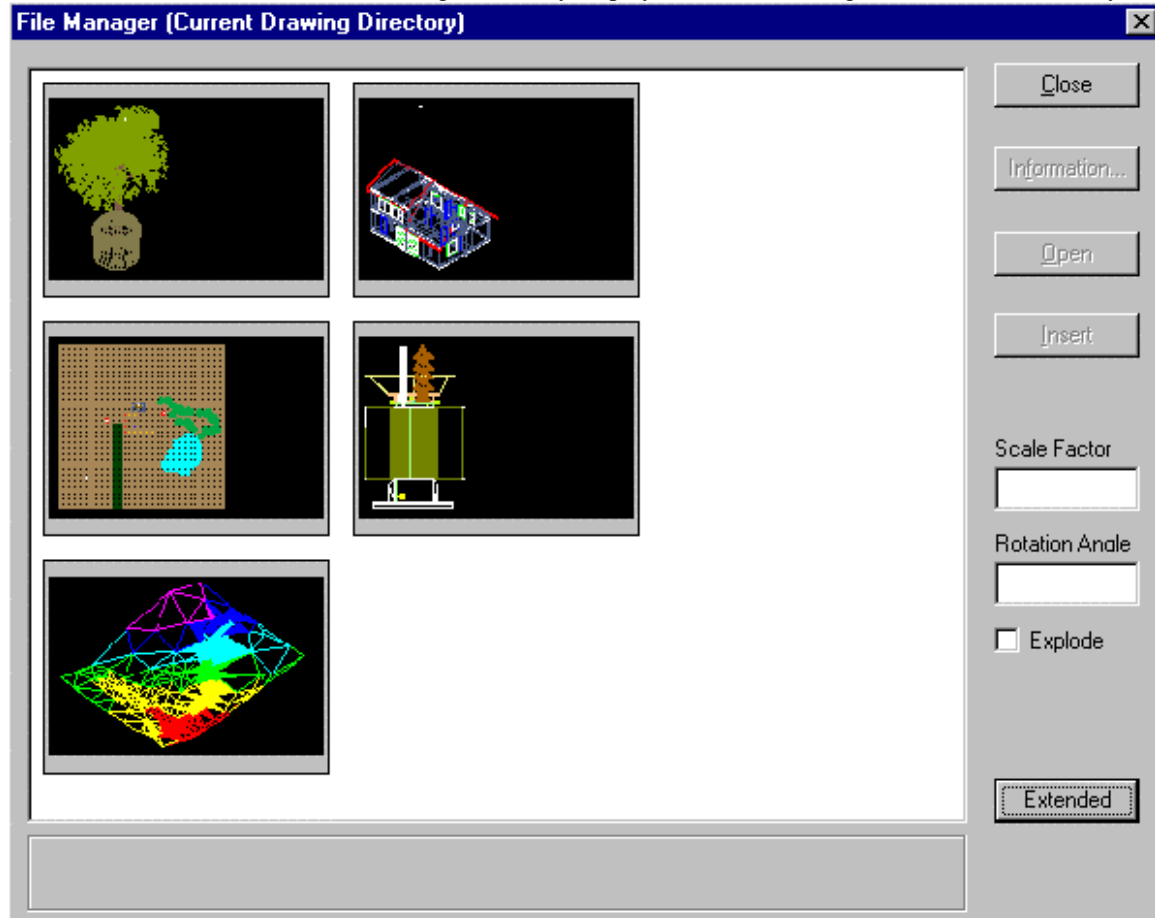
## File Manager

### COMMAND: FILEMAN

#### Located:

1. Pull-down Menu: File – File Manager
2. Or Entered Directly at the Command Line

The Fileman command (File>File Manager) visually displays all .FLX drawings in the current directory.



## Delete Lock-Files

### Command: DELFLK

#### Alias DLK

#### Located:

1. Pull-down Menu: File – Delete Lock File
2. Entered Directly at the Command Line

The program contains a function for deleting the so-called lock-files, called from the menu *File*, or entered by typing the command DELFLK.

Lock-Files are security files that are created by the program when a drawing file is opened. They carry the

same name as the original file and can be identified by the extension **.flk**. Lock-Files will be deleted automatically, if the drawing files are closed in the normal manner.

This procedure avoids two or more users working on one drawing at the same time when in a networking environment.

If the program is unexpectedly interrupted and the files have not been closed before the interruption, the same situation will occur. If the user tries to re-open a file after an interruption he or she will get the message *file locked*.

In order to re-gain access to the drawing, the corresponding Lock-Files must first be deleted.

In order to delete these files, first enter the command. Next, the user must perform the standard steps for the selection of files (similar to the steps in opening files). The program will offer the extension **.flk** as the desired file type.

Delete the offending Lock-File, and the drawing file can again be opened.

## Send Drawing(s)...

**Commands:** SENDFLX, SENDALL

**Alias** SD, SDA

**Located:**

1. **Pull-down Menu: File - Send**
2. **Or Enter Command or Alias at the Command Line**

The command SENDFLX allows you to send the current drawing via MS Mail, MS Exchange Server or Internet Mail.

The command SENDALL allows you to send all active drawings via MS Mail, MS Exchange Server or Internet Mail.

In the dialog below indicate which service you wish to use and have previously configured. You will be prompted to setup a selected service if it has not been previously configured.



## Drawing Output





## **Command: PRINT**

**Alias PT**

### **Located:**

- 1. Pull-down Menu: File – Print**
- 2. Top Function Bar**
- 3. Or Enter Command or Alias at the Command Line**

Start the output of a drawing on the printer/ plotter or plotter by selecting the option *Print ...* or by entering the command PRINT.

Select one of the options available in the options bar or menu (depending upon which you have selected for use in Desk Manager under Options. These options allow the user to determine the size and position of the print zone or the area of the drawing to be output. If you have Alternately, you may choose to use the first letter of the option to indicate your selection.

### **View (V)**

The currently visible window of the drawing will be copied on the print.

### **Border (B)**

This option determines the print zone so that all objects included in the drawing are covered.

Note that before using this option the drawing must be regenerated (Command REGEN).

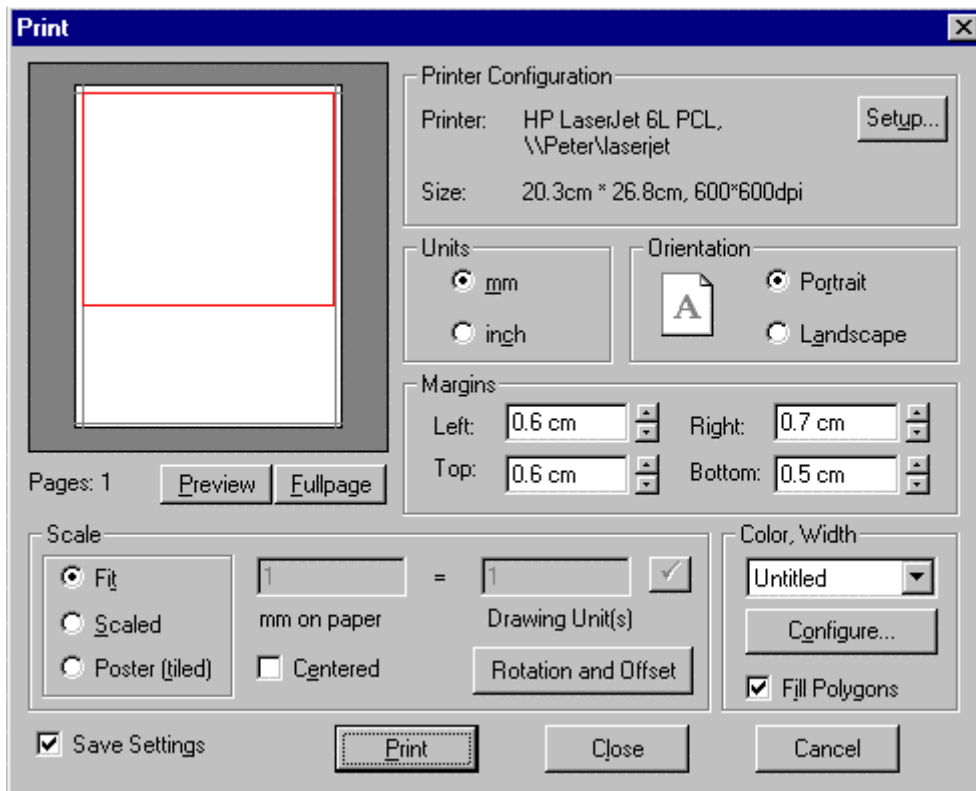
### **Window (W)**

Window enables the user to determine the print zone by selecting a window in the drawing. Indicate or enter the coordinates of two diagonal opposite corner points inside the drawing.

The command PRINT controls output of drawings to printers or plotters configured on your system. It uses the Windows print drivers and by this requires that the appropriate drivers have been set up under Windows.

### **The Print dialog box**

After defining the size of the print, the Dialog Box *Print* will open, and additional print parameters required for the orientation of the print in relation to the edges of the paper format and to the picture size must be set.



The Dialog Box is divided into the areas Configuration, Layout, Color/ Width, Pages and Scaling. It also contains a preview window and selections for *OK*, *Escape* and *Setup*.

The *preview window* will assist the user in controlling the effects of setting changes in Layout and Scaling.

### Printer Configuration Area

The Printer/ Plotter configuration area contains the options for the current Printer/ Plotter, including the desired paper size and print resolution (in dpi).

### Setup

Selection of the *Setup* switch allows the user to influence such parameters as the paper feed (printer/ plotter drawer), paper size, format position, resolutions and the number of the copies to be run.

Note: Only use settings supported by the selected device in the SETUP Dialog Box.

### Layout Area

The parameters of the Layout Area control the size and position of the print on the available format.

#### Margins

Controls the area that is not to be printed (in mm) at the sides of the page. Enter the desired values in the proper window or select with the Cursor at the right edge of the fields and set the values for margin width.

#### Fit

Selecting the option *Fit* will result in a format-filling picture of the object in the print zone.

This option is activated by default during any print or plot output. To de-activate, click the control field. A cross in the control field displays the active status of the option.

Note that when the option *Fit* is active, it is not possible to set the options for scaling.

#### Fill Area

*Fill Area* will cause filled polygon areas to be displayed with fill in the print output. If the option not

activated, only the contours of filled areas will be output.

The default position of this option is *Inactive*. To activate select the control field. A cross in the control field displays the *Active* status of the option.

### **Center**

*Center* has a very close relation to the option *Fit*. Because the relation between the pages of the print zone remains unchanged during *Fit*, a format-filling picture may only be reached either to the length *or* width of the picture, but not with both.

*Center* results in a centered placement of the print zone on the paper in relation to the side that could *not have been fitted* to the margins of the page.

Note that *Center* is only available when using the *Fit* option.

### **Area Scaling**

*Area Scaling* allows the user to define the exact scale of the drawing.

Note that *Area Scaling* is only available when the *Fit* option has been *de-activated*.

The definition of the picture scale is made in two steps:

1. Define the size of a drawing unit in mm. Place the cursor in the input field for the drawing unit - mm and enter the desired value.
2. In the input field Scale enter the scale for the print output or select a value by means of the arrow keys at the right side next to the input field.

The user can view the effects of the inputs in the preview window.

### **Area Pages**

The *Area Pages* contain a Dialog. Enter the number of pages to be used in the drawing.

Multiple pages will result when the user has selected a picture scale that results in a print output larger than the selected page size (minus the defined margins).

### **Area Color / Width**

By choosing the *Configure* button, the Color/Width Configuration Dialog will appear.

*Using saved stroke width-attachments*

Using saved stroke widths facilitates outputting drawings and assures constant settings.

In the selecting window of the area *Color/ Width* in the Dialog Window *Print* the current stroke width will be displayed. Selecting the arrow at the right side of the window will display a list of all saved configurations. Select the desired configuration.

## **Printer Setup**

**Command: PSETUP**

**Alias PS**

### **Located:**

1. **Pull-down Menu: File – Printer Setup**
2. **Or Enter Command or Alias at the Command Line**

The command PSETUP (Printer/ plotter Setup) or the selection of the option **Printer Setup...** from the *File* menu will start the selection and installation of the desired printer or plotter.

This procedure for installing printers calls up the standard Windows printer/plotter installation sequences.

The user will find detailed instructions regarding the operating mode of the specific options for the printer installation and for installing and connecting a printer or plotter in the Windows Manual or in the output device manual. Only the most important steps for printer/plotter installation are reviewed in this text.

The command will open a dialog box.

## Dialog Control Elements

### Printer

In the field Printer/ Plotter the user will specify the printer/ plotter or plotter to be used in the output of the drawing. Select either one of the standard devices defined in the Dialog Box control, or select another printer or plotter that is available under Windows. Available output devices are displayed in a list in the selection area under the heading *Special Printer*. To see the complete list, select the arrow at the right edge of the window. Select from the list of the installed output devices the desired printer.

### Orientation

Select the format **Portrait** or **Landscape** for the desired alignment of the paper.

### Paper

The size of the paper as well as the feed from a particular printer or plotter is controlled from the field **Paper**. Select the proper size and feed.

### Options

The **Options** switch (at the right edge of the Dialog Box) opens another Dialog Box that contains parameters that are mainly a function of the capability of the device to be connected.

### BATPROC

## Command BATPROC (Batch Processing)

### Alias BT

#### Located:

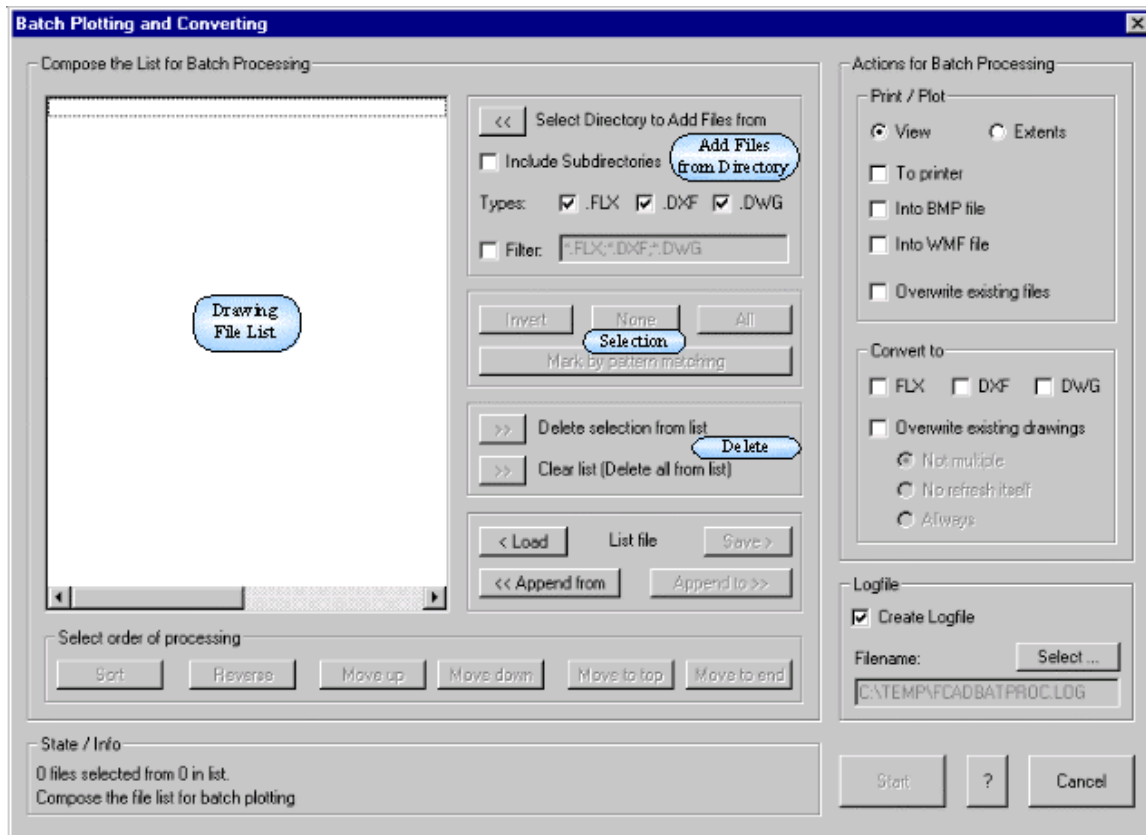
1. **Pull-down Menu: File – Batch Processing**
2. **Or Enter Command or Alias at the Command Line**

### Introduction

The command BATPROC makes it possible for you to arrange a list of drawing files and to do something with each drawing on this list. There are many built in capabilities to build the list and manipulate it. After arranging the list you can plot each file on the list not only to the printer but also to a Bitmap file (.BMP) file or into a Windows Metafile (.WMF). Moreover you can batch-convert drawings from one format (of the set DXF, FLX or DWG) to another of this set.

### Preferences and Start of the Command

You can start the command simply by typing "BATPROC" or by selecting the menu point BATCH PROCESSING inside of the menu TOOLS. Now the following dialog box appears at the screen:



This dialog is divided in two main parts, on the left side you find the functions to compose and manipulate the list, while on the right part the actions for the batch process can be defined. Following the meaning of the fields will be described.

## Exiting the Program

**Command: EXIT**

**Alias X**

**Located:**

1. **Pull-down Menu: File – Exit**
2. **Or Enter Command or Alias at the Command Line**

It is very important to end every working session correctly (otherwise you might lose work that you have performed on a drawing file). The best way to terminate the session is by executing the command EXIT in the menu *File*.

The command EXIT is correctly performed in any of these fashions:

1. Select the command Exit in the pull-down menu File and confirm.
2. Press the keys Alt + F4
3. Double-clicking to the system menu area (see Windows manual)

If there are open drawings that have not been saved, the program will send a safety question. If you confirm, the dialog *Save File As* will be called.

After saving the drawings (see page 7) the current working session with the program will be finished.

---

**See also:**

CLOSE

SAVE

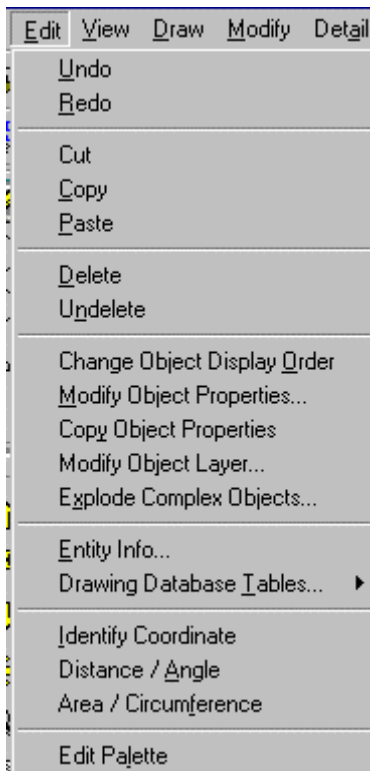
SAVE AS...

Drawing History

At the bottom of the File Menu is the Drawing History box. Here, the recently worked upon drawing files are displayed. Clicking on the filename will open that file in the drawing editor.

## Edit

### "Edit" Pull-down Window



The Edit pull-down menu contains command instructions that edit drawing elements. Additional editing commands may be found under the Modify pull-down menu. For a description of each command, please refer to the specific topics.

## Undo



**Command:** UNDO, U

**Alias** U

**Located:**

1. **Pull-down Menu: Edit – Undo**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

Invoking the command U (for Undo) will cause the last command made to be withdrawn and all changes in the drawing caused by this command to be canceled.

The command can be repeated until all working steps stored in the Undo List have been canceled. The program will then send out the message "Undo List is empty".

**Note:** The command U allows you only to reverse drawing database modifications. It does not undo modifications of the view. Use ZOOMBACK to restore previous views.

---

**See also:**

Command REDO

Command UNDO (Lisp)

## Redo (Restore an Undo Operation)



**Command: REDO**

**Alias RD**

**Located:**

1. **Pull-down Menu: Edit – Redo**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The command REDO offers the ability to Redo an Undo. The state of the task is restored to the last state prior to issuing the command U (or UNDO).

This is only possible if no other command has been executed between the commands U (or UNDO) and REDO, that have changed the drawing database.

REDO is used in this fashion:

- Call the Redo function by clicking the Redo symbol, or
- Enter REDO at the command line.

**Note:**

Only the last relevant working step can be restored.

---

**See also:**

Command U

## Cut Selection to Clipboard



**Command: CUTSELECT**

**Alias CT**

**Located:**

1. **Pull-down Menu: Edit – Cut Selection to Clipboard**

2. **Top Function Bar**
3. **Palette Edit**
4. **Or Enter Command or Alias at the Command Line**

With the command CUTSELECT selected objects are copied to a **AllenCad-specific** clipboard (as with the command COPYSELECT)

However, unlike COPYSELECT, with CUTSELECT, the originally selected objects are deleted from the drawing.

The command PASTE will insert the objects from the PCAD clipboard into any open drawing. PASTE will request the insertion point.

---

**See also:**

Command COPYSELECT  
Command PASTE  
Command DELETE

## Command COPYSELECT



### Command: COPYSELECT

Alias CST

**Located:**

1. **Pull-down Menu: Edit – Copy Selection to Clipboard**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

The command COPYSELECT copies selected objects from the drawing, placing them onto a **AllenCad-specific** clipboard. The selected objects remain in the drawing.

The command COPYSELECT is useful especially to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

**To copy objects onto the clipboard:**

1. First specify the base point for later insertion of the objects.
2. Then select the objects to be copied to the clipboard.

**Command line example:**

```
> COPYSELECT  
Insertion base point:  
Select objects:
```

**Note:**

Do not confuse this command with the COPY command found in the *Modify* menu that makes copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another drawing. They will remain on the clipboard until you place new objects on it with the COPYSELECT command.



---

**See also:**

Command PASTE

Command CUTSELECT

## Paste Selection from Clipboard



**Command: PASTE**

**Alias PST**

### Located:

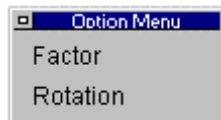
1. **Pull-down Menu: Edit – Paste Selection From Clipboard**
2. **Top Function Bar**
3. **Palette: Edit and Modify**
4. **Or Enter Command or Alias at the Command Line**

The PASTE command copies objects from the **AllenCad-specific** clipboard to point you select in the current drawing. These objects have typically placed on the clipboard by a preceding COPYSELECT or CUTSELECT command.

Use the command to copy graphic data from other drawings into your current drawing.

### To paste the objects from the clipboard into the current drawing:

1. Choose the PASTE command.



2. You are presented with the option dialog. If you plan to scale or rotate your objects during insertion, select your option now. You will be offered the opportunity to scale and rotate as a conclusion to the command later if you do not choose at this point, however you will not be able to visualize the scaled or rotated objects in advance, if you wait until later.
3. Specify the insertion point by entering coordinates of the insertion point, OR
4. Drag the highlighted objects into position on the drawing. The objects of the clipboard contents appear highlighted (as well as scaled and/or rotated - if you selected this in the last step), and attached to the cursor during the exercise. This feature allows you to visualize object placement prior to selecting the insertion point with a mouse click.
5. Specify the scale factor for insertion – you can specify a scale factor at this point (if you have not done so already) and have the objects inserted at a larger or smaller scale than they were originally. Note that the relative position of these objects to each other remains the same (is scaled too) during the operation).
6. Specify the rotation angle (if you have not done so already) – you can choose to rotate the object by any input angle during the exercise.

### Example:

```
> PASTE
Insertion point:
Scale factor <1>:
Rotation angle <0>:
```

When inserting the objects from the clipboard you can also specify the scale factor and/or rotation angle before you indicate the insertion point by choosing the appropriate option from the context bar when prompted for the insertion point:

```
> PASTE
Insertion point: Factor
Scale factor <1>: .5
Insertion point: Rotation
Rotation angle <0>: 90
Insertion point:
```

**Note:**

The objects selected with COPYSELECT remain on the clipboard, ready to be placed in another drawing, until you place new objects on it with the COPYSELECT command.

To copy objects within the current drawing it is recommended that you use the COPY or ARRAY command found in the *Modify* menu.

---

**See also**

Command COPYSELECT

Command CUTSELECT

Delete



**Command: DELETE**

**Alias DEL**

**Located:**

1. **Pull-down Menu: Edit – Delete**
2. **Top Function Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

During the creation of drawings, or, later during the editing, changing or detailing of drawings, it may be necessary to delete certain parts of the drawing.

After calling the command, the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for object selection. These same options will alternately appear in a dialog menu, depending upon your preferences specified in Desktop Configuration.

WPolygon	CPolygon	Fence	Last	Previous	Remove	Window	Crossing	Single	All
----------	----------	-------	------	----------	--------	--------	----------	--------	-----

When deleting, as many objects can be selected as the user desires. When the object selection is complete, the selected entities will be deleted without another confirmation, by simply pressing ENTER.

---

**See also:**

Command UNDELETE

Undelete: Withdraw Deletions



**Command: UNDELETE**

## Alias UD

### Located:

1. **Pull-down Menu: Edit – Undelete**
2. **Top Function Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

Entities that have been deleted by mistake can be restored by the command UNDELETE.

Call the command

- By clicking the UNDELETE symbol;
- By selecting the option **Undelete** in the menu *Edit*; or
- By keyboarding UNDELETE in the command line area.

This will return objects deleted by the last command that removed objects from the drawing.

---

### See also:

Command DELETE

Command DRAWORDER

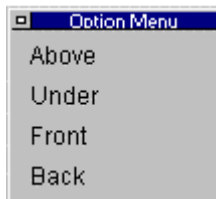
## Command: DRAWORDER

### Alias DO

#### Located:

1. **Pull-down Menu: Edit – Change Object Display Order**
2. **Or Enter Command or Alias at the Command Line**

The **Command\_DRAWORDER** command changes the drawing and plotting order of any object in the AllenCad drawing database. In addition to moving objects to the "front" or "back" of the sort order, you can order objects relative to another object (that is, above or below a selected object). After issuing the command, you are asked to pick entities to re-order. You are then presented with the following dialog, once this selection is complete:



You simply select the option that you want and then select the entities that you want the original entities placed in relation to.

AllenCad differs from AutoCAD, in that object database order is changed directly, using the Command\_DRAWORDER command. Different object sort methods are therefore not supported, and a SORTENTS system variable is not required in AllenCad.

See Also HIDE, FHIDE

## Modify Object Properties...



**Command: PROEDIT**

**Alias PED**

**Located:**

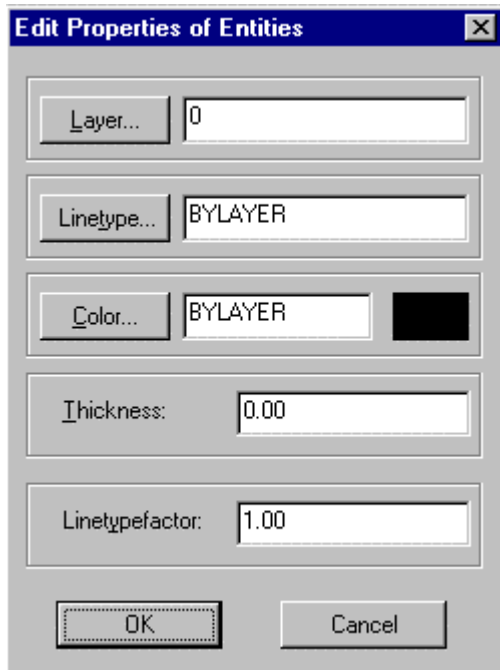
1. **Pull-down Menu: Edit – Modify Object Properties**
2. **Left Side Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

The function **Modify Properties...** invokes the dialog boxes that allow you to set the properties of entities, and make changes in **Color**, **Layer**, and **Linetype**.



As with the other commands in this set, first select one or more objects, using the selection dialog shown above, if required.

Then press RETURN, or use the right-hand cursor button.



The dialog box that appears displays the current properties Layer, Linetype, Color, Thickness and Linetype Factor (also called Linetype Scale Factor ) of the entities selected. You may then edit these properties.

### **Copy Object Properties**



**Command: COPYPROP**

**Alias CP**

#### **Located:**

1. **Pull-down Menu: Edit – Copy Object Properties**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

This command helps you to change the properties (layer, linetype and color) of selected objects to those of a selected entity.

#### **To change the properties of a selection:**

1. Select a single reference entity.
2. Select one or more objects to be changed.

#### **Command line example:**

```
> COPYPROP
Select entity for reference properties:
Select objects to modify. Select objects:
```

Modify Object Layer ...



**Command: PROPLAYER**

**Alias PP**

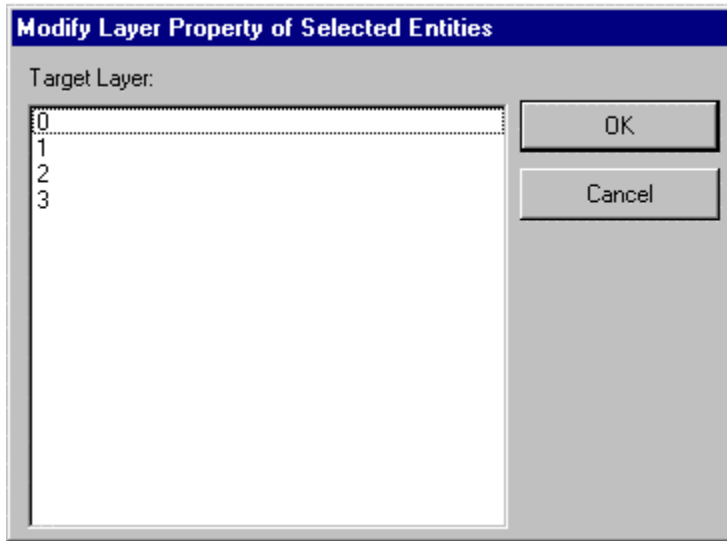
**Located:**

1. **Pull-down Menu: Edit – Modify Object Layer**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

This command transfers selected objects from their current layer to a chosen layer.

**To modify the layer property of drawing objects:**

1. Select the drawing entities to be transferred to another layer.
2. In the dialog box (shown below) choose a target layer and confirm the selection by OK.



**Explode Complex Objects...**



**Commands: EXPLODE, XPLODE**

**Alias EX. XPL**

**Located:**

1. **Pull-down Menu: Edit – Explode Complex Objects**
2. **Palettes: Edit and Parts**
3. **Or Enter Command or Alias at the Command Line**

Parts and other complex objects, like polylines, crosshatches, and dimensioning, inserted in the drawing constitute one single drawing object.

In this mode, it is not possible to edit any of the individual entities that make up the part. If, for any reason, there is a requirement to edit one of the entities comprising a part, then the part must be "exploded" (broken into its individual entities), the element(s) edited, and then the component parts reassembled again.

The explosion of complex objects is executed by the command EXPLODE, or by selecting and activating the option **Explode Complex Objects...** in the *Edit* menu.

When the object selection is complete, the objects (parts, polylines, hatches, or dimensions) will be

exploded.

Now you may choose any of the entities for individual editing.

**Notes:**

1. It is important to realize that, if there are attributes attached a part to be exploded, these attributes will be lost. Instead, the attributes defining names will appear.
2. Exploding a part does not change the part definition in the drawing. Another part or parts with the same name and original definition can be re-inserted into the drawing at any time.
3. To change the part definition in the drawing, you must modify the part components after exploding, then create a new internal or external part with the PARTEXP or PARTDEF commands, using the old part name. The program will ask you to confirm the re-definition when you do so.

## Entity Info...



**Command: EINFO**

**Alias LI**

**Located:**

1. **Pull-down Menu: Edit – Entity Info**
2. **Left SideBar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

The EINFO command prompts you to select objects for identification. It then opens the text window and displays object information as in the following EINFO report on a selected line entity:

```
> EINFO
Select objects:
  1 selected.
Select objects:
*** 1 selected. ***
Type:      LINE
Layer:      0
Line type:  Layer's line type
Color:      Layer's color
Handle ID:  "A"
Length:     0.44
Thickness:  0.00
Start Point (UCS):  5.09, 5.31, 0.00
End Point (UCS):    5.53, 5.35, 0.00
Delta (UCS):0.44, 0.04, 0.00
Start Point (WCS):  5.09, 5.31, 0.00
End Point (WCS):    5.53, 5.35, 0.00
Delta (WCS):        0.44, 0.04, 0.00
```

Projected length (in xy-plane): UCS: 0.44      WCS: 0.44  
Angle in xy-plane:    UCS: 6    WCS: 6  
Angle to xy-plane:    UCS: 0    WCS: 0

**Tip:** You may use this information in external reports, etc, by highlighting the text and using Windows' Copy (Ctrl C) and Paste (Ctrl V) key definitions to copy the text from the text window first to the Windows Clipboard (Ctrl C) and then to the external application (Ctrl V).

### Drawing Database Tables...



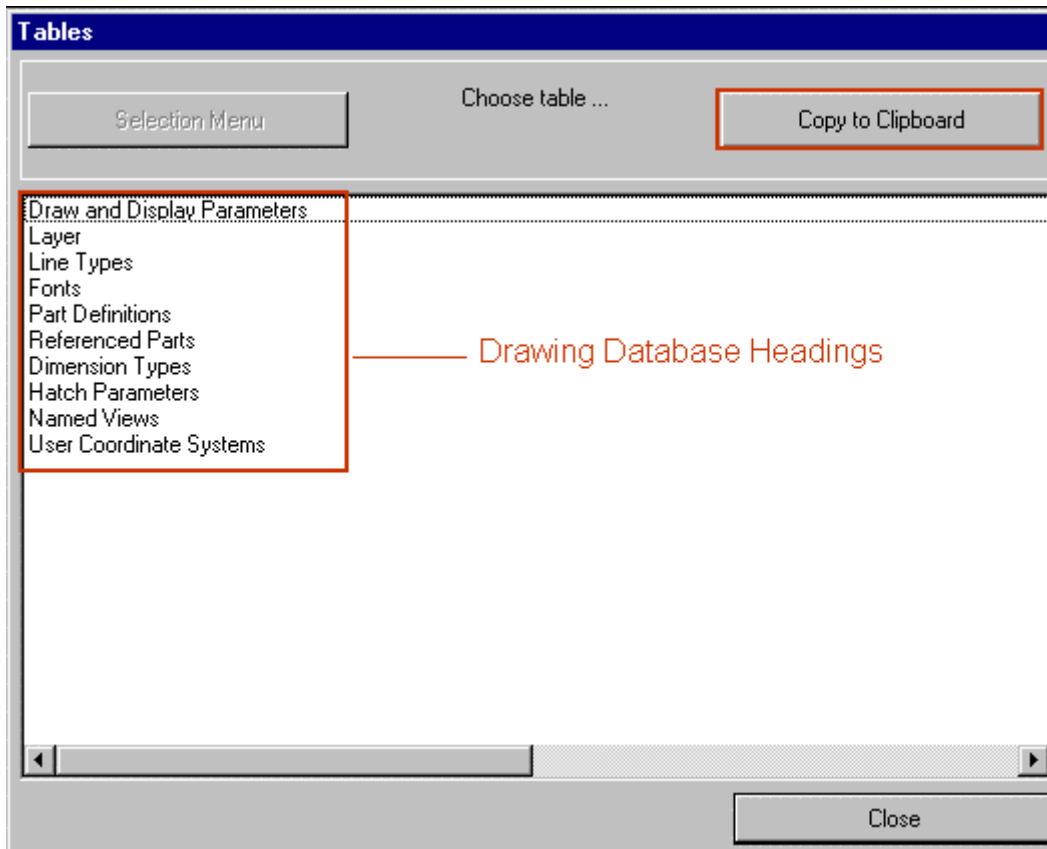
**Command: TABLES**

**Located:**

1. **Pull-down Menu: Edit – Drawing Database Tables-Tables Info**
2. **Entered Directly at the Command Line**

Call the command by keyboarding TABLES or select **Drawing Database Tables...** from the menu *Edit*.

You will be presented with the first dialog in which you can specify the database heading for the information that you require.



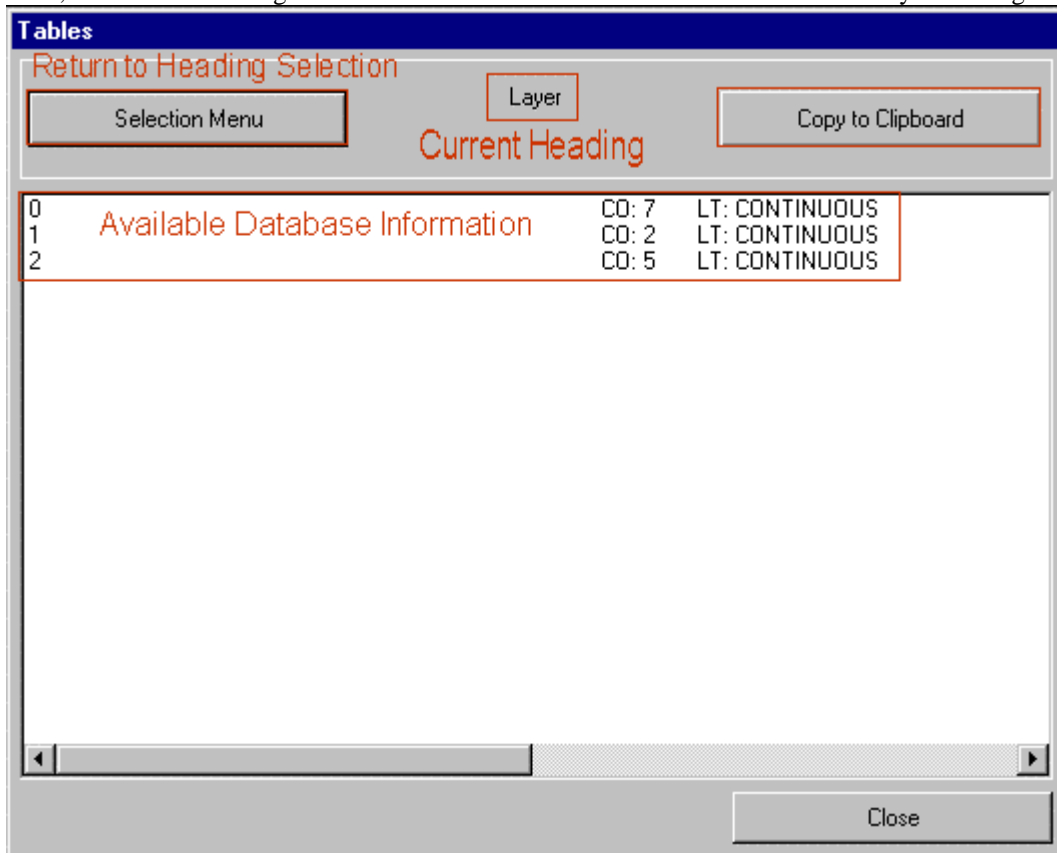
As can be seen above, these overviews can be requested:

- Draw and Display Parameters Layer
- Line Types
- Fonts



- Part Definitions
- Referenced Parts
- Dimensioning Types
- Hatching Parameters
- Named Views
- User Coordinate systems

The **Clipboard** option will copy the displayed heading information to the Windows Clipboard. Next, we select a Heading to examine database contents. Here we will select the Layer heading:



Here we can see drawing layer information stored in the database. Again, the **Clipboard** option will copy the displayed database information to the Windows Clipboard.

## Command TABSERVICE



Command TABSERVICE

Alias TB

### Located:

1. Pull-down Menu: Edit – Drawing Database Tables – Tables Maintenance
2. Top Function Bar
3. Palette Edit
4. Or Enter Command or Alias at the Command Line

The TABSERVICE command offers the user the opportunity to simply alter and maintain parameters used in the current drawings drawing database. The command is located on the Edit Pull-down menu as the "Drawing Database Tables – Tables Maintenance" selection. It can also be typed in at the command line, if preferred.

When the command is issued, you will be presented with the dialog pictured below. Here you can select the parameter that you wish to modify:



If you wish to modify Layers, for example, you can eliminate all empty (containing no entities) or unused table entries, by clicking on the Purge Table button. You can also rename or delete items, such as layers, by clicking on the other two buttons.



Purge

**Command: PURGE**

**Alias PRG**

**Located:**

- 1. Pull-down Menu: Edit – Drawing Database Tables – Purge Tables**
- 2. Or Enter Command or Alias at the Command Line**

The PURGE command cleans the drawing file of unused named objects. You can remove blocks, layers, linetypes, text styles, or dimension styles that you have defined or loaded before but are not utilized in the drawing.

The command line prompt of the command is as follows:

> PURGE

Purge un-referenced symbols (Blocks, Dimstyles, LAyer, LTypes, Textstyles or All):

The PURGE command can be utilized on drawing database tables and table entry names *selectively*:

- You can use the command for deletion of unused named objects within only one of the drawing database tables. To do so, first select the type of table to be purged. You will then be prompted with the names - one after the other - of unused named objects of the specified table.
- If you want to purge all drawing database tables, reply *All* to clean up the entire drawing in one operation. You may respond with a name pattern to limit the selection to certain names by using

wildcard specifications (for example: TEMP\*).

### To purge a drawing file

To purge all un-referenced symbols in any drawing database table, respond as follows:

```
> PURGE
Purge un-referenced symbols (Blocks, Dimstyles, LAyer, LTypes, Textstyles or All): ALL
Names to delete <* = All>: *
Confirm each name? <Y>: N
```

### Notes

Purging a drawing can significantly reduce its file size. This operation is recommended before transmitting, archiving or backing up a drawing file.

Some standard objects cannot be purged: These include the layer named 0, the linetype CONTINUOUS, and the text style STANDARD.

The PURGE command does not handle named views and user coordinate systems, because these named objects are never referenced by another component of the drawing. You may delete unused views and UCS. The corresponding commands (VIEW and UCS) provide options to delete named items from these tables.

With nested blocks in the drawing file, only one level of nesting is purged at a time.

### Rename

#### Command: RENAME

#### Alias RNM

#### Located:

1. Pull-down Menu: **Edit – Drawing Database Tables – Rename Table Items**
2. Or Enter Command or Alias at the Command Line

The RENAME command allows you to alter the names associated with blocks, layers, linetypes, text styles, dimension styles, user coordinate systems, and views.

### To rename a drawing database table entry

- First, specify the drawing database table that contains the item to be renamed.
- Next, enter the name of the named object you want to rename.
- Finally, type in the new name of the item.

### Example

```
> RENAME
Rename symbol (BLock, Dimstyle, LAyer, LType, Style, Ucs, or View): LA
Current name: dimensions
New name: dim
```

### Tip

You can rename these named symbols used within the current drawing from within the *Drawing Xplorer*.

## Identify Coordinate



Command: **ID**

#### Alias ID

#### Located:

1. **Pull-down Menu: Edit – Identify Coordinate**
2. **Or Enter Command or Alias at the Command Line**

The transparent command **ID** (identify coordinate) determines the coordinates of a selected point and then displays that information.

The function **Identify Coordinate** may also be selected from the menu *Edit*.

#### Using the command

Indicate the point to be determined with the Cursor. If it is a geometrically significant point of a drawing object (for instance, corner point, center of a circle, intersecting point of two lines etc.) use the Object Snap functions.

The coordinates will appear in the display.

```
> ID
Point: PI
X: 1.27 Y: 4.45 Z: 0.0
```

Distance / Angle



**Command: 'DIST**

**Alias D**

#### Located:

1. **Palette: Edit**
2. **Left Sidebar**
3. **Pull-down Menu: Edit – Distance / Angle**
4. **Or Enter Command or Alias at the Command Line**

The transparent command **Distance / Angle** will determine the distance or angle between two points and displays them in the text window:

1. The absolute distance (range);
2. The Distance on the X-, Y- and Z-axis;
3. The Angle on the XY-plane; or
4. The Angle on the XY-plane.

The points may be arbitrary points that do not have to be part of a single drawing object.

The function **Distance / Angle** operates as follows:

1. Determines the two points required by selecting them with the Cursor, using the Object Snap functions or by input of the coordinates; then
2. The distance and angle between the two points will be displayed.

The term *Distance* is understood to mean the absolute distance between points.

Example:

```
> DIST
First Point:
Second Point:
Distance: 0.37 Angle in XY plane: 41 Angle from XY plane: 0
Delta X: 0.28 Delta Y: 0.24 Delta Z: 0.00
```

The terms *Delta X*, *Delta Y* and *Delta Z* designate the distance between the two points in the direction of the respective axis of the coordinate system. All length values are given in drawing units.

*Angle in the XY-plane* measures the angle to the second point starting from the X-axis. It always displays the lesser-valued angle, no matter what the rotation direction.

*Angle from the XY-plane* measures the angle between the first and the second point from the XY-plane to the Z-axis. It is assumed that the first point lies on the XY-plane.

## Area / Circumference



**Command: AREA**

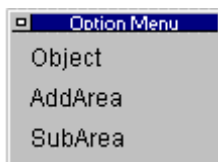
**Alias AR**

### Located:

1. **Pull-down Menu: Edit – Area / Circumference**
2. **Left SideBar**
3. **Palette: Edit**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

This function defines the circumference and the area of one or more drawing objects and displays the values. For objects that do not have width, the length instead of the circumference will be displayed. The areas of single objects can be also added or subtracted.

As soon as the command is issued, you are presented with the following option dialog:



These options are described below:

### Using Area / Circumference

To determine the circumference and the area of one or more objects proceed as follows:

Define the drawing object to be examined by tracing its outline or using the Object option. The preferred method of selection is by using the cursor and Object Snap functions, but inputting the coordinates of the points is also okay. Pressing RETURN at the request **Next Point:** will finish the selection. It is not necessary to select the first point again to close the boundary, as the program assumes a closure between first and last points. It also does not matter which direction you trace the outline in, so long as you complete the outline definition in either a continuous clockwise or anti-clockwise direction.

### Object: Regular Areas

"Regular" geometric areas (of circles, arcs, closed polylines) can be evaluated by choosing the **Object** option.

### Add or Subtract Areas

Select the first partial area or determine the points for describing the area geometry, as above. The determined values will be displayed in the text window.

Select the option *AddArea* to add a partial area, or *SubArea* to subtract a partial area, from the total area.

The values (circumference /length and area) for the total area will be displayed in the text window, after additions or subtractions are made.

When partial areas are added or subtracted from an area, the total area after the addition or subtraction is

regarded as the new (initial) parameter value for subsequent additions or subtractions unless the user explicitly selects a contrary option from the Options Bar.

The Options menu will display only that option that is not active at the moment.

---

**See also:**

System variable AREA

### Edit Palette



This command will place the palette containing Edit command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

### View

### "View" Pull-down Menu



The View pull-down menu contains commands that control the operator's view of the drawing. For a description of each command, please refer to the specific topics that follow.

### View Manager... (Saving / Retrieving Views)



**Command: VIEW**

**Alias V**

#### **Located:**

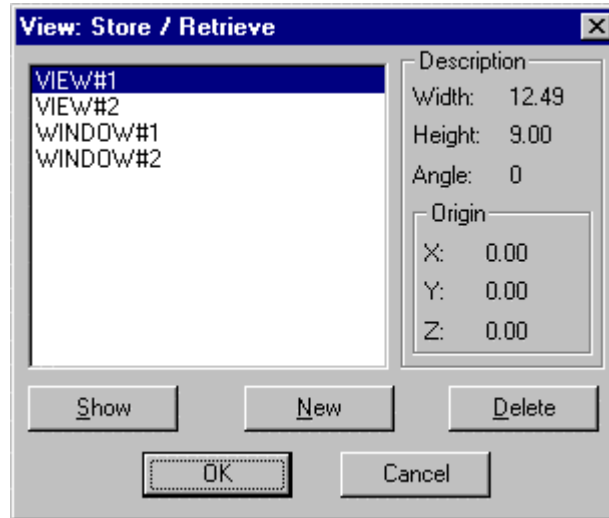
- 1. Pull-down Menu: View – View Manager**
- 2. Left Side Bar**
- 3. Palette: View**
- 4. Or Enter Command or Alias Directly at the Command Line**

It is convenient when working in either Model mode or in Layout Mode to save different views of the drawing for later retrieval and display. The view of the drawing is essentially the viewer's 3-dimensional

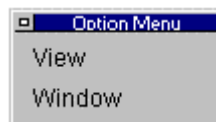
location and the direction of view toward the drawing. These are the elements that are stored in the named file and they correspond to the current drawing view when the view is saved.

### Save a Named View

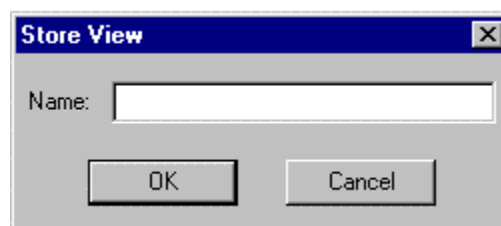
When selecting the option VIEW, the present portion will be saved as a user-defined view. Upon issuing the command, you will see the dialog:



Here we can see that we already have two views and two windows already saved. To save another, simply click on the **New** button. At this time you will be presented with the option:



If you choose view, the full screen image will be saved, whereas if you choose Window (a windowed view), you will be asked to define the window onscreen. This is done by picking upper left and lower right (or any diagonally opposite) window corners. After selection of option is complete and the window is defined, you are lastly presented with the dialog:



At this point, you specify the name for your view or window, and it will then be added to the list of other drawing views for later retrieval.

### Retrieve a Named View

A previously saved view can be retrieved through the same function, and can then become the present view. Follow these steps:

1. Choose one of the saved views in the dialog's list box;
2. Click on the **Show** button, located in the View Command dialog image above. The selected view



- will then become the present view;
3. Close the dialog box with OK to accept to display the chosen view. Choosing to abort will retain the previous view.

### Delete the Name of a Defined View

Views that are no longer needed, can be removed from the table of named views as follows:

1. Choose one of the names of a defined view in the list box;
2. Click onto the **Delete** button, located in the View Command dialog image above. The view will be removed from the list immediately.

Pan



Command: 'PAN

Alias P

### Located:

1. **Pull-down Menu: View – Pan**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias Directly at the Command Line**

The command PAN moves the visible portion of the drawing to the right, to the left, upward or downwards. It is now easy to look at objects that were previously not visible because they were off the viewing surface. In contrast to the ZOOM command, the drawing scale does not change.

The PAN commands are transparent ones that may be executed while other commands are being executed. It is possible, for instance, to execute the command PANDOWN (move downward) while entering points for drawing an element.

The user should remember that, when moving the drawing, it appears that the visible objects are moving in the direction of the command. The reason for this is that the PAN commands acts as does a photographer, to move the "camera", not the picture

'PANUP	Portion up
'PANDOWN	Portion down
'PANLEFT	Portion to the left
'PANRIGHT	Portion to the right

After selecting one of these commands, the visible drawing portion will immediately be moved in the opposite direction, allowing you to see more of the drawing in the direction you have indicated with PAN. The amount of movement is determined by a factor that is saved on the SETTINGS – View Parameters tab

---

### See also:

DPAN command

ZOOM command

## Command DPAN



Command: 'DPAN

Alias DP

**Located:**

1. Pull-down Menu: View – Dynamic Pan
2. Left Sidebar
3. Palette: 3D View
4. Or Enter Command or Alias at the Command Line

As described previously, panning means to move the display of the drawing in its current window without changing the magnification.

With the command **DPAN** you can simply grab the drawing and move it in any direction you want, any number of times, until you have the drawing view that you want. At this point, you terminate the command by pressing the ESC key or Enter (right mouse click) and then select the Exit option.

To pan the drawing in this manner, choose *Dynamic Pan* from the menu *View*. The cursor changes to an icon symbolizing a hand. Place the cursor anywhere in the current drawing, then left-click. Now drag in the direction you want the drawing to move, releasing the mouse button to release the "grip" on the drawing.

If you reach the edge of the drawing window, release the mouse button, move the cursor away from the edge, and left-click and drag again.

To exit the Dynamic Pan mode, press **ESC** or **ENTER**. You can also right-click to open the context menu that contains the *Exit* option. Alternatively, you can start a command via the pull-down menus or palettes to exit the dynamic pan (this works, as long as you do not call a transparent command from the menu or palette).

Beneath leaving the dynamic pan operation, the cursor menu (that you can open by clicking the right mouse button) also allows you to switch to dynamic zoom mode and to execute a Zoom Extents.

Dynamic Pan can be used transparently (in the middle of another command).

---

**See also:**

PAN command

DZOOM command

**Dynamic Zoom**



Command: 'DZOOM

Alias DZ

**Located:**

1. Pull-down Menu: View – Dynamic Zoom
2. Left Sidebar
3. Or Enter Command or Alias at the Command Line

With the command **DZOOM** you can zoom in or out to any extent within the current drawing window.

To zoom in this manner, choose *Dynamic Zoom* from the menu *View*. The cursor then changes to a

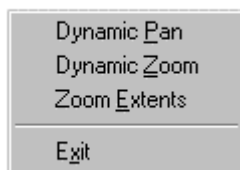
magnifying glass icon with a plus sign and a minus sign. To start the dynamic zoom process, click the left mouse button and drag the mouse, either up or down. Whether you zoom in or out depends on the direction of movement of the cursor. To zoom in, drag up (in the direction of the plus sign). To zoom out, drag down (in the direction of the minus sign).

If you reach the edge of the drawing window, you can release the mouse button, move the cursor away from the edge, and left-click and drag again.

To exit the Dynamic Zoom mode, press **ESC** or **ENTER**. You can also right-click to open the context menu that contains the *Exit* option.

Alternatively, you can start another command via the pull-down menus toolbars or palettes to exit the dynamic zoom process (this works, as long as you do not call a transparent command from the menu or palette).

While executing the DZOOM command, you can also right click with the mouse and produce the following option dialog:



Beneath leaving the dynamic zoom operation, the cursor menu (that you can open by clicking the right mouse button) also allows you to switch to dynamic pan mode and to execute a Zoom Extents.

Dynamic Zoom can be used transparently (in the middle of another command).

---

**See also:**

ZOOM command

DPAN command

### **Zoom Window**



**Command: ZOOMWIN**

**Alias ZW**

**Located:**

1. **Pull-down Menu: View – Zoom Window**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMWIN command allows a chosen portion of a drawing to be shown in the largest possible scale. The drawing portion is specified by a box. Identify opposite corner points with the cursor or enter the coordinates manually.

> ZOOMWIN  
First corner: **P1**  
Second corner: **P2**

---

**See also:**

ZOOM command

## Zoom Back



**Command: ZOOMBACK**

**Alias ZX**

**Located:**

1. **Pull-down Menu: View – Zoom Back**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The command ZOOMBACK allows you to undo the last ZOOM command and recreates the prior drawing portion. The deleted scale is saved as the last used scale and may now be called again.

Using this command allows the user to switch quickly between two definite portions in order to view an object as a whole, or to see its details more clearly.

It is still possible to recreate the last scale if other operations have been performed.

---

**See also:**

ZOOM command

Zoom to Fit



**Command: ZOOMFIT**

**Alias ZE**

**Located:**

1. **Pull-down Menu: View – Zoom Fit**
2. **Palette: Zoom**
3. **Or Enter Command or Alias at the Command Line**

The command ZOOMFIT allows the user to show the drawing with all its elements as large as possible on the viewing area.

The user should remember that points of deleted or modified elements that may still be saved in the drawing can influence the view resulting from this command. These points will be completely deleted when a drawing is regenerated. Thus, before zooming to the largest possible drawing size, the user should consider doing a REGEN.

---

**See also:**

ZOOM command

Zoom Factor

**Command: ZOOMFAC**

**Alias ZF**

**Located:**

1. **Pull-down Menu: View – Zoom Factor also**  
**View – Zoom In/Out/Pan Direct – Zoom 4.00 and Zoom 0.25**
2. **Or Enter Command or Alias at the Command Line**

The command ZOOMFAC allows the user to zoom with the specified scale factor. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

**Zoom In - Nearer to Objects**



**Command: ZOOMIN**

**Alias ZN**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom In**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMIN function enlarges the viewing window contents with each successive use by a factor determined by a given value. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

Zoom Factor

**Command: ZOOMFAC**

**Alias ZF**

**Located:**

1. **Pull-down Menu: View – Zoom Factor also**  
**View – Zoom In/Out/Pan Direct – Zoom 4.00 and Zoom 0.25**

## 2. Or Enter Command or Alias at the Command Line

The command ZOOMFAC allows the user to zoom with the specified scale factor. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

### **Zoom Out - Away from Objects**



**Command: ZOOMOUT**

**Alias ZT**

#### **Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom Out**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMOUT function is the opposite of the zoom-in function. It reduces the size of viewing window contents with each successive use by a factor determined by a given value. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

Zoom Factor

**Command: ZOOMFAC**

**Alias ZF**

#### **Located:**

1. **Pull-down Menu: View – Zoom Factor also**  
**View – Zoom In/Out/Pan Direct – Zoom 4.00 and Zoom 0.25**
2. **Or Enter Command or Alias at the Command Line**

The command ZOOMFAC allows the user to zoom with the specified scale factor. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

## Pan Up



**Command: PANUP**

**Alias PU**

### **Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Up**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately shift upwards on the drawing. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR).

---

### **See also:**

PAN command

DZOOM command

## **Pan Down**



**Command: PANDOWN**

**Alias PD**

### **Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved down. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

### **See also:**

PAN command

DZOOM command

Pan to the left



**Command: PANLEFT**

## Alias PL

### Located:

1. **Pull-down Menu:** View – Zoom In/Out/Pan Direct – Pan Left
2. **Left SideBar**
3. **Palette:** Zoom
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved to the left. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

### See also:

PAN command

DZOOM command

### Pan to the Right



**Command:** PANRIGHT

## Alias PR

### Located:

1. **Pull-down Menu:** View – Zoom In/Out/Pan Direct – Pan Down
2. **Left SideBar**
3. **Palette:** Zoom
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved to the right. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

### See also:

PAN command

DZOOM command

## Command ZOOMPAGE

**Command:** ZOOMPAGE

## Alias ZP

### Located:

1. **Pull-down Menu:** View – Zoom In/Out/Pan Direct – Zoom Page Limits



## 2. Or Enter Command or Alias at the Command Line

The **ZOOMPAGE** command zooms out to the page size defined for your printer (or plotter).

### Set Plan View to Construction Plane



**Command: PLANVIEW**

**Alias PV**

#### **Located:**

1. **Pull-down Menu: Tools – Set Plan View to Construction Plane**
2. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
3. **Left SideBar**
4. **Palette: View**
5. **Or Enter Command or Alias at the Command Line**

The PLANVIEW command sets the view planar to the current User Coordinate System (UCS). Normally, this command is used to return from an isometric 3-dimensional view of the drawing, to the standard (from above) view of the drawing.

---

#### **See also:**

PLAN

3DVIEW

UCS

Hidden Line Drawing...



**Command: HIDE**

**Alias HD**

#### **Located:**

1. **Pull-down Menu: View – Hidden Lines – Hidden Line Drawing**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**

The HIDE command allows you to create a separate drawing that contains a representation of the current view of the drawing with hidden lines removed.

Alternatively, those lines (or edges) that are computed as hidden from the current viewpoint may be placed on a special layer named HIDE LAYER in the target drawing. When working with the drawing containing the hidden line representation, you can turn this layer on or off at will, or remove all the lines on that special layer - except those you want to maintain intentionally for demonstration purposes.

Sometimes it is useful not to suppress the hidden lines, but to display all of these lines, for instance, to

demonstrate certain constraints. In such a case we recommend setting a certain linetype (for example, a dotted linetype) for the layer HIDE\_LAYER in the target drawing.

---

**See also:**

FHIDE

### **Fast Hide**



**Command: FHIDE**

**Alias FHD**

#### **Located:**

1. **Pull-down Menu: View – Hidden Lines – Fast Hide**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**

In contrast to the HIDE command, which creates a new drawing, the command FHIDE displays a hidden line view in the current drawing.

This representation is maintained until the drawing window is refreshed by a REDRAW, REGEN, ZOOM, or PAN command. This hide command offers you a visual impression of the 3D model from the current viewpoint.

Note that you can continue to draw in this display mode, even utilizing the object snap functions.

The command FHIDE is based on the OpenGL system from Microsoft that allows high performance hidden line removal and rendering.

---

**See also:**

HIDE

Command DVIEW

DRAWORDER

### **3D View Control...**



**Command: 3DVIEW**

**Alias 3V**

#### **Located:**

1. **Pull-down Menu: View – 3D View**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the View menu allows you to define the view parameters. .



The program uses as its standard viewing angle, the top view (also called Plan View). At this angle the x-y-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views.

Along with the ability to view the drawing from any vantage point, it is also possible to rotate the view in the X – Y plane. You may want to do this, for instance, when orienting your drawing for printing.

---

**See also:**

SETVIEWDIR command

View Manager: Saving / Retrieving Views

Automated Rendering (AutoRender)

**Command: ARENDER**

**Alias AE**

**Located:**

1. **Pull-down Menu: View – Automated Rendering**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

Rendering allows you to automatically create an image of your 3D drawing that illustrates the model or plan very realistically.

ARENDER provides this automatic rendering in a separate window of the program.

Light settings are automatic when using this command.

The command allows Automated rendering with fast zoom and rotation of rendered object. Use the left mouse button to rotate on the x/y plane and the right mouse button to zoom on the z plane.

Use the Auto-Rotate option from the system menu to let the model rotate automatically. Use the left mouse button (drag with button hold down and then release) to determine rotation speed.

---

**See also:**

RENDER

## Render



**Command:** RENDER  
**Alias** REN

### Located:

1. **Pull-down Menu:** View – Render – Render
2. **Palette:** View
3. **Or Enter Command or Alias at the Command Line**

Rendering allows you to create an image of your 3D drawing that illustrates very realistically the model or plan. To increase the realism, the rendering may perform smooth shading on the objects that meet at a specific angle and evaluate the lights set in the drawing.

Using rendering, you visualize and present your 3D geometry as it would appear, after construction.

To render the current viewport of a drawing, call the command RENDER. The command is immediately performed after you have specified where to display or output the rendered view:

| Screen | File | Clipboard |

The rendering is based on the preferences set with the RCONFIG command and evaluates the light sources found in the model. The command LIGHT allows you to place lights in the drawing and define their type and settings. The command LIGHTEDIT is used to modify the settings of light sources.

---

### See also:

RCONFIG

## Render Settings



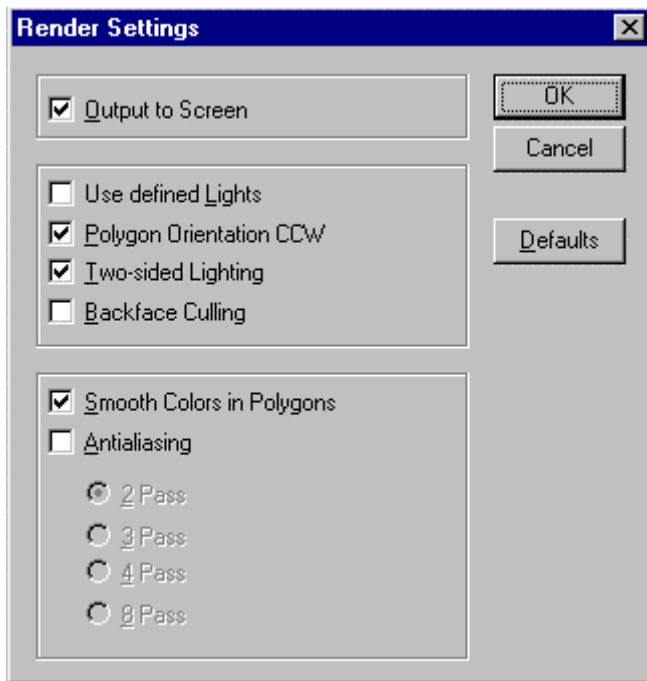
**Command:** RCONFIG

**Alias** RC

### Located:

1. **Pull-down Menu:** View – Render – Render Parameters
2. **View Palette**
3. **Or Enter Command or Alias at the Command Line**

The command RCONFIG displays a dialog box to view and set general preferences for subsequent rendering.



The button **Defaults** resets the preferences to default values (shown in the illustration above). This is convenient, because a number of settings can influence the performance of render computations dramatically.

The option **Output to Screen** controls whether the rendered polygons are output on the screen (to the front buffer) during computations or are written into a display buffer before presenting the rendered image. If this check mark is not set, the rendered image is not displayed before the entire rendered image is computed.

The option **Use Defined Lights** allows you to activate or deactivate the use of light sources. If disabled, the program generates - in computer graphic terms - a simple **shading** of the 3D model, otherwise the inserted lights are utilized during the rendering.

Three other options allow users who are familiar with computer graphics and advanced rendering techniques to save time when careful drawing methods have been used during the creation of a 3D model: **Polygon Orientation CCW**, **Two-sided Lighting**, and **Backface Culling**. It is recommended to use the default values if you are not familiar with these terms.

The option **Smooth Colors in Polygons** specifies whether upon each individual polygon surfaces smoothing colors are computed or not. Applying an adequate light environment allows smooth coloring on individual surfaces.

The option **Anti-Aliasing** allows you to control the process of smoothing jagged edges caused by aliasing. Aliasing is a term in computer graphics. It describes certain display effects like the staircase effect seen on diagonal lines displayed in a drawing. Anti-Aliasing can reduce those artifacts.

Turn anti-aliasing on for high-quality rendered images. To improve the rendering significantly you can choose between **2, 3, 4 or 8 passes** with the anti-aliasing algorithm. Because multiple passes will increase the rendering time dramatically, anti-aliasing should be applied only for high-quality image output (usually when rendering to a file).

By default the anti-aliasing option is disabled. It produces a faster output for viewing a rendered image on the graphics display, and you do not normally notice the elimination of artifacts on the computer screen.

---

#### See also:

RENDER

## Render Lights: Editing Light Sources



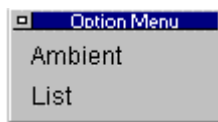
Command: **LIGHTEDIT**

Alias **LE**

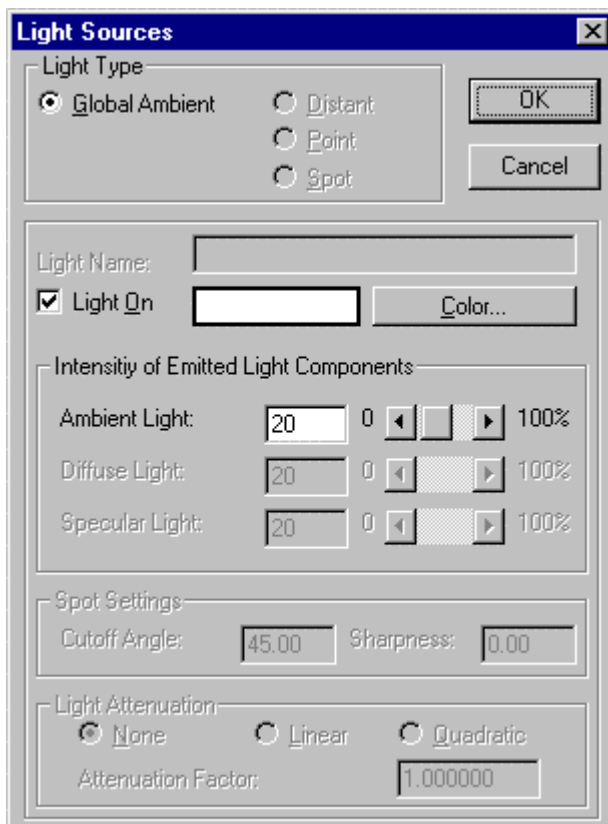
### Located:

1. Pull-down Menu: **View – Render – Edit Light Parameters**
2. Pull-down Menu: **View – Render – Ambient Light (LIGHTEDIT-A)**
3. Palette: **View**
4. Or Enter Command or Alias at the Command Line

When you issue the command, you are presented with the following dialog. Here, you specify whether you wish to edit ambient light settings, or else a previously defined (other) light source.



Below, we show the edit dialog, where we see that we can select/deselect global ambient light, where we can specify light color and intensity. In this example, we are simply editing ambient (natural) light characteristics. With other specifically set light sources, many different lighting effects become possible, which can be configured in this dialog.



---

See also:

3DVIEW

Render Lights: Inserting Light Sources



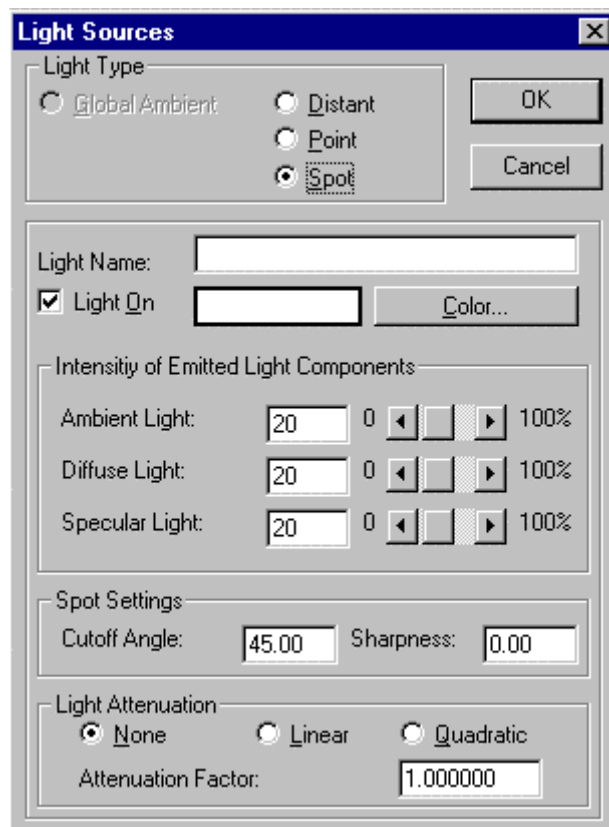
**Command: LIGHT**

**Alias LIT**

**Located:**

1. **Pull-down Menu: View – Render – Insert Light**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**

When you give the LIGHT command to create a new light source for your rendering, you will be presented with the following dialog:



For any new light source, the first step is to indicate its type. Selecting the Distant, Point or Spot button will signal that choice and also activate the light edit features on this dialog.

Next, we give the light a name, so that it can be called for later editing, as needed. You may need to switch the light on or off during the course of your work and there has been a selection button provided here for that purpose. By pressing on the Color button, you can select the color of light that you want. This color appears in the box to the left of the color button.

Below, you will find further advanced light editing controls. Unless you are an expert and familiar with

all of these features, you would be best advised to leave these at default values.

**Notes:**

1. Ambient light is given any time, but linear, point and spotlight sources must explicitly placed in the 3D model.
2. Note that although you can insert as many light sources you can have **only eight light sources turned on**.
3. Lights sources are represented in the drawing as complex objects (part/block objects)
4. A light insertion will always be placed on a layer with the name RENDER\_LIGHTS.

---

**See also:**

3DVIEW

LIGHTEDIT

Render Lights: Editing Light Sources



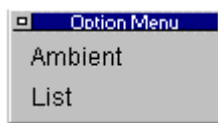
**Command: LIGHTEDIT**

**Alias LE**

**Located:**

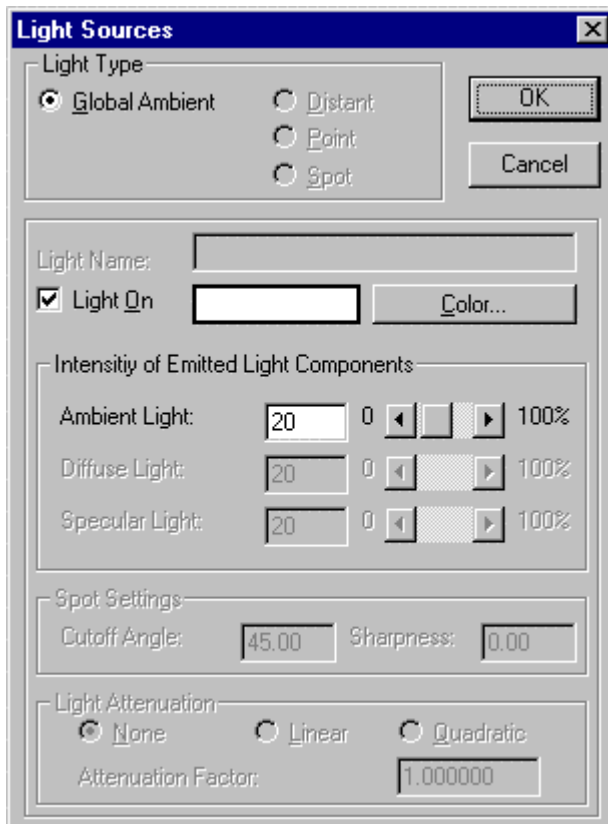
1. **Pull-down Menu: View – Render – Edit Light Parameters**
2. **Pull-down Menu: View – Render – Ambient Light (LIGHTEDIT-A)**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

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---

**See also:**

3DVIEW

Command DVIEW

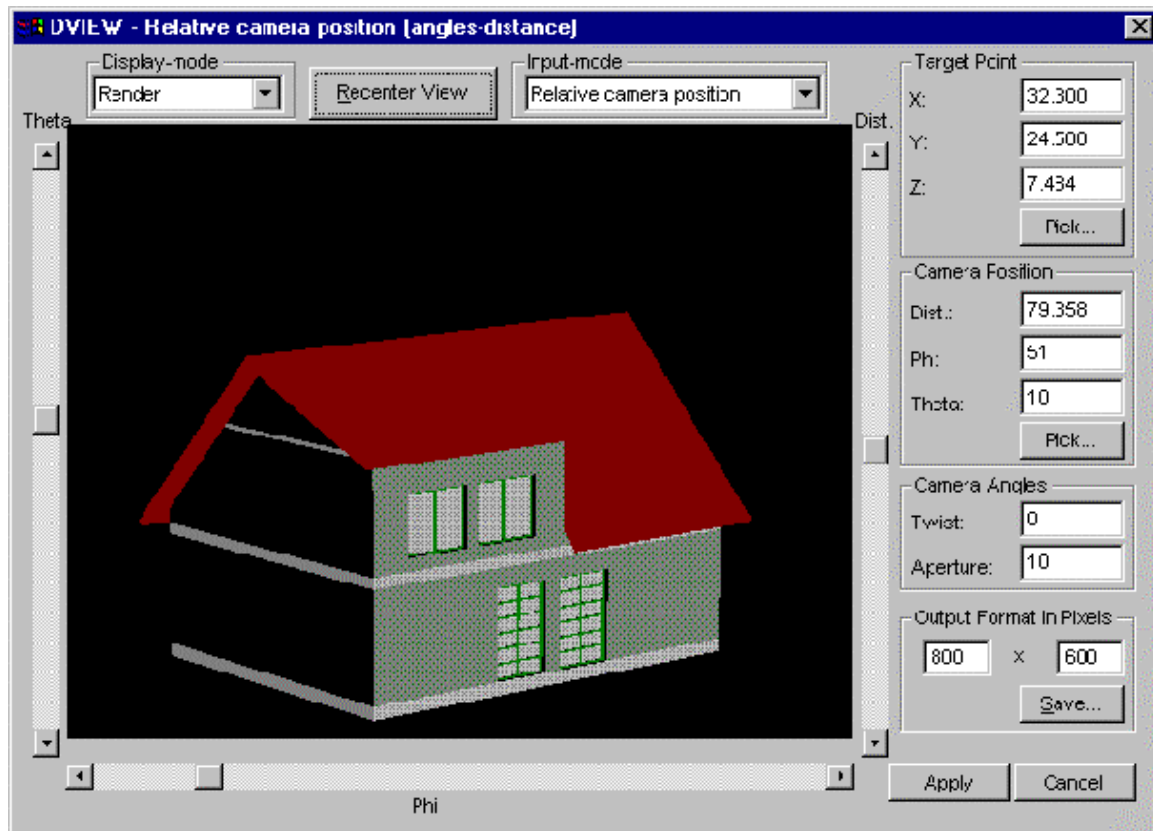
**Command: DVIEW**

**Alias DV**

**Located:**

1. **Pull-down Menu: View – Dynamic View**
2. **Or Enter Command or Alias at the Command Line**

The **DVIEW** command creates a dynamic 3D-perspective view using different display modes. The command shows a dialog with a preview window. It is possible to save this view as a separate bitmap (BMP) file as well as transferring this view to the drawing.



The functionality of DVIEW in detail:

#### Target Point

Target Point	
X:	31.272
Y:	25.228
Z:	3.566
Pick...	

This is the point the camera is aimed at. The x,y,z coordinates can be set either by entering values in the appropriate edit boxes or by picking (**pick...** button) a point in the drawing using the standard snap functions.

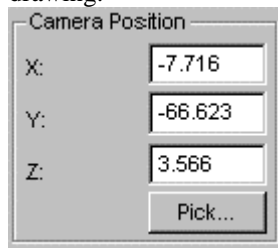
#### Camera Position

Camera Position	
Dist.:	99.783
Phi:	247
Theta:	0
Pick...	

This is the position of the camera relative to the target point, in other words, the position the drawing is viewed from. The distance (**Dist.**) as well as longitudinal (**Theta**) and latitudinal (**Phi**) settings can be set in this section.

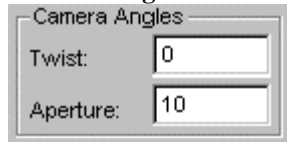
The **Theta** and **Phi** values correspond to a sphere of radius **Dist.** centered around the target point.

Again, these values can be adjusted by entering the values in the edit boxes or by picking a point in the drawing.



Alternatively, the absolute camera position can be set by selecting the **Absolute camera position** entry in the **Input** drop-down box pictured above.

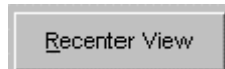
### Camera Angles



**Twist:** rotates the camera while facing the target point.

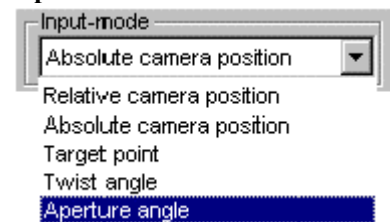
**Aperture:** changes the aperture of the camera.

### Re-center view



This button re-centers the view. Pressing this button shows a good starting point for the rotation. Note: when you first issue the DVIEW command, you may need to re-center the view to bring the drawing into view.

### Input Mode



The input mode changes the function of the sliders on the preview window. The different modes are:

**Relative camera position:** The sliders set the relative camera position values using Phi/Theta/Dist.

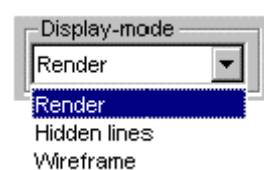
**Absolute camera position:** The sliders set the absolute camera position.

**Target point:** The sliders set the absolute target position.

**Twist angle:** A single slider changes the twist of the camera.

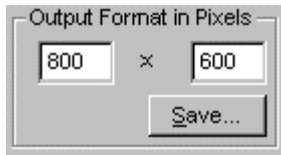
**Aperture angle:** A single slider changes the aperture of the camera.

### Display Mode



Three display modes are supported: Render, hidden line and wire frame.

### Output Format in Pixels



The 3D-view displayed can be exported as a bitmap file at the resolution entered in the edit boxes.

**Save ...** opens a standard save file dialog.



**Apply** assigns the changes to the view to the drawing coordinate system.

Please note that panning of objects (pressing the mouse in the picture and then dragging the mouse to a new location) in the preview does not result in a panning of the object in the actual drawing as only changes in the perspective view are transferred to the drawing.

## Creating and Viewing Slides

**Commands: MSLIDE**

**MSLIDE is Located:**

1. **Enter Command or Alias at the Command Line**

**VSLIDE (Alias VS),**

**VSLIDE is Located:**

2. **Pull-down Menu: View – View Slide**
3. **Or Enter Command or Alias at the Command Line**

**SLDOUT (Alias SO)**

**SLDOUT is Located:**

1. **Pull-down Menu: View – Take Snapshot (Slide)**
2. **Or Enter Command or Alias at the Command Line**

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.

You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

**To create a slide file**

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.

3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

#### **To view a slide file**

Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

## Creating and Viewing Slides

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#### **VSLIDE (Alias VS),**

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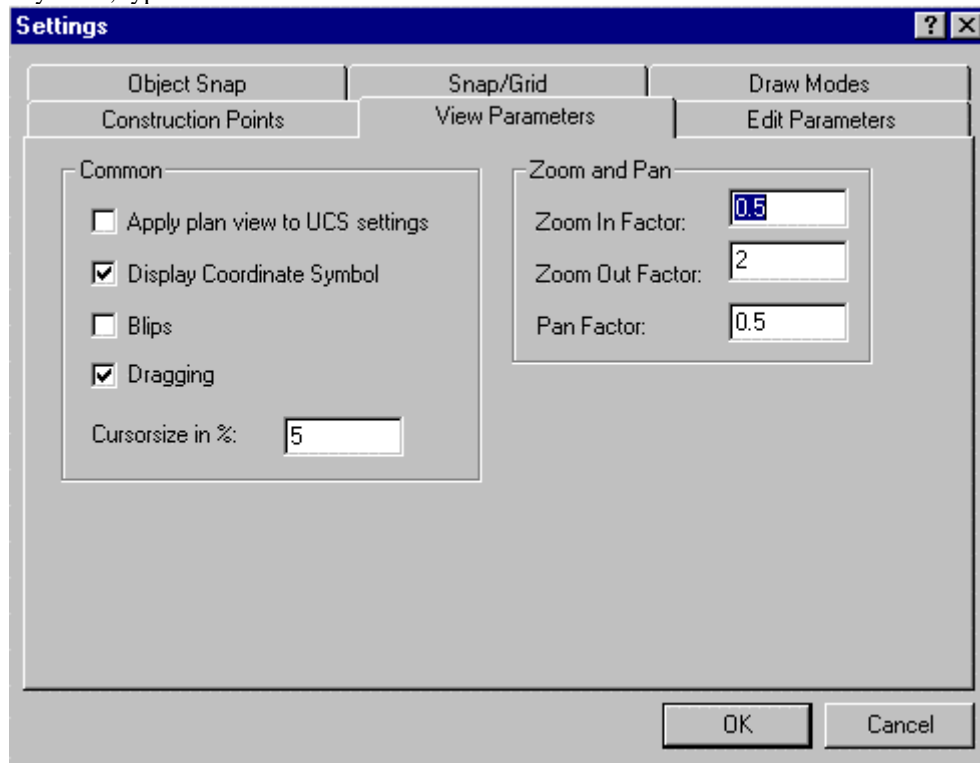
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4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

### Parameters for the Zoom and View Commands

To activate the function *View Parameters* choose the item of that name from the menu *View*. Or, using the keyboard, type the command VIEWPAR.



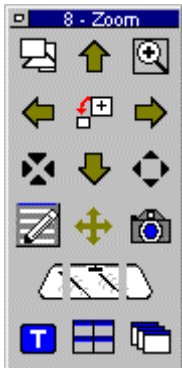
#### Zoom Factor

In the entry fields Zoom In Factor and Zoom Out Factor the user may choose the factor by that the drawing on the screen is to be enlarged or reduced. These factors affect only the visible portion of the drawing.

#### Pan Factor

In the entry field Pan Factor the user may choose the factor by that the visible portion of the drawing is moved using the command PAN. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area.

### Zoom Palette



This command will place the palette containing Zoom command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

### View Palette



This command will place the palette containing Edit command icons on the desktop, for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

### Draw

#### "Draw" Pull-down Menu



The Draw pull-down menu contains command instructions that create new drawing elements in your drawing. For a description of each command, please refer to the specific topics that follow.

## Line



**Command: LINE**

**Alias L**

### Located:

1. **Pull-down Menu: Draw - Line**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The **Line** command is used to draw lines or line segments or to connect lines to existing lines or arcs. Inputs to the command are two points; the starting point and the ending point of the line.

### Options:

#### Continuous

**Continuous** is the standard option when drawing lines, and will automatically be used when no other option is selected. It allows the user to draw line segments in sequence until the command is terminated.



The command will request a start-point for the line, and will continue to request points until it is terminated. Each subsequent point will be connected to the previous point by a line.

From Point:

To Point:

To Point:

...

### Undo

The option **Undo** can be used repeatedly to erase the last drawn line segment of a single **Continuous** line command in succession.

### Close

The option **Close** connects the last drawn point P<sub>n</sub> with the start point P<sub>1</sub> in the **Continuous** line command.

To terminate the command, press RETURN or ESC.

### Line Segments

The option **Segments** allows the creation of single line segments, as opposed to continuous lines.

**Line Segments** are useful when the line that is being drawn for future drawing construction purposes (IE a temporary line that defines a parameter to be used in drawing subsequent objects).

As with continuous lines, you determine the start- and end-point of the first segment. The program will continue to ask for start- and end-points.

In this mode the end of one line segment is not automatically used as the start point of the next segment.

To end the drawing of line segments, press RETURN.

### Append

The option **Append** allows the connection of a line to either the start or end-point of another line or arc.

Choose the line or arc that you wish to connect to. Note that the end of that object closest to the pick point will be used as the start point for the line segment.

#### Line to line

When appending a line to a line, select the line to which the appended line will connect. The point (start or end) of that line closest to the selected point on the line will be the start point for the new line.

#### Line to arc

To append a line to an arc, select the arc to which the line is to be appended. The prompt will ask for the length of the line. Either keyboard the length, or "pick" the length with the cursor. For precision, it is usually better to keyboard the length.

The appended line will be connected to the end point of the arc as a tangent.

To end the command press RETURN or ESC.

### Circle



**Command: CIRCLE**

### Alias CI

#### Located:

1. **Pull-down Menu: Draw - Circle**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

There are several ways to describe a circle, geometrically. The program allows the user to choose from among these ways, and draw the circle in the most convenient manner.

A circle can be drawn by choosing one of the following options:

1. Center of the circle and radius or diameter;
2. Two Points on the circumference (that define the diameter and center point)
3. Three Points on the circumference
4. Two Tangents

These options will appear on the *Options Panel* when the command circle is active.

## Options

### Center

Since the *Center Point* option is the most often chosen, it has been selected as the default option and is activated automatically if no other option is chosen. The first prompt for this option will be the location of the center point; the second will ask for the radius. If the circle being drawn is preceded by another circle, the last radius will appear at the prompt as the default radius value. To verify the default, just press RETURN, or to change the option, enter or "pick" a new radius.

If it is more convenient to enter the diameter than the radius, the user may choose that option from the options panel.

### 2P

If the circle is to be tangent to two other lines, this option is handy.

The prompt will ask for two points in succession. Again, the points may be picked using the object snap options, or may be entered from the keyboard. The mid-point between the picked points will be the circle center and the distance between the points, the circle diameter.

### 3P

This option allows the user to draw a circle by choosing three points on the circumference of the circle. It is convenient to use when the circle must intersect three tangent lines, or when drawing the circle within a polygon.

The prompt will ask for three points successively. The user may point to these points on the screen, perhaps using the option **object snap**, or enter the coordinates through the keyboard.

### Radius

This geometry is particularly useful when the radius of the circle is a known, and the center point is undefined, or when the radius follows from geometry already established.

You are first asked to pick two points that define the circle radius length. These options are for entering the radius:

- Verify the default radius value (the radius of the last circle drawn) by pressing RETURN;
- Enter a new value for the radius manually; or
- Pick the radius on the screen. This option is especially useful when the radius follows from the geometry and only has to be copied.

Next, you are asked to locate the circle center point.

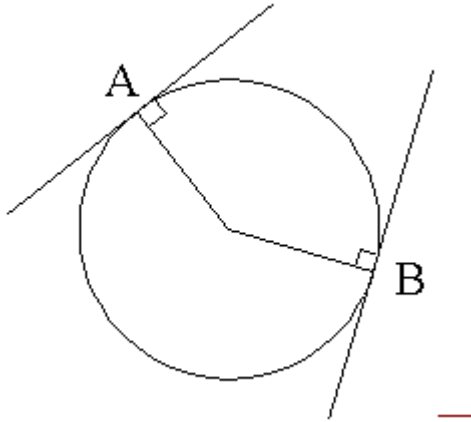
The circle is drawn once the center has been located, however the command assumes that you may wish to draw another circle with the same radius and it asks for another center point. This continues until the command is terminated by pressing Esc, Return, or selecting another command.

### Diameter

The use and the implementation of this option follows exactly the use and implementation of the option **Radius** except that instead of choosing the radius of the circle, the user must specify the diameter.

## Tangent

This option is frequently useful, as it will draw a circle of given radius or diameter **tangent to two lines**.



The first prompt will ask for the objects that the circle is to be tangent to. These objects must exist in the drawing before this command is issued. Pick both objects on the screen. In the example shown, we pick the two lines.

After choosing the second object, enter the radius of the circle to be drawn. The radius of the last circle drawn is offered as default by the program. Verify this choice by pressing RETURN or enter a new value manually, or by "picking" it on the screen.

The circle will be placed so that it touches, tangentially, the two chosen elements with its circumference. In other words, the line from the circle center to the intersections is at right angles to the objects. This is illustrated in the example above, where points A and B are the points of calculated tangency.

If the chosen radius is smaller than the distance of the two objects to each other, the circle will be moved in the direction of the imaginary point of intersection, so that the defined radius will touch the extension of the elements. Note: the original objects are not extended. The extensions are only calculated to determine circle location.

If a tangential intersection is impossible (parallel lines, two concentric circles or arcs as chosen objects, etc.), the program gives the following message: "Circle does not exist!"

**Note:** The "smoothness" with which the circle is represented onscreen is controlled by the variable CIRCLES. By issuing the SETTINGS command, users can go to the Draw Modes tab and enter a new value (between 8 and 1024) in the Arc Segments box that controls smoothness. A low value will result on a rough arc or circle, while a higher value will produce a smooth arc or circle.

## Arc



**Command: ARC**

**Alias A**

### Located:

1. **Pull-down Menu: Draw - Arc**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

There are several ways, geometrically, to describe an arc, just as with a circle. The program allows the user to choose from among these methods, and draw the arc in the most convenient manner.

The following options for drawing an arc are available:

1. Three Points (starting point, second point, end point);
2. Two points and radius;
3. Two points and angle of opening;
4. Two points and center point;
5. Two points and tangents through starting point;
6. Center point, starting point and angle; or,
7. Center point and two angles.

Variations of the options of this command may combine some of the basic geometry inherent to each method.

**All arcs will be measured to the left, starting from the positive x-axis.** In other words, you pick points on the arc in an anti-clockwise direction. It is important to recognize this directional convention, as failing to do so will produce an arc that is complementary to the one intended (the one that if added to the intended arc, would form a circle). To "pick" points for the construction of arcs, the user should always use OSNAP options.

### Options:

The options are displayed in the *Options Panel* when the command is active.

#### 3Point

Since the method most used for drawing arcs is to select three points on the arc, that is the option that has been chosen as the "standard", or default, option. It will be used if no other option is chosen.

The arc to be drawn is defined by three points. At the prompt, enter the starting point, a second point on the arc's circumference and the end point of the arc. Remember the directional convention; the arc is described anti-clockwise to the left starting with the positive X-axis.

Starting point: P1  
Second point: P2  
End point: P3

This option is also available in other options of the drawing command ARC.

#### SEnd (Starting point - End point - Radius/Angle)

Using the option **SEnd**, the user may create arcs with only the starting point and end point defined. The "arc" of the element is defined by entering or "picking" a third point on the circumference, or by specifying an angle.

The basic variation of this option simply follows the prompts, entering the starting point and end point and the radius of the arc by "picking" or by entering through the keyboard.

Starting point: P1  
End point: P2  
Radius: Value

When the user is "picking" the radius of the arc, it must be remembered that the distance of the center point to the **x-axis** of the end point of the arc is important, not its distance to the y-axis.

It is also possible to enter a radius, an angle or a direction from the starting point instead of the starting point and the end point. To do that, choose the option **Angle** or **Starting point**.

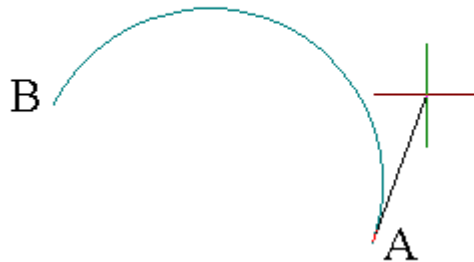
#### Option Angle

With this option the arc is defined by the included angle of the arc (angle between the starting point, the center point of the arc and its end point), the starting point, and the end point. The prompts will request a value with the words **Included angle**:

Enter that value either through the keyboard, or by "picking" a value. When picking a value, remember that the x-axis is the point of reference, not the last point entered.

#### Direction

The arc is defined by a tangent that touches the arc at its starting point, next to its starting point and at its end point.



In this example, we have defined the start and end points of the arc (A and B) and have selected the direction option. This results in a line being represented from the arc's start point to the current cursor position. The arc radius varies with the cursor location such that the arc remains tangential to that line representation at the start point.

The program will ask the user to pick or enter the coordinates of a point that fixes the cursor location and thereby defines the arc radius.

#### SCenter (Starting point - End point - Center point)

This is a variation of the option **SEnd**, in which the arc is drawn by determining its starting point, end point and center point.

Choose the option **Center point** from the options panel after having entered the starting point. Enter the coordinates of the center point manually or "pick" them with the cursor. The arc is defined when the end point is entered.

It is also possible to end the arc by entering an angle manually or "picking" it instead of the end point. The program expects a closed angle and defines it as the angle between the starting point, the center point and the end point of the arc.

#### CStart

The option **CStart** draws the arc starting from the center point.

After choosing the option **CStart** from the options panel, define the center point. Then enter the starting point and the end point of the arc (it is possible to enter a closed angle instead of the end point).

Center point: P1  
Starting point: P2  
End point: Angle

#### CRadius

**CRadius** also starts an arc by defining the center point. After that, determine the radius as well as the starting angle and the ending angle of the arc to be drawn. A rubberband line that starts in the center of the arc helps the user to determine the points to enter for the angle.

Center point:  $P1$   
Radius:  $r$   
Starting angle:  $P2$   
Ending angle:  $P3$

### Append

It is possible to append, or connect, arcs to lines or other arcs with the option **Append**. The arc is determined by two points, the starting point being the connection point, and the end point of the new arc.

First choose a line or an arc to which the new arc should be appended. Then determine the end point of the arc to be drawn.

Choose the line or arc:  $P1$   
End point:  $P2$

**Note:** The "smoothness" with which the arc is represented onscreen is controlled by the variable CIRCLESRES. By issuing the SETTINGS command, users can go to the Draw Modes tab and enter a new value (between 8 and 1024) in the Arc Segments box that controls smoothness. A low value will result on a rough arc or circle, while a higher value will produce a smooth arc or circle.

## Polygon (N-gon)

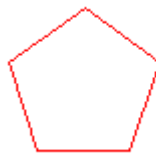


**Command: NGON**

**Alias NG**

### Located:

1. **Pull-down Menu: Draw - Ngon**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**



The command **N-gon** is used to draw regular polygons. It is possible to draw polygons with up to 1024 sides, a value that should lie above any practical need.

When using this command, three options, based on different principles of construction, are available:

1. Definition of the length of sides of the polygon;
2. Definition of a center point and a radius of a circle outside ; or
3. Definition of a center point and a radius of a circle inside.

Select the command for drawing a polygon and choose one of the options from the options panel:

**Sides**  
**Outside**  
**Inside**

## Options:

### Length of the sides

**Sides** is the standard option for drawing regular polygons. It will be used if no other option is chosen.

The regular polygon is defined by the length and placement of one side.

To draw a regular polygon, enter the number of its sides first. To do that, enter a value between 3 and 1024 and verify this entry by pressing RETURN.

If there have been polygons drawn previously, then a number will appear in the prompt. That number is the number of sides of the polygon drawn last. The user may elect to use that number by simply pressing RETURN.

After specifying the number of sides, determine the starting point and the end point of the first side either by entering the coordinates or by picking it with the cursor. Remember that the polygon will be drawn counterclockwise, starting from the first side.

```
> NGON  
Number of sides <4>: 6  
First corner point of the side: P1  
Distance or second point of the side: P2
```

### Circumscribing the Polygon

#### Outside

The option **Outside** defines a polygon by determining a center point and the distance to its corner points. This distance is the radius of a circle that contains the polygon, a point on whose circumference all corner points of the polygon lie.

After entering the number of sides, choose the option **Outside** from the Options Panel.

Determine the center point of the polygon by entering its coordinates or picking it. Next, determine the distance to its corner points. The polygon will be drawn counterclockwise, starting from the first corner point.

```
> NGON  
Number of sides <4>: 6  
Center point of the polygon: P1  
Distance to its corners: P2
```

#### Inside

**Inside** allows the polygon to be defined by the circle whose circumference touches each side of the polygon at its mid-point.

Choose the option **Inside**, then enter the number of sides. Next determine the center point of the polygon by entering its coordinates or "picking" it; then determine the radius of the circle. The polygon will be drawn counterclockwise, starting from the thus defined side.

```
> NGON  
Number of sides <4>: 6  
Center point of the polygon: P1  
Distance to its corner points: P2
```

### Rectangle



**Command: RECTANGLE**

**Alias RT**

#### Located:

1. **Pull-down Menu: Draw - Rectangle**

**2. Palette: Draw**

**3. Or Enter Command or Alias at the Command Line**

The command RECTANGLE allows you to construct rectangles of any size and side ratios.

To draw a rectangle, enter or pick two opposite corners.

When the command is active, the option *Width* is available in the Options Panel. The option **Width** allows you to specify a value for the line width applied on the rectangle creation. To use the last entered value (that will be offered by the prompt) just press RETURN.

**Examples**

```
> RECTANGLE
First point of the rectangle: P1
Opposite corner point: P2

> RECTANGLE
First point of the rectangle: Width
Width <0.05>: 1.00
First point of the rectangle: P1
Opposite corner point: P2
```

Trapezoid

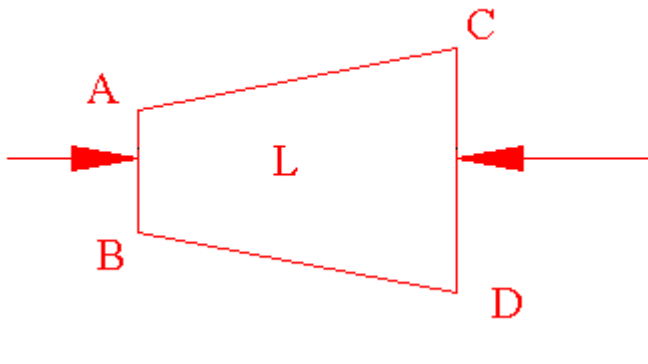


**Command: TRAPEZOID**

**Alias TZ**

**Located:**

- 1. Pull-down Menu: Draw - Trapezoid**
- 2. Palette: Draw**
- 3. Or Enter Command or Alias at the Command Line**



With the command TRAPEZOID it is possible to draw a single conical element (trapezoid) on the active layer.

Start this function by choosing the like-named option in the menu *Draw*, its palette symbol, or through entering the command TRAPEZOID.

After entering the point and insertion angle, the cone is defined by its length and the diameter of its two ends. To do that, determine the needed points successively through pointing or entering the values or coordinates manually.



The cone can also be drawn using a **Ratio** of tapering. To do that, press RETURN after the question of the diameter of the second side and enter a tapering ratio at the asking.

Command input sequence:

> TRAPEZOID

Insertion point: **Point A**

Insertion angle <0>: **Angle L will be oriented to**

Diameter of first side: **Dimension A-B**

Length: **Dimension L**

Diameter of second side: **Dimension C-D**

## Ellipse



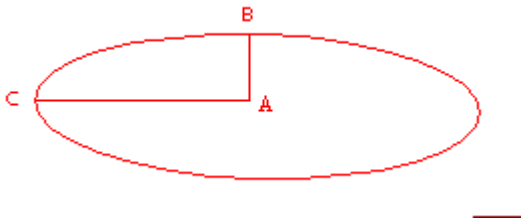
**Command: ELLIPSE**

**Alias ELL**

### Located:

1. **Pull-down Menu: Draw - Ellipse**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The command ELLIPSE allows you to draw a full ellipse.



The default is to identify *Center* point "A" of the ellipse first and then specify Point "B" (the first axis of the ellipse) Note that in this step, however, the orientation of the ellipse is established by either determining the location for "B" by picking it onscreen, or through distance/angle input as indicated below. Lastly, the secondary ellipse axis (A - C - at a 900 orientation from A - B) is input, either by picking point C. or entering the A-C dimension.

> ELLIPSE

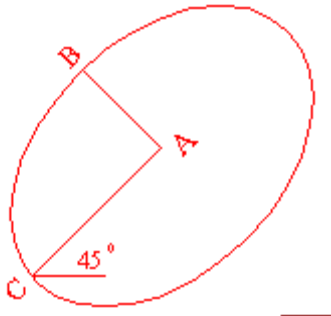
Center point: **Point A**

Primary radius: **Dimension A-B**

Secondary radius: **Dimension A-C**

When you know the orientation of the A-B and A-C axes of the ellipse, you can specify this orientation when you input the primary A-B axis, by specifying the distance and angle of the A-B axis in relative

terms, using the format (@distance<angle).



> ELLIPSE  
Center point:  
Primary radius: @3<45  
Secondary radius: 2

In this example, pictured above, the A-B alignment (3 units) has been rotated 45 degrees to the left, as has the C-D alignment.

Note: The ellipse will be created as a closed polyline with straight-line segments.

## Chain



**Command: CHAIN**

**Alias CN**

### Located:

1. **Pull-down Menu: Draw - Chain**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The Chain command is a combination of the commands Line and Polyline.

The Chain command makes it possible to draw continuous lines in a manner similar to the command LINE using the option *Continuous*. The procedure is the same as drawing continuous lines.

But there are some important differences in drawing lines, polylines, and chains:

- Firstly, when you draw a continuous line, each different segment is a separate element. That way, each element can be chosen separately, using the object selection function.
- With a chain, in contrast, the resulting entity is a single entity, either for editing or for other purposes.
- Second, for a chain, it is possible to choose the line width. This is similar to the command Polyline. The last width value entered for a polyline will be offered as the default width for the next chain, and
- Finally, (differently from a Polyline) it is only possible to choose a single width of line for the entire line when drawing a Chain.

A chain is drawn by describing the points that will be the starting and end point of each chain segment. The end point for each segment will serve as the starting point for the next segment. A single width of the line must also be specified.

```
> CHAIN
First point: P1
Next point: P2
Next point: P3
...
```

You can terminate the Chain command by pressing RETURN or ESC.

### Options:

The following options will be available in the options panel:

#### Width

The option *Width* allows the user to define the width of the entire line. **The width must be specified before entering the first point.** Picking or entering the first point is understood as a verification of the required value.

The previous entered value from either a Chain or Polyline command will be offered as the default width value for the new chain. Verify this predetermined value by pressing RETURN or enter a new value for the width.

#### Undo

The option *Undo* lets the user reverse the last step taken. In other words, the last-drawn chain segment will be erased. The end-point of the previous segment is then used as a starting point of the next segment. Undo's can be repeated as long as segments exist.

#### Close

The option *Close* allows the user to close the chain to the start point. The two end points of the chain will be connected to each other and the command will be terminated.

#### Polyline



Command: POLYLINE

Alias PO

#### Located:

1. Pull-down Menu: Draw – Polyline
2. Left Side Bar
3. Palette: Draw
4. Or Enter Command or Alias at the Command Line

**Note:** For boundary polylines see Command PCONTOUR.



Polylines are drawing elements that consist of connected lines or connected arc segments. Different segments can be of different widths. The width of a polyline segment can also be defined differently for the starting and end-point.

A polyline is defined as a single element and treated as such for further operations (like Edit, or Undo), in contrast to continuous lines drawn with the command Line.

For Polyline Type control, please refer to the variable PLINETYPE.

Polylines will always be drawn with solid fill.

When you give the POLYLINE command, several options appear on the Options Bar or Menu::

- Arc
- Width
- Halfwidth
- Back
- Closing
- Append

Selecting Arc will make the next segment an arc,

Width will specify the width if the next segment, Tapered, if the initial (default) width is adjusted in the end width entry.

Halfwidth will specify a linewidth based in a half linewidth measurement. Like Width, the begin and end widths of each segment can vary, giving a tapered segment width.

Regardless of whether they are arcs or lines, polyline segments are always drawn by defining their starting and end as well as width.

The program expects that the user will want to draw more than one segment when drawing a polyline. After drawing one segment the user will be asked to name another end point for a new polyline segment. The starting point will be the end point of the segment last drawn.

The user can end the drawing of the **Polyline** by pressing RETURN or ESC.

- First point: **P1**
- Next point: **P2**
- Next point: **P3**
- ...

## Options

### Width / Halfwidth

The options **Width** and **Halfwidth** are available from the Options Bar to assist in defining the width of a polyline segment. **Width** defines the entire width of a polyline segment, whereas **Halfwidth** defines the width as the distance of an imaginary centerline to one side of the polyline.

Verify the last entered value by pressing RETURN or enter a new width manually or by "picking" it.

The following rules are in effect when using **both** options:

If the user chooses the option **Width** or **Halfwidth** before determining the starting point for the first polyline segment, then the chosen width is valid for the entire length of the segment.

If the user chooses the option **Width** or **Halfwidth** after entering the starting point for the first polyline segment, it is possible to enter different values for the width at the starting point and end point.

The last chosen width will be used for all following polyline segments on its entire length. To determine another width, choose the option **Width** or **Halfwidth** again.

### Arc

**Polylines** has an option for drawing arcs, and a width may be correlated to the arced polyline. The method of drawing arcs is identical to drawing elements using the command ARCS.

To draw polyline arcs with a special line width, follow these steps:

- Determine the first point of the arc. This can also be the end point of the previous polyline segment when drawing a polyline;
- Call up the chosen option, **Width** or **Halfwidth**, and enter the parameters for the starting width and ending width.
- Determine the other parameters for drawing arcs using the same procedure as drawing "regular" arcs.

## Append

**Append** allows the user to connect a polyline with another line or an arc.

Choose the element (line or arc) to which the polyline still to be drawn is to be appended. Call the options **Width** or **Halfwidth**, if required, and determine the parameters for the starting width and the ending width.

## Polyline to Line

To choose an object, select an already existing line. Remember that the polyline will be connected to the end point of the line that is closest to the point selected when choosing the element.

To append a polyline, the user must now determine the end point of the line to be connected. This point will be appended to the end point of the chosen existing line.

## Polyline to Arc

To append a polyline to an existing arc, the user must enter a polyline length after choosing the existing arc. A polyline with the specified length will be connected tangentially to the end of the arc.

To end the polyline (appending), press RETURN.

## Undo

The option **Undo** allows the user to "undo" the last step taken. The end point of the last segment may then be used as a starting point of another segment if required. Undo's can be repeated as long as a segment exists. The last segment will be erased with each successive UNDO.

## Close

The option **Close** allows the user to draw a closed polygon. The two end points of the chain will be connected to each other and the command will be terminated.

---

## See also:

[POLY3D](#)

Chain

Create Boundary Polyline for Hatching



Command: BPOLY

Alias BP

## Located:

1. Pull-down Menu: Draw – Polyline Contour
2. Left SideBar
3. Palettes: Draw and Detail
4. Or Enter Command or Alias Directly at the Command Line

The command BPOLY creates a boundary polyline from existing objects that form an enclosed area, most often for use with the HATCH or AREA commands.

The original entities can be made up from a combination of lines, arcs, circles, and polylines.

They can be in any arrangement as long as the boundary will make up an enclosed area without looping back upon itself.

Note: The BHATCH command can be used to create its own boundary polyline, while applying a hatch pattern to an enclosed area.

---

**See also:**

Command PCONTOUR

PCONTOUR is the command line version of the BPOLY command.

### Filled Faces



**Command: 2DFACE**

**Alias 2F**

**Located:**

1. **Pull-down Menu: Draw – Filled Faces**
2. **Or Enter Command or Command Alias Directly at the Command Line**

**2D Faces** make it possible to draw solid filled planes that have straight-line borders. This is done by creating and connecting any number of three or four-cornered planes. It may be done continuously or using segments. It is also possible to connect 2D planes to already existing 2D faces.

2D Faces are always drawn by determining the corner points (three or four). The option **Close** is available to draw triangular planes so that the third entered point is connected with the starting point of the plane.

### Indication

Remember, when defining the end points of the edges that the order in which these points are entered influences the resulting shape.

The user should always specify the end points of the edges in the same direction. If the end points of the edges of a plane are determined clockwise or counterclockwise, a "butterfly" effect results.

Options

### Continuous Filled Faces

"Continuous" is the standard option for drawing filled planes; it will be used automatically if no other option is chosen.

When drawing rectangular planes, determine the end points of the edges. Remember that the first and third and the second and fourth point are connected. The edges will make up the plane.

After determining the end points of the first two edges, the user may enter additional points, and they will be used as third and fourth edge (and so on) points for another plane immediately connected to the first one. To terminate the command, press RETURN.

### Close

To draw a triangular plane, enter three end points for two of the edges successively, and choose the option **Close** from the options panel or enter the command **Close** manually.

### Filled Face Segments

The option **Segments** is used to draw a single rectangular or triangular filled plane. In contrast to the

option **Continuous**, this option will end after the entry of the points required for the plane.

### Appending Filled Faces

This option makes it possible to append newly created planes to already existing ones. It is only possible to append a plane to the edge last drawn.

After entering the option **Append**, choose a side from an existing plane to which the new plane is to be appended. Determine the corner points of appending planes.

```
> 2DFACE
Choose a side: Choice
Third point: P5
Fourth point: P6
...
```

Terminate the command by pressing RETURN.

## Rings



Command: **RING**

Alias **RI**

### Located:

1. Pull-down Menu: **Draw - Ring**
2. Palette: **Draw**
3. Or Enter Command or Alias Directly at the Command Line



A ring is a special case of a closed polyline. It is used to draw solid rings or circles. Two concentric circles are drawn, the smaller one is the inner diameter, and the larger one the outer diameter, of the ring. The space between the circles will be filled.

Rings can now be edited using any of the editing commands for polylines and will behave just like a polyline object.

The last entered values will be used as an option for the next entry when drawing rings. If the user wants to keep using the same values, just enter a new center point. This step can be repeated as often as needed, always creating identical rings.

```
> RING
Center point: P1
Center point: P2
...
```

### Options:

#### Diameter

To determine new values for the inside or outside diameter, just choose the option **Diameter** from the options panel.

With this option it is possible to set the inside and outside diameter of the rings, where new values can be input either via keyboard or by picking points on the screen.

```
> RING
```

Center point of the ring: **Diameter**  
Inside diameter <0.500>: **0.75**  
Outside diameter <1.00>: **1.25**  
Center point: **P1**  
Center point: **P2**  
...

With command repeat terminated by Esc, it is possible to draw as many identical rings as required by simply locating their successive center points.

#### **Filled Circle**

A filled circle is a special case of rings. It is a ring with the interior diameter equal to zero.

## Command FREEHAND

### **Command FREEHAND**

**Alias FH**

**Functional in: AllenCad CE Pro YES**

**AllenCad CE YES**

**Located:**

1. **Draw Menu > Markup > Freehand Sketch**
2. **Left Side Bar**
3. **Palette Draw**
4. **Enter Command or Alias Directly at the Command Line**

The FREEHAND command gives the operator the ability to simply draw 3D polylines that track cursor movement while the mouse button is depressed. In this way, it can be used for quick insertion of handwritten notes or linear polyline drawing. The resulting entities are drawn on the current layer in red to conform with redline markup convention. The resulting entities are LWPolylines with a specified display width equal to 1/100 of the screen size. The POLYEDIT command can be used to subsequently set a specified width and the PROPEDIT command used to change color, as necessary. FREEHAND is similar to the SKETCH command, except that with SKETCH you are prompted for a segment length that can provide a measure of smoothness to the resulting polyline. The command is repeating, when you use the left mouse button to begin the line and the right button to terminate it. You can use the Esc key, Cancel button or issue a different command to terminate Freehand drawing.

## Construction Points



### **Command: POINT**

**Alias POI**

**Located:**

1. **Pull-down Menu: Draw – Construction Point**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

It is possible to manually insert points of construction into a drawing using this command. Points can also be created in the drawing as the result of other commands and inputs, such as MEASURE and DIVIDE



and the importation of survey data. The primary purpose of points is to serve as a reference in the construction of other drawing elements.

The form and size of the points is determined by specifying these parameters using the menu option *Drawing Parameters...* (See: Command DRAWMODE).

### Options:



#### **Point (Individual Points of Construction)**

This option is activated automatically when calling upon this command and thus does not have to be called up separately.

Determine the position of the points to be inserted either via keyboard or via pointing on the screen.

The command remains active after the insertion of the first point, and the user can repeat this procedure as often as is required. End this function by pressing RETURN.



#### **Measure (Using Construction Points to Measure)**

**Measure** allows the user to insert points of construction on 2-D lines, polylines, circles and arcs in a defined spacing.

This divides the entity into sections of a certain size. The drawing object is not cut into different pieces but remains one complete object.

Remember that all objects that are not an exact multiple of the length of the segment will have a remainder.

First choose a drawing element and determine the spacing of the segments (in drawing units), either by entering a numerical value or by cursor picks.

The insertion of the points into lines, polylines and arcs will be done beginning at the starting point of the segment and progressing to the end point. When inserting points of construction into a circle, the starting point lies at the intersection of the radius that has its starting point at the center point of the circle and is in the direction of the positive x-axis.

**Segments** makes it possible to separate lines, polylines, circles and arcs into segments of equal length by inserting points of construction. The size of the segment is determined by the length of the distance to be dissected and the number of construction points to be inserted. The DIVIDE command is the equivalent to Measure - Segments option.

In contrast to the option **Measure**, there will not be a remainder when using the **Segments** option.

The object being segmented is not cut into different pieces but remains one complete object.

First choose a linear object, and determine the number of segments into which the object is to be segmented. The number of segments should be a whole integer between 2 and 4096.

The insertion of points of construction into lines, polylines and arcs is done starting from the beginning of the segment and progressing to its end point, without inserting a construction point at the beginning or at the end. When inserting points of construction into a circle, the starting point lies at the intersection of a radius that begins at the center of the circle and continues along the positive x-axis. The beginning point is determined to be the end closest to the point at which the object was picked.

### **Point Representation**

Points can be shown onscreen in a number of different ways, apart from the common dot representation. The SETTINGS command's Construction Point tab has a number of alternate symbols that you may set as default. Note, however, that **all** construction points in the drawing will be shown with the same symbol, regardless of which you choose.

---

**See also:**

DRAWMODE

## Commands Measure and Divide

**Command: MEASURE, DIVIDE**

**Alias MR, DI**

**Located:**

1. **Pull-down Menu: Draw – Measure / Divide**
2. **Or Enter Command or Alias Directly at the Command Line**

The commands MEASURE and DIVIDE order points or multiple blocks (parts) along a 2D-Object. This object can be an arc, a line, a circle or a 2D-Polyline. Before a block may be used with these commands, **it must exist in the drawing database before the command is given**. This may be accomplished, either through insertion or part definition. When the command is given, the block can be inserted aligned, or not aligned to the object. Both commands work in a similar fashion. While the command MEASURE asks for a distance, DIVIDE requests for the number of points with the distance between points or parts dependent upon the length of the linear entity selected.

### Command sequence for MEASURE and DIVIDE

After typing in "**measure**" or "**divide**" (or selecting it from the menu DRAW), the following request appears:

/↻> measure

Select a line/circle/2d-polyline or an arc to measure:

After selecting the entity it is highlighted and the following question appears:

Block or distance:

Here you can input either a distance directly, (by putting in a real value or selecting two points), or you can input "**Block**" (or "**B**". "**BI**" etc), to use a block. In the first case the points at each distance from the start to the endpoint of the object are inserted and so the command is completed. If you select the option "**Block**" you will be asked for the block name:

Name of block to insert:

Now type in the name of an **existing** block. You will be asked for the offset angle for the alignment of the block along the object.

Offset angle for alignment the block along the object (or ENTER for no alignment):

Here you can press return to avoid the alignment, or to put in any offset angle that is added to the tangential angle of the object at the point of insertion. Lastly, you will be asked for the distance again.

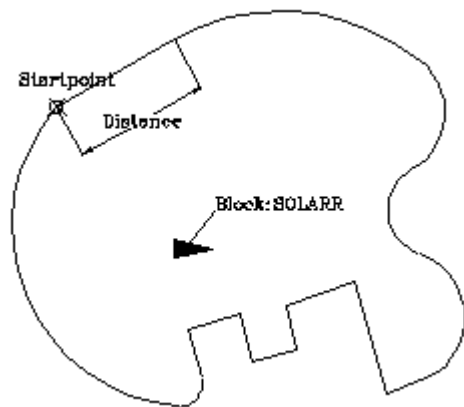
Distance for dividing the object:

You have to put in the distance here, either by giving a real value or by selecting two points. Now the Block at each position is inserted and the command will terminate.

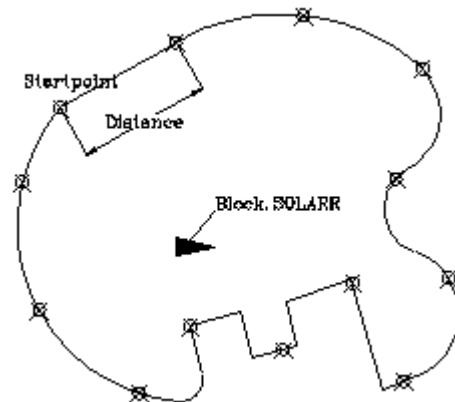
The sequence of the command DIVIDE is the same as above, except that you are asked for the number of points or blocks to insert, instead of a distance. In this case the command will compute the length between the insertion points itself.

### Example:

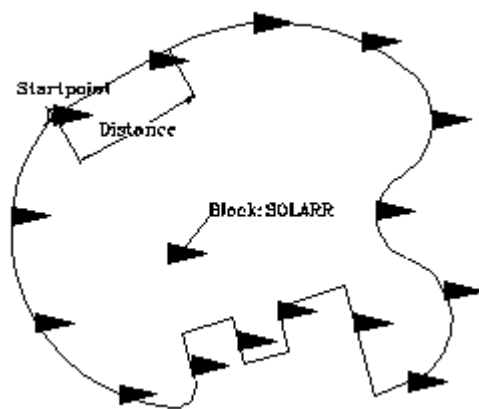
The following pictures show at the sample of a polyline, how the commands MEASURE and DIVIDE are working:



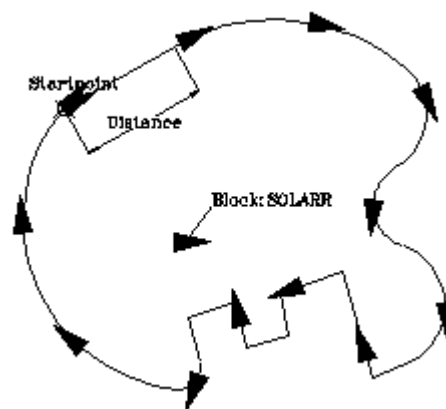
The initial state: The entity to measure is the polyline. Moreover the block SOLARR is already defined in the drawing.



After measuring the polyline using the distance, we receive points at each distance along the polyline, starting at the polyline start point



After dividing the polyline in ten pieces the block SOLARR is inserted ten times along the polyline, but it is not aligned.



In this case the polyline is divided into 12 pieces with the option block, and aligned with an angle of 0 degrees, this means the block is inserted locally tangential to the polyline.

### Technical notes

The command Divide inserts number+1 entities along the object unless a circle is selected. So it may be necessary to delete the first or last inserted entity, if a closed polyline is selected. This is much easier, however, then creating an additional entity manually.

The command is not an internal command and so it available for the usage inside the (command...) function. To use it inside of a LISP-application use the related LISP-Function (MesDiv EntName Dist|Nr B|Name OffsAngle) instead that is defined in MesDiv.LSP in the PCAD-Applic directory.

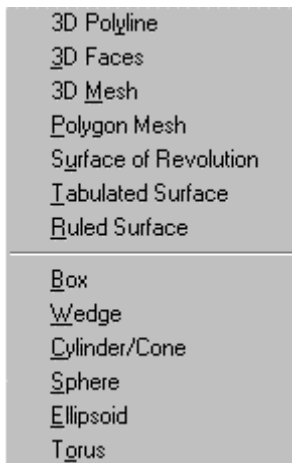
---

**See also:**

POINT

## 3-D Entities

Selecting 3-D from the Draw pull-down menu opens a secondary menu, shown below, that offers the opportunity to draw 3 dimensional entities directly.



## 3D Polyline



**Command: POLY3D**

**Alias 3P**

### **Located:**

- 1. Pull-down Menu: Draw – 3D – 3D Polyline**
- 2. Or Enter Command or Alias Directly at the Command Line**

3D Polylines are drawing elements created from straight-line segments. It is possible to define 3D coordinates for each individual 3D Polyline control point (start and end points of the individual segments).

The user may include arc segments in 2D polylines, but this is not possible with 3D polylines. It should also be noted that each individual element is assigned a standard width of zero.

As opposed to a line characteristic created using the line drawing command, the program treats 3D polygon characteristics as a single item, this also applies when the object is being processed with the edit or undo functions.

3D Polylines may be created by selecting the option from the menu item **3D Entities**, in the **Draw** menu, or by typing in the command POLY3D.

Conclude the 3D Polyline function by pressing the RETURN key.

```
> POLY3D
First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: RETURN
```

### Options:

#### Undo

The **Undo** option allows the user to undo the last line created. The end point of the last remaining segment will act as the starting point for the next segment.

The **Undo** function can be repeated as many times as necessary until the Undo List is empty. Each **Undo** step will erase the last drawn 3D polygon element.

#### Close

This option enables the user to create closed 3D Polygon objects.

Either selecting **Close** from the options bar or typing the **close** command at the keyboard will cause the last two remaining end points to be joined together.

---

#### See also:

2D Polylines

## 3D Faces



**Command: 3DFACE**

**Alias 3F**

#### Located:

1. **Pull-down Menu: Draw – 3D – 3D Faces**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

The **3D Faces** drawing function creates three or four sided surfaces "in space", or, in other words, in an X, Y, Z coordinate system. The difference between 2D and 3D surfaces is that the 3D surfaces are not shown as solid areas, but are drawn as wire frame models.

The 3D Faces definition points are fixed in either a clockwise or counter-clockwise direction so that one edge of a 3D surface is used as the base line for the next 3D surface.

The user may also draw "invisible" lines, a function that may increase the clarity of complicated 3D structures.

#### Options

##### Continuous

The option **Continuous** is the default option when drawing 3D surfaces.

To draw a continuous 3D surface, define the corner points in a clockwise or counter-clockwise direction.

For a triangular surface, define all three corner coordinates and then end the drawing with the **Close** option located. The last defined coordinate will then be joined to the first coordinate location, forming a triangle, and the drawing will be completed.

```
> 3DFACE  
First point: P1  
Next point: P2  
Next point: P3  
Next point: Close
```

For a rectangular 3D surface, define the four corner coordinates. After having completing the fourth entry, the last coordinate will be joined to the origin, thus creating a four-sided object.

This process is then continued. The user will be asked to define further points or coordinates. The next 3D surface to be defined will use the last drawn line as the starting point or base line for the next 3D surface.

End the 3D surface drawing process by pressing the RETURN key.

```
> 3DFACE  
First point: P1  
Second point: P2  
Third point: P3  
Fourth point: P4  
Third point: P5  
Fourth point: P6  
Third point: P7  
Fourth point: P8  
... and so on.
```

### 3D-Face Segments

To draw a single 3D Face, use the **Segment** option. The method is almost identical to that just described.

```
> 3DFACE  
First point: P1  
Second point: P2  
Third point: P3  
Fourth point: P4
```

In contrast to the **Continuous** option, the drawing process will end with the definition of the fourth point and the consequent closing of the contour. No further surfaces will be "connected".

This option also allows the use of the **Close** function, thus enabling triangular shaped 3D surfaces to be created.

Should the user wish to add further 3D surfaces to an existing one, this can be achieved at any time by utilizing the **Append** option.

### Append

This option allows the user to connect new 3D surfaces to existing objects. First select the edge that is to act as the base line for the connection, then proceed by defining the required corner points for the new 3D surface.

The **Append** option will automatically change the mode to **Continuous**. This means that after having connected one 3D surface the user can define additional pairs of coordinates to create more surfaces.

### Invisible Edges

The **Invisible** option is used to draw hidden lines in a 3D-Face. This feature is particularly useful when dealing with complex drawings containing a large number of 3D surfaces. The use of hidden lines will make the drawing more understandable.

This function is available for all modes used in 3D surface drawings. The function must be activated

prior to the definition of the starting coordinate of the invisible line.

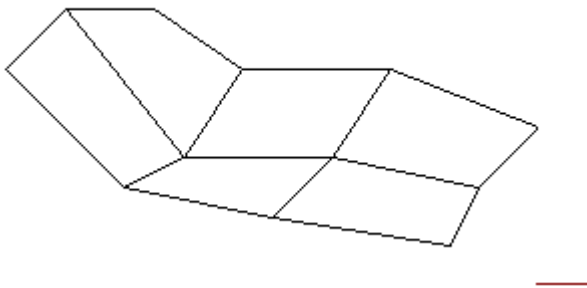
## 3D Mesh

**Command: 3DMESH**

**Alias 3M**

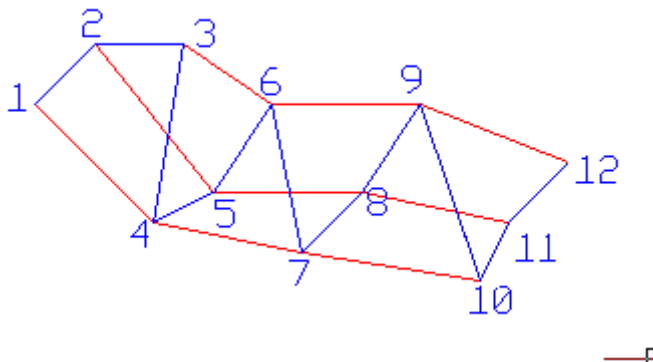
**Located:**

1. **Pull-down Menu: Draw – 3D – 3D Mesh**
2. **Or Enter Command or Alias Directly at the Command Line**



The 3DMESH command creates polygon meshes. Usually it is used to generate irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

A 3D mesh is a wireframe rectilinear blanket composed of  $M$  column lines by  $N$  row lines passing through a matrix of  $M \times N$  three-dimensional points in space.  $M$  and  $N$  are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh.



### To Create a Polygon Mesh

First you are prompted to specify the mesh  $M$  size and the Mesh  $N$  size. This is simply the number of wire intersections wide and long.

Then the program continues to ask for corners or vertices, until you have defined all of the vertices in the one ( $M$ ) direction and the other ( $N$ ) direction. Remember that to complete the mesh, you will need to enter  $M \times N$  vertices. You are prompted for each vertex in order, starting with vertex (0,0), continued with the second vertex in the first column (0,1), and so on. Once all vertices of a column are specified, the first vertex of the next column is requested. The blue line traces the input order in the example further below.

1. During vertex input, the mesh does not appear onscreen but is instead being constructed in memory. Terminating the command places the mesh onscreen
2. Meshes can be open or closed depending on whether the mesh joins in either the  $M$  or  $N$

direction, or both. A donut mesh would be an example of a mesh closed in both directions.

### Example

```
> 3DMESH
M-value of the mesh (between 2 and 256): 4
N-value of the mesh (between 2 and 256): 3
Vertex (0,0): 110,170,4
Vertex (0,1): 130,190,4
Vertex (0,2): 160,190,6
Vertex (1,0): 150,130,0
Vertex (1,1): 170,140,0
Vertex (1,2): 190,170,2
Vertex (2,0): 200,120,0
Vertex (2,1): 220,140,-2
Vertex (2,2): 240,170,0
Vertex (3,0): 260,110,0
Vertex (3,1): 270,130,0
Vertex (3,2): 290,150,0
```

### Polygon Face

**Command: PFACE**

**Alias PF**

#### Located:

1. **Pull-down Menu: Draw – 3D – Polygon Mesh**
2. **Or Enter Command or Alias Directly at the Command Line**

The PFACE command creates three-dimensional polygon meshes of arbitrary topology and surface properties.

#### To create a polyface mesh

Specify the control points (vertices) first.

Then specify that vertex is on that face of the polyface mesh.

#### Note:

The PFACE command is rarely used as an explicit command in drawing practice. However, the command provides an easy way to generate meshes by Lisp routines based on 3D point lists (like surveyor's measurement points). The command is primarily designed for use within applications (either based on Lisp or on other programming languages) to construct general polyface meshes.

Do not confuse polyface meshes created by PFACE with polygon meshes created by the 3DMESH command.

### Example

```
> PFACE
Control point 1: Specify a point
Control point 2: Specify a point
Control point 3: Specify a point
Control point 4: Specify a point
Control point 5: Specify a point
Control point 6: Specify a point
Control point 7: Specify a point
Control point 8: RETURN
Face 1, Control point 1: 1
Face 1, Control point 2: 2
```



Face 1, Control point 3: **3**  
 Face 1, Control point 4: **4**  
 Face 1, Control point 5: RETURN  
 Face 2, Control point 1: **1**  
 Face 2, Control point 2: **4**  
 Face 2, Control point 3: **5**  
 Face 2, Control point 4: RETURN  
 Face 3, Control point 1: **5**  
 Face 3, Control point 2: **6**  
 Face 3, Control point 3: **7**  
 Face 3, Control point 4: RETURN  
 Face 4, Control point 1: **4**  
 Face 4, Control point 2: **5**  
 Face 4, Control point 3: **6**  
 Face 4, Control point 4: RETURN  
 Face 5, Control point 1: **3**  
 Face 5, Control point 2: **4**  
 Face 5, Control point 3: RETURN  
 Face 6, Control point 1: RETURN

Another null response terminates the command.

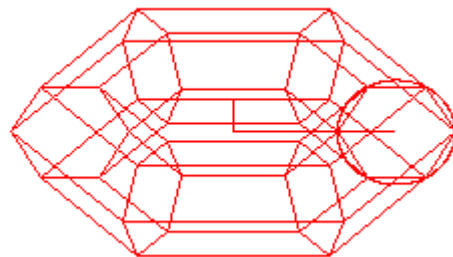
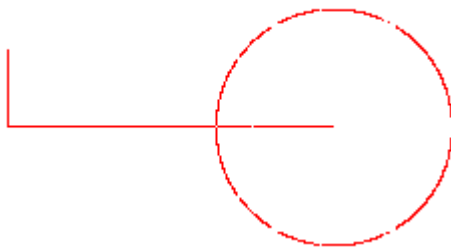
## Surface of Revolution

**Command: REVSURF**

**Alias RS**

### Located:

1. **Pull-down Menu: Draw – 3D – Surface of Revolution**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**

The REVSURF command creates a surface by revolving a path curve (the profile) around an axis of revolution.

The curve sweeps around the selected axis to define the surface in the *N* direction of the mesh. The *M* direction of the mesh is determined by the axis of revolution. The result is determined by the SURFTAB1 and SURFTAB2 settings.

### To generate a surface of revolution

First, pick the entity (a line, arc, circle, or polyline) to revolve. This determines the profile entity.

Next, pick the axis (a line or open polyline) around that the profile entity revolves.

Finally, you are prompted to determine the start angle and the included angle to revolve the profile. The default is a full circle.

> REVSURF

Select path curve to revolve:  
Select axis of revolution:  
Start angle <0>:  
Included angle to revolve entity (+ccw, -cw) <360>:

**Tip**

To close the mesh in the *N* direction, select a circle or closed polyline as the curve.

Please check in the Learning Center for a fully documented example of this command's use.

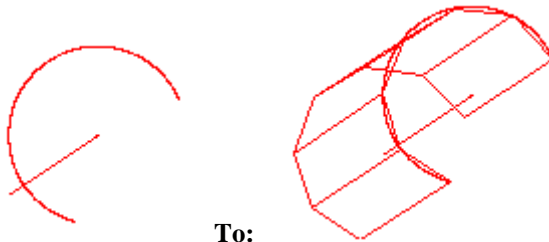
**Tabulated Surface (Extruded Surface)**

**Command: TABSURF**

**Alias TF**

**Located:**

1. **Pull-down Menu: Draw – 3D – Tabulated Surface**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**

The TABSURF command creates a tabular surface using a path curve and a direction vector.

You can extrude lines, arcs, circles, 2D or 3D polylines. The entity provided as path curve defines the surface of the mesh. The direction vector (extrusion path) must be determined by picking a line or an open polyline.

The system variable SURFTAB1 defines the number of *M*-direction surfaces on the extruded entity. When extruding a line, arc, or circle, the number of *M*-direction surfaces is evenly divided determined by the value of the SURFTAB1 system variable. When extruding a polyline, extrusion lines are drawn at the ends of the line segments, and the arc segments are divided evenly into the number of surfaces determined by the value of the SURFTAB1 system variable.

If the polyline has been curved (fitted) or splined, the number of *M*-direction surfaces is evenly divided using the SURFTAB1 variable. Like with the command RULESURF, you only need to set SURFTAB1 system variable.

The resulting surface is a polygon mesh (stored in the drawing database as a complex polyline entity).

**To create a tabulated surface**

After you select the entity for the path curve, you specify the direction vector:

> **TABSURF**

Select path curve:

Select direction vector (line or open polyline):

The direction and length of the line determine the direction and length of the extrusion. Only the first and last points of a polyline are used when determining the extrusion path. The point at that you specify the entity determines the extrusion direction.

Please check in the Learning Center for a fully documented example of this command's use.

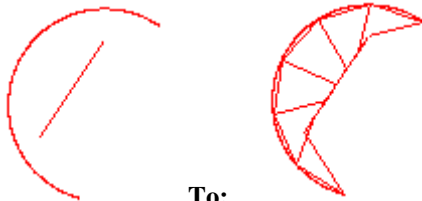
## Ruled Surface

**Command:** RULESURF

**Alias** RSF

### Located:

1. **Pull-down Menu:** Draw – 3D – Ruled Surface
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**

The RULESURF command allows you to create a ruled surface between two boundary curves (two profiles).

Valid entities that define the edges of the surface can be points, lines, arcs, circles, 2D polylines or 3D polylines.

The system variable SURFTAB1 determines the number of equal intervals between the resulting vertices on the defining profile entities. The number of intervals is the same for each curve.

The resulting surface is a polygon mesh (stored in the drawing database as an complex polyline entity).

### To create a tabulated surface

First pick one entity defining the first profile, then select another entity for the other boundary outline:

> **RULESURF**

Select first defining curve of ruled surface:

Select second defining curve of ruled surface:

### Note

If the first entity is an open entity (such as an arc or line), the second entity must also be an open entity. If the first entity is a closed entity (such as a circle or a closed polyline), the second entity must also be a closed entity. If one entity defining an edge is a point, the other entity cannot be a point entity.

### Tip

To create a commonly used polygon mesh, select each entity at the same end. If you select each outline entity at opposite ends, you generate a self-intersecting polygon mesh.

Please check in the Learning Center for a fully documented example of this command's use.

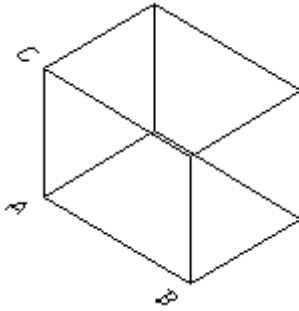
## Command 3DBOX

**Command:** 3DBOX

**Alias** 3B

### Located:

1. **Pull-down Menu:** Draw – 3D – Box
2. **Or Enter Command or Alias at the Command Line**



The command 3DBOX allows simple creation of a 3 dimensional box. The command can also be accessed from the Draw - 3D pull-down menu.

Upon executing the command, you are asked to input the first corner ("A" in the diagram). As with the input of all location definition, you may enter coordinate values in the form x,y,z or you may pick an existing reference point in your drawing. Unless the point you want has a z or height/depth value of zero (default value for 2D objects), be sure that the point you pick has the correct z value.

Once the first corner has been established you will be presented with an option dialog offering you the opportunity to input a length to establish the second corner. The z value (height) of the second point is the same as the first and the first side is parallel to the x-axis (Y values of A and B are equal).

You will be prompted for width if you chose the length option and presented with a dialog offering Cube and Square options.

Selecting **cube** indicates that the x, y and z dimensions are equal to A-B and the cube is completed. Choosing **square** indicates that the x and y values (only) are equal to A-B and the square is drawn with a following prompt for height input, that completes the box.

The second point (picked point "C") would be regarded as the diagonally opposite corner of a rectangle and be drawn, if you were to ignore the Length option and input a point with a different y value, after setting the first corner. A prompt for height would follow.

For height, an absolute value can be input. Picking a point "D", however, will set height as being the 3 dimensional distance between points "A" and "D".

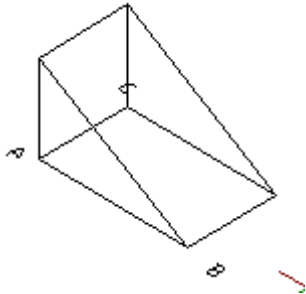
## Command 3DWEDGE

**Command:** 3DWEDGE

**Alias** 3W

### Located:

1. **Pull-down Menu:** Draw – 3D – Wedge
2. **Or Enter Command or Alias at the Command Line**



The command 3DWEDGE allows creation of a 3 dimensional wedge. This command can also be entered from the Draw - 3D pull-down menu.

### Command 3DCONE

**Command:** 3DCONE

**Alias** 3DC

#### **Located:**

1. **Pull-down Menu: Draw – 3D – Cylinder / Cone**
2. **Or Enter Command or Alias at the Command Line**



The command 3DCONE allows creation of a 3D cone, cylinder, prism, or pyramid. This command can also be executed from the Draw - 3D pull-down menu. Object creation is determined by user input. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

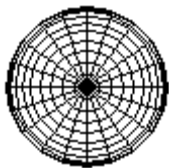
### Command 3DSPHER

**Command:** 3DSPHER

**Alias** 3DS

#### **Located:**

1. **Pull-down Menu: Draw – 3D – Sphere**
2. **Or Enter Command or Alias at the Command Line**



The command 3DSPHER creates a 3D sphere (ball). This command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

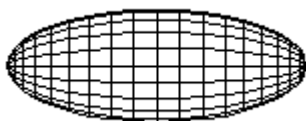
## Command 3DELLIPS

**Command: 3DELLIPS**

**Alias 3L**

**Located:**

1. **Pull-down Menu: Draw – 3D – Ellipsoid**
2. **Or Enter Command or Alias at the Command Line**



The command 3DELLIPS (Draw>3D>Ellipsoid) allows creation of a 3D ellipsoid object. This command may also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

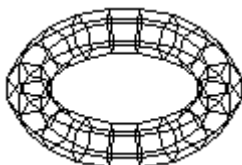
## Command 3DTORUS

**Command: 3DTORUS**

**Alias 3T**

**Located:**

1. **Pull-down Menu: Draw – 3D – Torus**
2. **Or Enter Command or Alias at the Command Line**



The command 3DTORUS allows creation of a 3D torus. The command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

## Drawing Modes...



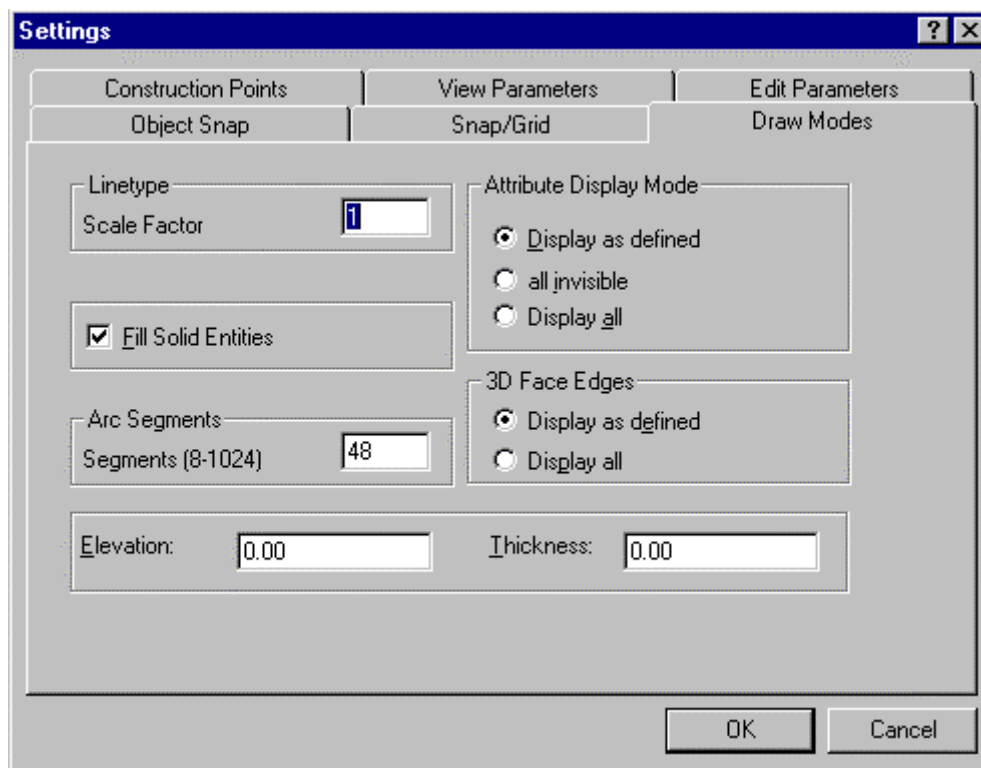
Command: **DRAWMODE**

Alias **DR**

### Located:

1. **Pull-down Menu:** Draw – Drawing Modes
2. **Palette:** Draw
3. **Or Enter Command or Alias Directly at the Command Line**

The Drawing Modes tab (on the SETTINGS command dialog) allows you to specify default settings for subsequent drawing commands. Parameters may be changed, activated, or inactivated.



### Dialog Box Items

#### Linetype Scaling Factor

*Linetype Scaling Factor* determines a scaling factor for the line pattern selected. The line patterns are shown extended (long dash lines, for instance) or contracted (short dash lines, for instance) according to this scale. The default value is 1.00. To change the scale, enter a new factor and verify it.

Remember that, after a regeneration of the drawing, the scales of line patterns that have been drawn already will be adjusted to the new value. Results of *Ltscale* changes will therefore not be apparent until after a REGEN.

#### Filling Solid Entities

*Fill Solid Entities* specifies whether or not to "fill" a polygon. Polygons are closed 2-D planes, or polylines with a width greater than zero.

A cross in the box indicates whether or not the "fill" function is activated.

Remember that activating the function is required in order to draw filled planes. If the option *Fill Solid Entities* is deactivated, then the option *Fill Polygons* will not be available in the Dialog Box *Print/ Plot*.

### Attribute Display Modes

This option allows the user to choose the desired method of display of attributes. The options are:

<b>Show as Defined</b>	All attributes are displayed according to their definition.
<b>Show Nothing</b>	All attributes are made invisible, e.g. their status is defined as invisible.
<b>Show All</b>	All attributes are displayed, independently of how their attributes were defined.

Choose a display option by clicking its field.

### 3D Frame Visibility

If, in the display of 3-D views, intersections or lines that are "behind" other drawing segments are to be hidden, then the user should select the option *normal*. If, on the other hand, all intersections are to be displayed, then the selection should be *show all*.

### Arc Segments

Arcs and circles, understandably, are always displayed as a series of short straight-line segments.

In this area the user selects the number of segments that will appear in the definition of any arc or circle that is drawn. The number selected may be any whole number between 8 and 1024. A smaller number will make the arcs or circles appear "jagged"; a larger number will tend to smooth them out. The default number used is 48.

Remember that a large number of line segments in an arc or circle will tend to slow the performance, whereas a small number of segments will increase working speed.

### Point Representation

This dialog box area allows you to choose between 20 different display forms for construction points. The point forms will be updated when the drawing is regenerated.

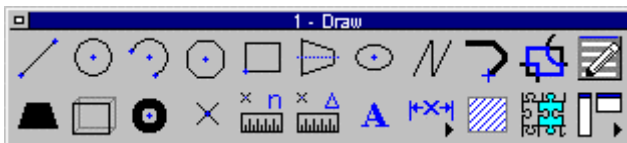
### Point Size

The user chooses the size of the construction point in this area.

<b>Absolute</b>	The size of the construction points is determined in the absolute scale of drawing unit .
<b>Relative</b>	The display of the construction points is determined relative to the size of the visible drawing plane.

This parameter defaults to *Absolute*, and uses 5.00% of the drawing unit size as the size of the construction point.

### Draw Palette



This command will place the palette containing Draw command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.



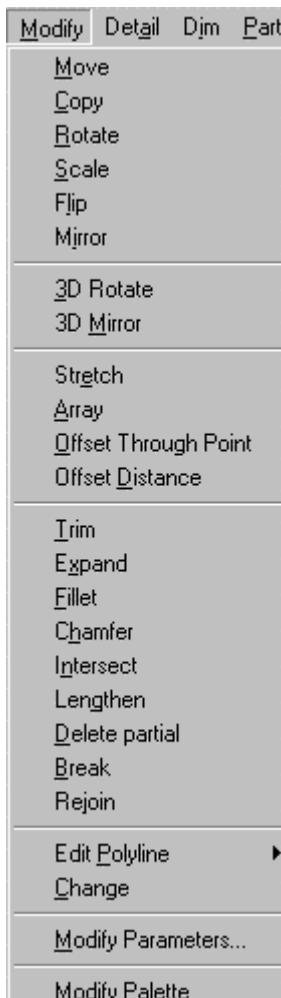
## Snap Palette



This command will place the palette containing Snap command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

## Modify

### "Modify" Pull-down Menu



The Modify pull-down menu contains command instructions that edit drawing elements. Additional

editing commands may be found under the Edit pull-down menu. For a description of each command, please refer to the specific topics.

## Move



**Command: MOVE**

**Alias M**

### Located:

1. **Pull-down Menu: Modify - Move**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command is used to move objects within the coordinate system. An object can be moved in any combination then, of the x, y, or z directions.

The first prompt will be to select the object to be moved. Multiple objects may be selected, and they will be moved together in exactly the same relationship as they are currently placed. Select the object or objects, and confirm the selection by pressing RETURN.

### Options

#### Basepoint

It is the standard reference point for moving objects. It denotes the point that the user is choosing as the base from that to begin the command.

#### Rel. Point

The prompt will then ask for distances in the X, Y, and Z direction. Enter these values (if there is no Z value, enter only the X and Y values). The move will occur when the values are verified.

#### Quick Move

This method of accomplishing the move is to simply pick up the object(s) with the cursor, and "rubberband" them to the new desired location. This works well if no particular precision is required.

## Copy



**Command: COPY**

**Alias C**

### Located:

1. **Pull-down Menu: Modify – Copy**
2. **Top Function Bar**
3. **Palette: Modify**
4. **Or Enter Command or Alias Directly at the Command Line**

### Duplicating drawing objects

This function allows the user to copy drawing elements to AllenCad's clipboard and later put them at a specified place.

The copies created by this command are exact copies of the selected entities. All entity attributes such as layers, line type, color, and all other attributes are copied as well as the outline of the object(s).

### Options:

#### Basepoint

If **Basepoint** is the selection, enter the base point, then, as prompted, the **Target Point** (the position to which the Copy is to move). The copy will be made in the position specified.

#### Rel.Point

**Rel.Point** may be chosen if the copy is to be placed in a position relative to the current image of the object.

The prompt will ask for the relative points by specifying the move in the X, Y, and Z directions. If the move is to be made in the X and Y directions only, simply do not specify a Z delta. The copy will be placed in the specified position.

## Rotate



**Command: ROTATE**

**Alias RO**

#### Located:

1. **Pull-down Menu: Modify - Rotate**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Rotate will serve to rotate an entity or entities through an angle, about an axis.

As with the other modification commands, Rotate requires the selection of the object/ objects to be rotated. There are several methods suggested by the Option Bar of selecting the object(s). Choose a method, and make the selection.

### Options

#### Basepoint

The command will prompt for the Basepoint. The Basepoint is the point about that the rotation will occur. It is handy to pick a corner, or a center, but any point may be picked that will give the desired result. Enter, or pick, the Basepoint.

Note that the Basepoint may not be a point on the described geometry of the object or objects, but may fall off of their geometry.

The next prompt will ask for the angle of rotation. The user may elect, through the Option Bar, either the **Angle of Rotation**, or a **Reference Angle**.

#### Angle of Rotation

The option Angle of Rotation is the standard option. It is used if no other option has been selected.

Angle will prompt the user for a rotation angle. Enter that angle, and rotation will take place to the specified angle.

#### Reference Angle

The Reference Angle will ask the user to specify the angle at that the object is now placed, and then prompt for a New Angle, to which the object will be rotated. Rotation will always be counter-clockwise. As before, rotation will occur about the base point.

## Scale



**Command: SCALE**

**Alias SC**

**Located:**

1. **Pull-down Menu: Modify - Scale**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The **Scale** command allows the user to change the "scale" of the objects that is, to enlarge or reduce them while keeping all proportions the same.

The first prompt will be to select the object(s) to be scaled. Use one of the options presented in the Options Bar for this. Remember that if more than one object is selected, the objects will be treated as one by the command.

The next prompt will be to identify the **base point**, and that should be done as it was in the two previous commands. This will bring up the Options Bar with the following options:

Sc.Factor | Reference

**Options**

**Scaling Factor**

The option **Scaling Factor** is the standard option for scaling objects; and is used if no other option has been selected.

Note that it is important to realize that the **base point**, if is a part of the selected object, will hold its original position during the scaling, while all the other points will be moved by the same relative values in direction of the X- and Y-axis. As in other cases, however, the user may select any other point, whether on or off the geometry of the object(s). In this case, the distance from the base point to all of the other points that comprise the object will be changed by the same scale factor.

To continue, enter a scaling factor. The use of a scaling factor bigger than 1 results in enlarging the selected objects by this factor. A scaling factor between 0 and 1 results in reducing the objects by that factor.

For instance, the factor <3> enlarges the selected objects three times, while the factor <0.25> results in reducing the objects to one-quarter of their original size.

**Reference**

This option allows an exact change of the size of an object to a particular scale.

For instance, if the user were to enlarge an object with a side length of 3.45 drawing units up to 6.90 drawing units, he or she would first have to calculate the scaling factor. **Reference**, however, facilitates this calculation by the input of a reference length (here 3.45) a precise changing of the size up to the desired scale (here 6.90), without the user having to calculate a scaling factor.

To use this variation of scaling, select the option **Reference** from the Option Bar. Enter the **Reference length**, and then enter the **New length** that is desired.

**Flip**



**Command: FLIP**

**Alias FL**

**Located:**

1. **Pull-down Menu: Modify - Flip**

## 2. **Palette: Modify**

## 3. **Or Enter Command or Alias Directly at the Command Line**

The command **Flip** contrasts to the **Mirror** command, in that the original object does not remain in its original position; it is simply "flipped" about an axis specified by the user.

The selected objects have, after "flipping", the same relative position to the symmetry axis as they had prior to "flipping", but are now "flipped" in a new direction.

The reflection axis can be defined freely, or parallel to the X-axis or the Y-axis

The first prompt will be for object selection.

Next, select one of the options for choosing the axis from the option bar:

### **Axis**

The option **Axis** allows the user to freely define the flip axis. It is the standard option for axis definition, and is used when no other option has been selected.

Define the position of the axis by the indicating two points or by the input of their coordinates. The axis can have any alignment toward either the objects or the axes of the coordinate system.

If the mirror axis should run through any of the points of existing geometry of the object(s), you should specify object snap functions to define the points precisely.

### **Horizontal Flip**

To flip the marked object about a horizontal axis choose the option **Horizontal**. This option creates a flip axis that runs parallel to the Y-axis of the coordinate system. Only a single point is required to define this axis.

### **Vertical Flip**

To flip the marked object about a vertical axis choose the option **Vertical**. This option creates a flip axis that runs parallel to the X-axis of the coordinate system. Only a single point is required to define this axis.

### **Mirror**



### **Command: MIRROR**

### **Alias Mi**

#### **Located:**

1. **Pull-down Menu: Modify - Mirror**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Mirroring drawing objects produces identical copies of selected entities that are placed in a "mirrored" position about an axis specified by the user.

The selected objects have, after "mirroring", the same relative position to the symmetry axis as they had prior to " mirroring".

The mirror axis can be defined freely, or parallel to the X-axis or the Y-axis

In order to use the command the user must select an object to be mirrored. Select the object or objects, remembering that the selection of multiple objects will result in the treatment of the objects as one.

The next prompt will ask for the definition of the mirroring axis.

### **Options:**

### Mirroring With a Free Axis

The Option **Axis** allows the user to freely define the mirror axis. It is the standard option for mirroring objects and is used when no other option has been selected.

Select this option, then define the position of the mirror axis. This can be made by indicating two points or by inputting their coordinates. The mirror axis can have any alignment with regard to the objects to be mirrored as well as with regard to the axes of the coordinate system.

If the mirror axis should run through points of an existing geometry (for instance, along a solid edge), the user should employ the object snap options for precise definition.

### Mirroring on a Horizontal Axis

To choose a horizontal-mirroring axis, select the option **Horizontal**. This option creates a mirror axis that runs parallel to the Y-axis of the coordinate system. Its exact position is indicated by a single point.

### Mirroring on a Vertical Axis

To choose a vertical-mirroring axis, select the option **Vertical**. This option creates a mirror axis that runs parallel to the X-axis of the coordinate system. Its exact position is indicated by a single point.

### Note: Mirroring Text

If there is text associated with the selected object or objects, it is likely that the user does not want to mirror that text. In order to mirror the text it is necessary to activate an option called **Text Mirroring**. Activating this option will result in the text also being mirrored.

## Command 3DROT

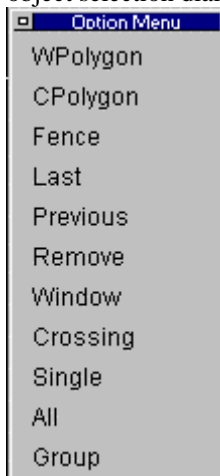
**Command: 3DROT**

**Alias 3R**

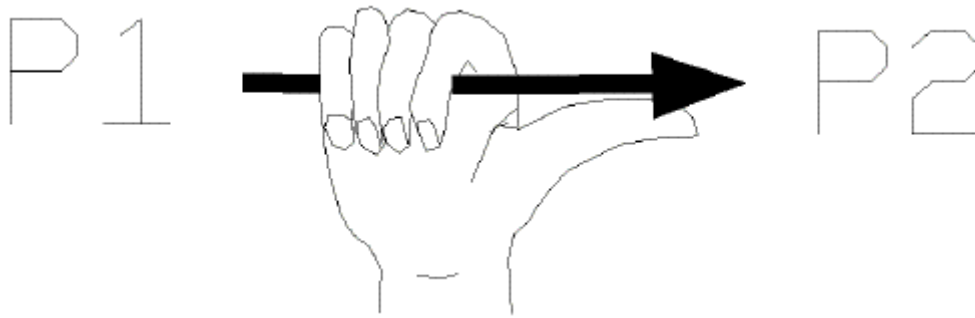
**Located:**

1. **Pull-down Menu: Modify – 3D Rotate**
2. **Or Enter Command or Alias at the Command Line**

When you issue the command, you are asked to select the objects for rotation and presented with the object selection dialog.



Next, you are asked to define the axis of rotation, at which point you may enter the 3 dimensional point coordinates for P1 (x1,y1,z1) and P2 (x2,y2,z2) or pick points on the drawing editor. You are then asked to define an angle of rotation, where a positive rotation is a rotation counter-clockwise around the axis and a negative angle is clockwise. If you find this confusing, don't feel too badly. Fortunately for us all there is a "**Right-Hand Rule**" that helps to explain things.



In this rule, the axis of rotation is shown as the arrow gripped by the right hand, where the direction thumb points from P1 to P2 in the direction of the thumb. Rotation, from the start or existing point, is indicated by the fingers on the hand (counter-clockwise when viewed along the axis from the P2 position).

The input sequence is as follows:

```
> 3DROT
```

```
Select objects: {PICK}
```

```
*** 6 selected. *** [ a crossing selection was used in this instance]
```

```
First point of rotational axis <0,0,0>: x1,y1,z1 [IE P1 coordinates]
```

```
Second point of the rotational axis <0,0,1>:x2,y2,z2 [IE P2 coordinates]
```

```
Rotation angle: 45 [A positive angle of 45o is selected]
```

```
The view then changes according to the input parameters.
```

## Command 3DMIR

**Command: 3DMIR**

**Alias 3MR**

**Located:**

- 1. Pull-down Menu: Modify – 3D Mirror**
- 2. Or Enter Command or Alias at the Command Line**

This command creates a mirror image in a 3-dimensional plane around a defined 3 dimensional axis. Essentially, what is originally visible in this 3-D Plane is copied in a reverse image on the other side of the axis at the end of the command, much as with the 2D MIRROR command.

The input sequence for this command is as follows:

1. First, you are asked to select the objects for mirroring
2. You are then asked to define the mirror plane by picking three points or entering their coordinates
3. Next, you define the mirror axis by picking two points or entering their coordinates
4. With this information, the command completes its operation.

IE.

```
> 3DMIR
```

```
Select objects:
```

```
1 selected.
```

```
Select objects:
```

```
*** 1 selected. ***
```

```
First point of the mirror plane <0,0,0>:0,0,0
```

```
Second point of the mirror plane <0,0,0>:0,5,10
```

```
Third point of the mirror plane <0,0,0>:0,20,10
```

```
Select objects: Echoed only
```

Select objects: **Echoed only**  
\*\*\* 1 selected. \*\*\*  
First point of mirror axis: **No prompt - bypassed**  
Second point: **No prompt - bypassed**

## Stretch



**Command: STRETCH**

**Alias SR**

### **Located:**

1. **Pull-down Menu: Modify - Stretch**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The **Stretch** command is useful for the enlargement or reduction of objects in only one direction (in contrast to scaling). Subsequent stretching of the same object in a second direction is not supported. Standard geometric entities (circles, rings, etc.) are not distorted, only shifted in position relative to the other objects being stretched.

Selecting entities for stretching must be done very carefully to ensure the desired effects, as **selection is by crossing and crossing polygon only**. During stretching, the method of selection has an influence on the question of which entities are moved and which entities are stretched.

- All entities included totally in the selecting windows will be moved;
- All entities crossing the selecting window will be stretched (lengthened); and
- All other drawing entities will retain their position and shape.

After having completed object selection, choose an option for stretching from the Option Bar..

### **Options:**

#### **Basepoint**

The option **Basepoint** is the standard option for moving entities. It is used if no other option has been selected.

The option is useful for moving selected drawing entities to a particular position and for "stretching", or moving, entities on a border, without changing the relationship to the border. The target position may be a point defined by coordinates of a position determined by the geometry of the object.

Next, determine the end point of the stretch. The base point for the stretch will be moved to the end point.

#### **Displacement**

The option **Displacement** allows stretching selected entities by a straight line (a vector). After selecting the objects, select the option from the Option Bar. Enter the value for the move, a straight line, in the form X,Y, (Z).

## **Array**

**Command: ARRAY**

**Alias RA**

### **Located:**



1. **Pull-down Menu: Modify - Array**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Array** represents a variant of Copy, except that Array will create multiple copies, and they can be arranged at random, or in rows, or in vertical order, or on a circle, or on an arc.

The entities created by the command are exact copies of the selected entities in which all entity attributes such as layers, line types and color are copied.

After entering the command, select an option from the Option Bar:

Individual  
Rectangular  
Circ-NRot  
Circ-Rot

Options



#### **Individual Array**

This mode allows the user to copy the Array as many times as is required during one command sequence and place them at random on the drawing area.

The use of this option is very similar to the command Copy.

After choosing the option, select the entities to be copied using the selection modes described in the Option Bar. Always confirm object selection by pressing RETURN.

Determine the base point for the copying. A particularly discreet point in the geometry or in one of the selected objects (edges, corners, intersecting points etc.) will be useful. Next, either input the coordinates of the target point(s) or "pick" the points for placement of the Array. The program will create a copy of the Array after each definition of a target point.

As long as the command remains active the user can produce and place as many copies as are required. Terminate the function by pressing ESC or RETURN.



#### **Rectangular Array**

This option will arrange the Array copies in rows or columns. The use of this function is the same as was the use of the **Individual** mode.

Determine the number of the columns and rows in which the Array copies should be arranged. Define the distance between the columns and the rows.

Note that, when planning the arrangement, the rows and columns must be in the positive X or Y-axis. This infers that the original Array is situated in the lower left-hand corner of the quadrant plus-XY. If it is not located there, it may be useful to move the original Array before attempting to begin defining the copying positions. Use the command Move to do this.

### **Circular Arrays**

The Array can also be reproduced in a circular path. The circle (arc) itself is not displayed on the screen. The reproduction of the Array is described as follows:

1. The center of the circle (arc)
2. The number of times the Array is to be reproduced;
3. The included angle of the arc (360 if the path is to be a full circle) to be filled;

There are two ways to reproduce the Array on a circular path:



### Placing the Arrays Without Rotation

This Option, will arrange the reproductions on the circle (arc) in the same alignment to the principal axes as the original Array.

Define the center of the circular path. Then determine **either**;

The number of the entities to be arrayed (including the original); and  
the included angle of the circle (arc),

or

The included angle of the circle (arc); and  
the included angle between the Arrays to be reproduced (measured **between** the reference points of the objects).

Note that, during the planning of the reproduction of the Array on a circle that the circle (arc) runs **counter-clockwise** from the starting point. For a **clockwise rotation** enter negative values for the angle inputs.



### Placing the Arrays With Rotation

This option will rotate the Arrays when placing them on the circular path. The Arrays will be rotated in the same direction as the circle arc that describes the path of reproduction. The rotation angle will correspond to the angle between the Arrays (measured between the reference points of the Arrays).

After selecting the command, choosing the objects in the Array, and choosing the option **Ci.Rot** from the Option Bar, define the center of the circular path and then determine **either**;

the number of the entities to be arrayed (including the original); and  
the included angle of the circular arc,

or

the included angle of the circular arc; and  
the included angle between the entities to be arrayed (measured between the reference points of the objects).

As parameter value for the Arc angle there is always used the standard value 360 degree, i.e. the copies are arrayed on a full circle, if this value has been confirmed.

Please note during the planning of the arraying on a circle of multiple copies that the circle (arc) runs from the starting point that is the original object anti-clockwise. For a clockwise rotation please enter negative values for the angle inputs.

### Offset Through Point

#### Located:

1. **Pull-down Menu: Modify - Offset Through Point**
2. **Or Enter Command or Alias at the Command Prompt**



The Offset command (Modify > Offset Through Point) creates parallel copies of lines, circles, arcs, and 2D polylines. The command copies the selected entity and places it a specific distance from the original. The original entity stays in place.

**Note:** The Offset command can create unexpected results, depending on the entity selection:

Line and straight 2D polyline: an identical copy is created, to make parallel lines.

Circle and arcs: n copy is created, but is scaled by a factor to make concentric curves. The center of the original circle or arc is used as a base point for scaling.

3D polyline made of line and arc segments: due to the arc portions being scaled, the offset polyline may look very different from the original.

## Lines & 2D Polylines

Start the Offset command, and select the line(s) or polyline(s) to be offset.

There are two ways to specify the offset distance. One method is to pick a point, using object snaps if necessary. AllenCad offsets the copy at the pick location.

> OFFSET

Select objects (line, circle, arc, 2D-polyline): P1

Offset by point: P2

Select objects (line, circle, arc, 2D-polyline): [Enter]

The Offset command continues, until you press Enter or Esc.

## Distance Offset

**Command: DOFFSET**

**Alias DOF**

**Located:**

- 1. Pull-down Menu: Modify – Offset Distance**
- 2. Or Enter Command or Alias Directly at the Command Line**

The DOFFSET command is similar to the command OFFSET, but not identical. Like OFFSET, the command DOFFSET creates lines and curves parallel to an existing entity. DOFFSET has been added for convenience of those users who are familiar with other CAD programs.

The main difference is that the command DOFFSET assumes that you want to input an offset distance first before you offset selected entities:

> DOFFSET

Distance or select option ThruPoint <1.00>: **0.30**

Select entity (line, circle, arc, 2D-polyline):

Side to offset ?

Select entity (line, circle, arc, 2D-polyline):

Side to offset ?

Select entity (line, circle, arc, 2D-polyline): <RETURN>

If you want to indicate a through point (such as a significant point in the given geometry), select the option *ThruPoint* (or type, for short, T):

> DOFFSET

Distance or select option ThruPoint <0.30>: **T**

Select entity (line, circle, arc, 2D-polyline):

Offset through point:

Select entity (line, circle, arc, 2D-polyline):

Offset through point:

Select entity (line, circle, arc, 2D-polyline): <RETURN>

You can offset an entity to both sides by the specified distance by selecting the option *BothSides* when you are asked to specify the side to offset:

> DOFFSET

Distance or select option ThruPoint <0.50> :

Next point:

Select entity (line, circle, arc, 2D-polyline):

Side to offset ? **BothSides**

Select entity (line, circle, arc, 2D-polyline): <RETURN>

Side to offset ?

---

See also:

OFFSET

## Trim



Command: TRIM

Alias T

### Located:

1. **Pull-down Menu: Modify - Trim**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The TRIM command cuts off sections of linear objects, using another linear object as the "cutter".

The first prompt for this command will be a request to identify the entities to make up the "cutting edge". These entities will be the ones that will result in the "trimmed" intersection of lines. Pick these entities with the cursor.

The next prompt will ask you to select the entity to trim. When you pick an object, that portion of the object on the "selected" side of the cutting object, is trimmed off and deleted.

Note: when trimming multiple objects at once, using the same "cutter", consider using the object selection tool "fence" to trim all linear objects at once that cross the "cutter" along its selected length.

### Expand



Command: EXPAND

Alias EXP

### Located:

1. **Pull-down Menu: Modify - Expand**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Expand** allows the user to extend objects to a selected common boundary edge. In this way, it is almost identical to the EXTEND command.

During the execution of the command the drawing entities to be expanded are extended. They do not change direction. A line will remain a line, an arc will remain an arc. For chains and polylines, only the last segment will be extended to the selected boundary.

Select the entities that are to serve as boundary edges for the entities to be expanded. These boundary entities will remain unchanged. Object selection functions are presented in the Option Bar.

Next, select the entities to be expanded. In doing this, make sure that the selecting window of the cursor is on the part to be trimmed.

The expansion of selected entities is always executed from the endpoint that lies nearest to the cursor when it selected the entity.

If more than one boundary edge has been selected, the entity is expanded to the first boundary edge first. "Picking" a second time will result in the expansion being continued to the second boundary edge.

## Fillet



**Command: FILLET**

**Alias F**

**Located:**

1. **Pull-down Menu: Modify – Fillet or  
Modify – Edit Polyline – Fillet Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Often two solid edges do not intersect exactly, in acute or right angles. With the Fillet command solid edges can be filleted quickly and precisely.

The first prompt will request the selection the two lines whose intersection is to be filleted. These can also be lines of an object such as a rectangle.

**Options:**

The following options will be shown in the Options Bar.

**Polyline**

Instead of placing a radius between two separate linear entities as with the standard FILLET command, FILLET-POLYLINE will apply the radius at all vertices in the selected polyline. Note: this command acts upon a polyline only once. To change the radius you must UNDO the first fillet command or else reconstruct the polyline by exploding it, deleting the arcs and INTERSECTing the segments and then using POLYEDIT-JOIN to reconnect the segments to form the original polyline.

**Radius**

The option **Radius** will defined the radius value for the fillet. The program will always offer the value of the last radius used in the current command operation. A radius value of zero means no radius will be applied to the fillet and that entities will simply be trimmed off where they cross each other.

**Trim (Off)**

**Trim** functions as a switch. It decides whether the parts of lines that lie beyond the fillet are erased or whether they will be left as separate entities.

If the switch in the Options Bar displays the term **Trim**, you can **switch off** the function **Trim** by clicking the switch. It will then change to **Trim (Off)**.

If the switch in the Options Bar displays **Trim (off)**, by clicking the switch the function will change to **Trim**.

**Chamfer**



**Command: CHAMFER**

**Alias CH**

**Located:**

1. **Pull-down Menu: Modify – Chamfer or  
Modify – Edit Polyline – Chamfer Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

When executing the command, the first prompt will ask for the entities to be chamfered. Select those entities by picking them with the cursor.

In the Options Bar, the option **Distance** will appear.

There are two choices: either select, and enter, a distance for the chamfer, or choose to "pick" the distance for the chamfer. The distance back from the point of intersection for the chamfer must be specified for both of the intersecting entities, since there are often non-symmetrical chamfers.

The option **Trim (Off)**, or **Trim**, functions in exactly the same manner as described with the command **Fillet**, as does the option **Polyline**.

The option **Angle** specifies the relative angle that will be applied to the chamfer, a given distance back from the intersection, on object 1.

---

**See also:**

FILLET

Intersect



**Command: INTERSECT**

**Alias IN**

**Located:**

1. **Pull-down Menu: Modify - Intersect**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Intersect** is a special case of trimming that causes intersecting lines to be trimmed following the intersection. Essentially this command is the same as issuing the FILLET command with a specified radius of zero.

Enter the command by selecting the option **Intersect** from the menu *Modify* or by typing the command INTERSECT.

Select the two lines to be trimmed. Make the selection by picking, and be sure to make the pick on the segment of each line that is to remain.

---

**See also:**

TRIM

FILLETCommand FILLET

## Lengthen



**Command: LENGTHEN**

**Alias LEN**

**Located:**

1. **Pull-down Menu: Modify - Lengthen**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Lengthen** provides the ability to extend the length of existing lines or arcs.

Unlike the command EXPAND, **Lengthen** extends an entity to a specific value or in a particular ratio (percentage value) of the original size.

Entities will be only lengthened in their existing direction. All entities will always be lengthened on the end that lies closest to the point where the cursor was positioned during the selection.

### Options:

#### Dynamic

Lengthen normally works in the **Dynamic** mode. This mode is active after the selection of a drawing entity and needn't be selected from the Option Bar. After any lengthening operation the mode will return to Dynamic, no matter which of the options has been used.

In the Dynamic mode the line or the arc will be lengthened to a point or to an angle indicated the cursor. A preview will show, during this operation, the result of the command.

In addition to the standard mode Dynamic, the following modes are available for lengthening lines and arcs from the Option Bar:

1. **Total** allows the entity to be lengthened to a dimension that is defined. The dimension will apply to either the length of a line or to the circumference of an arc. To assist the user, the current length of the selected entity is displayed in parentheses. If the option **Angle** been selected the included angle of the arc is displayed instead of the length, and the input should also be the additional angle required.
2. **Percent** will permit the entity to be lengthened by a defined ratio of the original size. This percentage value relates to either line length or the circumference of an arc. For instance, the input of the percent value 200 doubles the length of the line or the arc, the value 50 diminishes it to the half of the original value. If the option **Angle** has been selected, the input of the percentage value will apply to the arcs original angle.
3. **Relative** lengthens a line or an arc by a length that is defined by indicating as distance between two points. This is an advantage particularly if the entity is to be lengthened by a straight line.

## Delete Partial



Command: DELPARTIAL

Alias DPL

#### Located:

1. **Pull-down Menu: Modify – Delete Partial**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Delete Partial** will cause a part of a line, circle, or arc to be erased.

These entities can also be **divided** by this command. The result of such a division will be two independent entities that can be edited, or have another operation performed on it, separately.

#### Deleting Part of a Line

Deleting part of a line is the standard option for this command. to perform this deletion, determine the end points of the part of the line to be deleted. The line will be divided at this point, and the portion of the line between the two points will be deleted.

#### Deleting Part of a Circle or Arc

If the entity is a **circle**, you have only the two-point option available that removes a section of the circle circumference, as a 360o arc is not supported. It must therefore be some angular dimension less than 360o. Arcs, on the other hand, can be either broken into two at a specified point (the one-point option), or else a section can be removed, also leaving two arc objects (the two-point option). With arcs or circles, removal of a section is measured counter anti-clockwise from picked points 1 and 2.

---

**See also:**

BREAK

Break

**Command: BREAK**

**Alias B**

**Located:**

1. **Pull-down Menu: Modify - Break**
2. **Or Enter Command or Alias Directly at the Command Line**

The BREAK command operates identically to the command DELPARTIAL – One Point option. Like this command, the command BREAK breaks objects at a point or points you select. BREAK has been added for convenience of those users who are familiar with other CAD programs.

The main difference is that the command BREAK assumes that you might want to use the pick point when selecting the entity as the first point of the part to delete:

> **BREAK**

Select entity (line, circle, arc, 2D-polyline):

Second point (or F for first point):

However, you can respond to the prompt for the second point the Option *First* (short *F*) to explicitly specify both break points independent from the entity selection:

> **BREAK**

Select entity (line, circle, arc, 2D-polyline):

Second point (or F for first point): **F**

First point:

Second point:

Also, you have the choice to break an object into two pieces without erasing any part of it. To do this, use the command and specify the same point for both the first and the second points; type @ at the request for the second point that signifies the previous point entered. This is useful especially if you want to break an entity at an intersection point with another entity:

> **BREAK**

Select entity (line, circle, arc, 2D-polyline):

Second point (or F for first point): **F**

First point: **int**

of object:

Second point: **@**

---

**See also:**

DELPARTIAL

Rejoin broken Lines or Arcs





## Command: REJOIN

Alias RJ

### Located:

1. **Pull-down Menu: Modify - Rejoin**
2. **Palette: Modify**
3. **Or Enter Command or Alias at the Command Line**

Rejoin will unite separated co-linear lines segments or arc segments that share the same center and the same radius. The resulting joined line or arc will be a single entity.

A common function for this command is to reunite separated line or arc segments that have been divided by the command DELPARTIAL (Delete partial).

Select the command, then pick the entities to be rejoined.

> REJOIN Select first entity (line or arc): *P1*  
Select second entity (line or arc): *P2*

---

### See also:

**Delete partial**

## Modify Polylines



## Command: POLYEDIT

Alias PE

### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Edit Entire Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Polylines may be modified in the following fashion:

- Editing of a Polyline as a whole;
- Filleting of a polyline;
- Chamfering of a polyline; and
- Editing individual polyline segments by editing their control points.

### Polyline Editing

The command allows the user to:

- Open or close the Polyline;
- Join the polyline with other drawing entities such as line, arc or polyline;
- Change the width of the whole polyline; or
- Undo the last step.

The first option panel will display the Option Open or Close, depending on whether the selected polyline is currently opened or closed. Note that the option "Back" is only available after the execution of the first

operation under the command.

### Close / Open

The option **Close** joins the start and end points of the polyline with a straight polyline segment. The width of this segment equals the ending width of the end point of the last drawn polyline.

The option **Open** is a possible option if the polyline to be edited has already been closed during drawing, or in a successive working step. The polyline segment between the start and end points of the polyline will in this case, be deleted.

### Join

**Join** allows lines, arcs and other polylines to be joined with the selected polyline. The prerequisite to this is that the polyline to be edited and the object to be joined meet in one ending point.

After the calling of the option *Join*, indicate the drawing entity to be joined to the previously selected polyline. The polylines chosen are now joined as one entity.

### Width

**Width** offers the opportunity to change the width of the polyline. Polylines that have different start and end widths as single segments, can be made a uniform width for all segments.

Define the desired width of the polylines by numeric input or indicating a width with the cursor. As a "help" parameter for this input, the value used last for the width of the polyline segment to be joined will be displayed. This value may be used by pressing RETURN.

### Undo

**Undo** will remove the last operation. The POLYEDIT command is continued, and the user will see the request "Input:".

---

### See also:

VTXEDIT

If the vertices only are to be edited that option may be picked from the sub-menu, or keyboarded by typing VTXEDIT.

## Fillet



**Command: FILLET**

**Alias F**

### Located:

1. **Pull-down Menu: Modify – Fillet or  
Modify – Edit Polyline – Fillet Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Often two solid edges do not intersect exactly, in acute or right angles. With the Fillet command solid edges can be filleted quickly and precisely.

The first prompt will request the selection the two lines whose intersection is to be filleted. These can also be lines of an object such as a rectangle.

### Options:

The following options will be shown in the Options Bar.

### Polyline

Instead of placing a radius between two separate linear entities as with the standard **FILLET** command, **FILLET-POLYLINE** will apply the radius at all vertices in the selected polyline. Note: this command acts upon a polyline only once. To change the radius you must **UNDO** the first fillet command or else reconstruct the polyline by exploding it, deleting the arcs and **INTERSECT**ing the segments and then using **POLYEDIT-JOIN** to reconnect the segments to form the original polyline.

### Radius

The option **Radius** will define the radius value for the fillet. The program will always offer the value of the last radius used in the current command operation. A radius value of zero means no radius will be applied to the fillet and that entities will simply be trimmed off where they cross each other.

### Trim (Off)

**Trim** functions as a switch. It decides whether the parts of lines that lie beyond the fillet are erased or whether they will be left as separate entities.

If the switch in the Options Bar displays the term **Trim**, you can **switch off** the function **Trim** by clicking the switch. It will then change to **Trim (Off)**.

If the switch in the Options Bar displays **Trim (off)**, by clicking the switch the function will change to **Trim**.

### Chamfer



Command: **CHAMFER**

Alias **CH**

#### Located:

1. **Pull-down Menu: Modify – Chamfer or  
Modify – Edit Polyline – Chamfer Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

When executing the command, the first prompt will ask for the entities to be chamfered. Select those entities by picking them with the cursor.

In the Options Bar, the option **Distance** will appear.

There are two choices: either select, and enter, a distance for the chamfer, or choose to "pick" the distance for the chamfer. The distance back from the point of intersection for the chamfer must be specified for both of the intersecting entities, since there are often non-symmetrical chamfers.

The option **Trim (Off)**, or **Trim**, functions in exactly the same manner as described with the command **Fillet**, as does the option **Polyline**.

The option **Angle** specifies the relative angle that will be applied to the chamfer, a given distance back from the intersection, on object 1.

---

#### See also:

**FILLET**

Command **SPLINE**

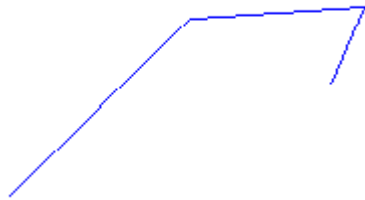
Command: **SPLINE**

Alias **SP**

**Located:**

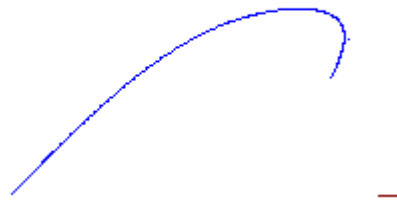
1. **Pull-down Menu: Modify – Edit Polyline – Polyline to B-Spline and Decurve B-Spline**
2. **Or Enter Command or Alias at the Command Line**

The SPLINE command operates on Chains, Polylines and LWPolylines (2-D only), and converts the entity to a B-Spline.



**Changes**

**To**



**Note:** The same command can also be issued to reverse the process. On the pull-down menu, however, it has been made as a separate selection, "Edit Polyline – **De-curve** B-Spline". Issuing the command at the Command prompt is also supported. In the following command sequence, note that the "smoothing factor" is controlled by the "number of segments for approximation" entry. The higher the number input here, the greater the degree of smoothing performed.

```
> SPLINE
Loading d:\Pcad2000\SUPPORT\flx_spln.lsp... OK.
Select polyline to alter to B-spline:
Number of segments for approximation (2 to 1024) <8>: 100
```

**Command SPLINE**

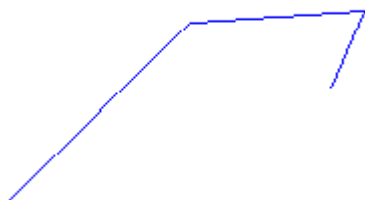
**Command: SPLINE**

**Alias SP**

**Located:**

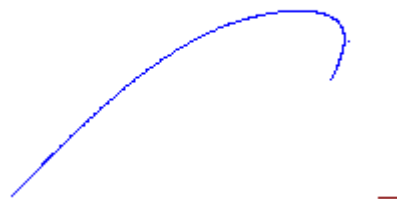
1. **Pull-down Menu: Modify – Edit Polyline – Polyline to B-Spline and Decurve B-Spline**
2. **Or Enter Command or Alias at the Command Line**

The SPLINE command operates on Chains, Polylines and LWPolylines (2-D only), and converts the entity to a B-Spline.



**Changes**

**To**



**Note:** The same command can also be issued to reverse the process. On the pull-down menu, however, it has been made as a separate selection, "Edit Polyline – **De-curve** B-Spline". Issuing the command at the Command prompt is also supported. In the following command sequence, note that the "smoothing factor"

is controlled by the "number of segments for approximation" entry. The higher the number input here, the greater the degree of smoothing performed.

> SPLINE

Loading d:\Pcad2000\SUPPORT\flx\_spln.lsp... OK.

Select polyline to alter to B-spline:

Number of segments for approximation (2 to 1024) <8>: 100

## Vertex Editing



**Command: VTXEDIT**

**Alias VX**

### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Vertex Editing**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Editing of single segments of a polyline is done through the command **Vertex Editing**. This command allows the user to change the vertices (the start and end points) of the polyline. Vertices of a polyline segment can be inserted, deleted, and moved. Also, the width of the polyline segments can be changed.

Select the function by selecting the option Vertex editing in the submenu *Modify Polyline* of the menu *Modify*, or by typing the command VTXEDIT.

### Vertex Selection

Select the polyline containing the vertex to be edited. The selection of the polyline will also cause the selection of the vertex containing the start point as the first vertex to be edited.

The vertex selection may be changed by "stepping" through the vertices using the **Next** and **Previous**. The user should experiment with these selections until he or she understands how they work.

The selected control point will always be marked by a "rubber band" line joining the selected point with the current cursor position.

After selecting the control point to be edited, select one of the editing options from the Options Bar:

### OPTIONS:

#### Insert

The option **Insert** allows the addition of vertices, and thereby of further segments, to an existing polyline.

Define (either by picking with the cursor or by input of coordinates) the position of the additional vertex.

The width of this new segment will correspond to the width values of the polyline, at the bordering control points, before the addition of the new segment.

After inserting a segment the command VTXEDIT stays active. The user may select another operation or complete the use of the command by pressing ESC.

#### Move

The option **Move** enables the user to relocate a vertex of the polyline to a new position.

First determine the vertex to be relocated. Then define (by "picking" with the cursor or by input of coordinates) the new position of the chosen vertex. The vertex will then be moved to this position.

Other segments of the polyline will remain unchanged.

The command will remain active. Finish editing the polyline segments by pressing ESC.

### Delete

Define the vertex to be deleted in the same fashion as was used with the previous options. Then select the option **Delete** from the Options Bar.

The segment having its start point in the selected vertex will be deleted. The previous (in order of drawing) segment will be lengthened to the next control point.

The command VTXEDIT remains active. Select another option or finish editing the polyline segments by pressing ESC.

### Width

Width allows the width of single segments of a polyline group to be changed.

Define the vertex to be deleted in the same fashion as was used with the previous options. Then select the option **Width** from the Options Bar.

Input, or select with the cursor, the desired width of the polyline segment. The width of the selected segment will be changed according to the inputs.

The command VTXEDIT remains active. Select a another operation or finish editing the polyline segments by pressing ESC.

---

### See also:

POLYEDIT

Change



**Command: CHANGE**

Alias CV

### Located:

1. **Pull-down Menu: Modify – Change**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Change** permits the editing of entities including lines, circles, arcs, text, attributes and parts with regard to their geometrical character or their attributes.

The command is called by typing the command CHANGE or by selecting **Change** from the *Modify* menu.

Select the entity to be edited using the functions shown in the Options Bar.

During the changing operation the user may wish to be supported by the preview provided by **Dragmode**. Select this mode from the Options Bar.

The attributes or geometry that can be changed depends on the entity. In the following paragraphs is a summary of the possible operations, listed by the type of entity

### Line

One of the two ending points of the line may be moved to a new position, while the length of the line remains the same.

If the end point of a line is moved without using **Dragmode**, the point to be moved must be the point nearer the selection point. If **Dragmode** is activated, the point lying nearer to the cursor position will be moved. By changing the cursor position the end point of the line to be moved can be changed.

### Circle

**Change** reference geometry can only alter the diameter or radius of a circle. After the object selection, determine a point by either "picking" it, or by coordinate input. The diameter or radius will be enlarged or reduced accordingly.

### Arc

Arcs may be changed as were circles.

### Text

To change text, these attributes, or parameters, may be changed. Existing values are displayed as parameters and can be taken over by pressing RETURN.

- Insertion point of the text object
- Text style
- Text height
- Rotation angle
- Text string (character sequence)

### Attribute definition

These attributes can be changed;

- Inserting point;
- Text style;
- Text height; and
- Rotation angle

These are specific characters of those attributes that may lend themselves to change:

- Name
- Request
- Parameter value

The constant or variable (parameter) value of an attribute can be changed. **Value** is that part of the attribute that is written, together with the part, into the drawing. It may either name the part or contain information about the part. Enter new values (sequence of characters or numeric input).

You find detailed information to the characters of attributes in the section *Parts and Attributes*.

### Part

The user may change the insertion point and/ or the rotation angle of a part. The existing values are displayed as parameters and can be repeated by pressing RETURN.

- Insertion point of a part
- Rotation angle

**Dragmode**, with its dynamic preview capability, is automatically activated during the change of this attribute.

See also: PROPEDIT and CHPROP

Parameters for Modify Commands...



**Command: EDITPAR**

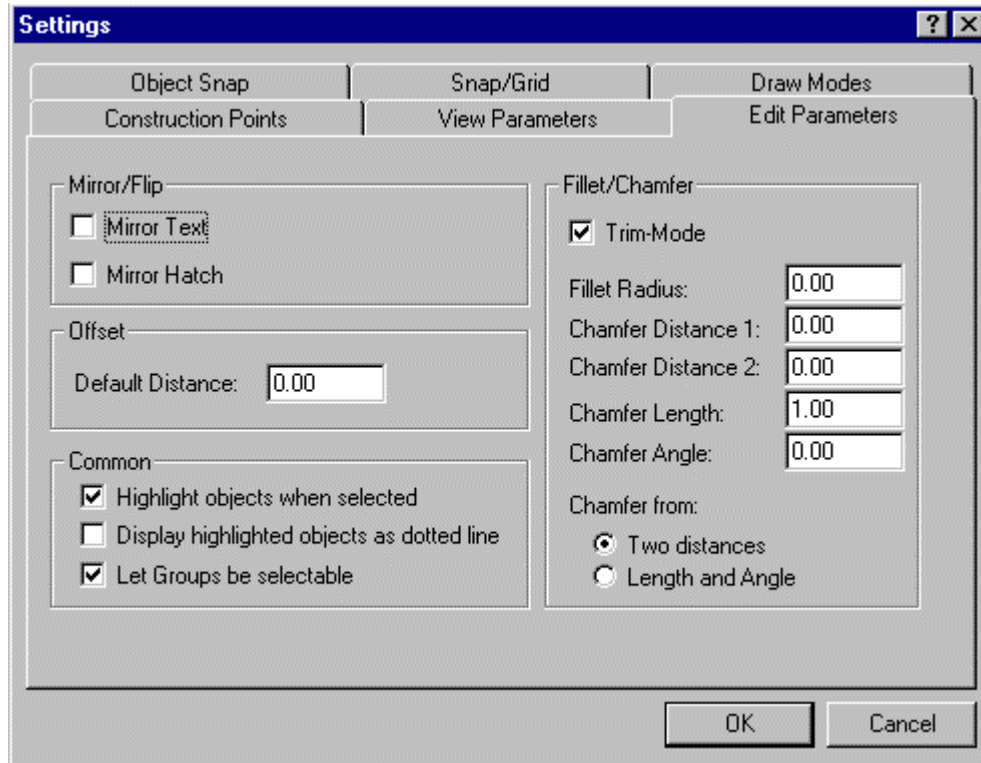
**Alias EP**

#### Located:

1. **Pull-down Menu: Modify – Modify Parameters**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The function *Modify parameters*, allows the user to define a series of parameters that are used as preset values for some editing functions. These parameters are mirroring of text entities, trimming and chamfering, and the radii of fillets.

*Modify parameters* is called by selecting that option from the pull-down menu *Modify* or typing in EDITPAR. A dialog box will appear. This dialog is the Edit Parameters tab dialog for the SETTINGS command. Using EDITPAR or the menu selection, is simply a more direct route to this dialog.



### **Mirror / Flip**

Mirror / Flip determines the position and orientation of text objects in the drawing.

Activate MIRROR / FLIP, by checking the box MIRROR TEXT in that position, from the dialog box *Edit Parameters*.

Activating the option will cause the text objects to be mirrored, with the original text object retained.

With the option de-activated, the text objects will be copied, but not mirrored, about the specified axis.

### **Offset Distance**

*Offset* determines the parameter value for the distance in which an object will be offset when the command OFFSET is used. This value defines the distance of each point of a copy from the original drawing entity. There is more detail on the command OFFSET in this section. Enter the desired value in the input area.

Note that, during a working session the value used last will always be displayed as the parameter value for the next call to the command.

### **Fillet / Chamfer**

This area decides whether the trim function is used by default when these commands are called, or whether a separate TRIM will be required.

### **Trim Mode**



Trim Mode, when activated, will cause the erasure of those parts of objects that lie between, inside, or outside of intersecting points. For more detail see the TRIM command.

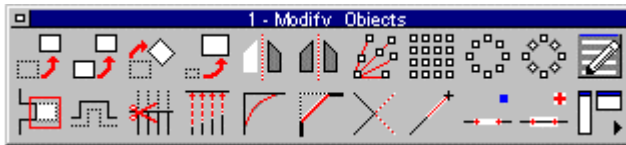
### Fillet Radius

This field allows the user to specify a fillet radius as a default value. Enter the desired value in the input area. Please note thereby that during a working session the value last used will always be displayed as the parameter value for the next call to that command.

### Chamfer Distances

This values establish default values for the chamfer distance from the point of intersection. Each value may be specified independently of the other.

### Modify Palette



This command will place the palette containing Modify command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

### Detail

#### "Detail" Pull-down Menu



The Detail pull-down menu contains command instructions that add text and detail to drawing elements. See the Dim (Dimension) pull-down for adding dimensions to your drawing. For a description of each command, please refer to the specific topics that follow.

Text...



Command: TEXT

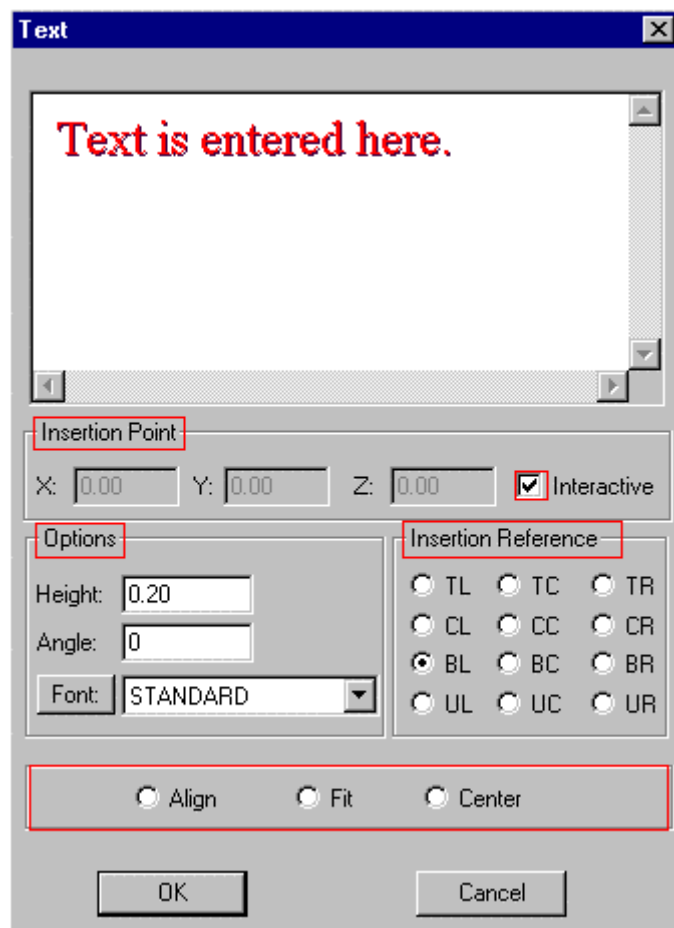
Alias TX

**Located:**

1. **Pull-down Menu: Detail – Text**
2. **Left SideBar**
3. **Palettes: Draw and Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

The TEXT functions are used for inserting text into drawings. They provide labeling, create legends, descriptive notes, and other text elements in a drawing. Most of these commands and options are found under the menu *Detail*.

The command TEXT comes up with a dialog box that allows you to input text and to set several parameters for the text to be added to the drawing:



## Text input

Text is entered into the input field in the upper area of the dialog box, as indicated. The cursor will be positioned in this field upon the opening of the dialog box, so the user may start the input of the text immediately.

The text editor in this dialog box offers the basic functions for editing text input. Incorrectly entered text can be marked and erased just as with a word processor.

When the text entry is completed, the user should make any of the changes that are desirable and within the capabilities of this dialog box. The changes possible are discussed below.

When a line or block of text is complete, press ENTER.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's TEXT command dialog window, either through picking on the window and using Windows' "Ctrl V " function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry

## Insertion point

The Dialog Box offers two methods to define the insertion point. The insertion point may be entered using (absolute) coordinates (boxes are available for input of the X-,Y- and Z-coordinates). The second option is to define the insertion point interactively. The option **Interactive** (shown highlighted) must be checked to activate this option. Unchecking interactive will enable absolute coordinate entry.

## Options:

The following options aid the user in determining the height, the rotation angle, and the font (typeface) of the text object to be inserted.

### Height

Define the size of the text in the *Height* edit box. Enter the desired text height in drawing units.

### Angle

Define the insertion angle of the text object by this input field. Any rotation is possible, and the text can also be aligned to vertical or sloping objects. Parameter value for this is a rotation angle of 0 degrees (the text object is inserted horizontally). For a counter-clockwise rotation, enter a positive value; and for a clockwise rotation, enter a negative value between 0 and 360.

### Font

A window allows the user to define the font. The list of available fonts is displayed in this window. Select the desired font from this list.

## Alignment:

The alignment of the text object(s) can be specified by the following options:

### Insertion Reference

Defining an **Insertion Reference** point places the text in a relative position to the selected insertion point. For this, a reference point on the text object is defined, as described below.

If the reference point, for instance, is situated at the lower left corner of the text object the text will be placed such that the insertion point is at the lower left corner of the text, with text running to the right of the insertion point. If bottom right is selected, the text will lie to the left of the insertion point.

In total there are twelve reference points available for the user's choice. They are represented by the initials of the locating corner (or center) of the text block. UL, for instance, represents the Upper Left corner of the block. Select the position for the reference point by selecting the appropriate radio button.

## Align

Align allows text objects to be rotated and scaled during insertion. The rotation angle and the scaling factor are defined by indicating two points on the drawing. These are the procedures for using Align:

- Activate the option by clicking the box *Align* at the lower left side of the Dialog Box *Text*. The options insertion point, text height, angle and position will not be available any longer. The parameters required for alignment, result from the definition of two points during the insertion of the text object. The reference point of the text object is positioned left justified, lower corner;
- Confirm the use of the option Align by clicking OK;
- Enter the first point for insertion of the text object. This point is interpreted by the program as both the insertion point and the first point for the rotation angle and scaling factor;
- Enter the second point that completes the definition of the rotation angle and the scaling factor, and sets the text height.

## Fit

Fit enables the simultaneous rotation and expansion of a text object during the insertion. The text object is not scaled that is, the text height remains unchanged, while the string is expanded or compressed to fit. The procedure is:

- Activate the Fit option in the dialog box;
- After inputting the text string(s) confirm the settings by clicking OK;
- The prompt will be for the first point - it may be specified or "picked" - do this; then
- The prompt will be for the second point - specify or "pick" this point.

The text will be fit between the two points, rotated so that it follows the line between the points.

## Center

Center enables the user to insert a text object with both vertical and horizontal alignment centered. Text height and rotation angle may be chosen.

For onscreen display of text characters, see TEXTMINDISP.

## Command DTEXT

### Command: DTEXT



Alias DT

#### Located:

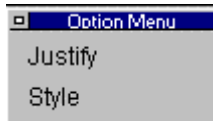
1. **Pull-down Menu: Detail – Single Line Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

Unlike the other text input commands, DTEXT places your text on the drawing as you enter it.

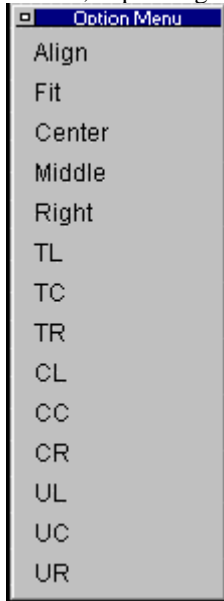
AllenCad's DTEXT command operates essentially the same way that AutoCAD's DTEXT command does. When you issue the command, you are prompted at the command prompt to pick the start point for your text on the drawing.

```
> *** Cancel ***  
> dtext  
Specify start point or select option (Justify, Style):
```

At the same time, you are offered two options:



These will either be offered at the command line as well as in the option menu as shown, or else on the toolbar, depending upon your desktop preferences. Choosing Justify will raise another dialog:



On this dialog you can select the justification you want the text to have.

Choosing Style will raise an input prompt at the command line requesting the required text style name.

Next, you will be prompted to enter the required text height, if other than the current default.

```
> dtext
Specify start point or select option (Justify, Style):
Text height <0.20>: |
```

Note that as with most dimension prompts, you can also respond by picking two points on the drawing that will define the height to apply to the text.

Next, you are asked to input the angle (orientation) at which the text is to run. Again, picking two points on the drawing will establish the text orientation.

```
Specify start point or select option (Justify, Style):
Text height <0.20>:
Insertion angle <0>: |
```

Lastly, you will type in your text. Note that it appears immediately on your drawing as you type. Pressing the Return key will move you to a position one line below, on which to type a second or subsequent lines of text. Pressing Return key twice in a row will terminate text entry and pressing the Esc key will cancel the current command.

Reference Text...



**Command: RTEXT**

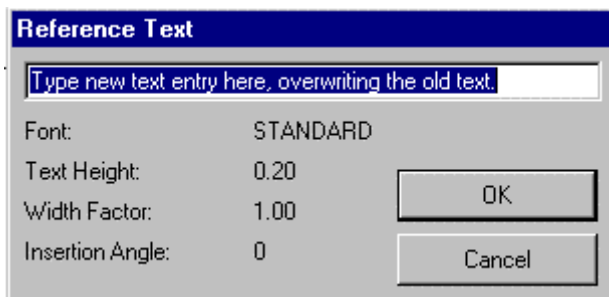
## Alias RX

### Located:

1. **Pull-down Menu:** Detail – Reference Text
2. **Palette:** Detail
3. **Or Enter Command or Alias at the Command Line**

This command will permit you to select regular text entries (not mtext entries) and enter new text with the same characteristics as the text entry selected.

When you issue the command, you will be asked to select the text that you want duplicate the properties of. Once you have done this, the following dialog will appear:



Simply overwrite the text that appears in the dialog box and press OK. Lastly, you will be asked to specify the location of the new text entity in the drawing. Simply drag the text to the location required and click to anchor it. The new text will have the same font, height, width factor, and rotation angle as displayed in the dialog, along with the same layer and color.

The command scheme is:

```
> RTEXT
Select reference text:
<Dialog box to edit text>
Insertion point:
```

---

### See also:

TEXT

## Modify Text...



**Command:** TEXTEDIT

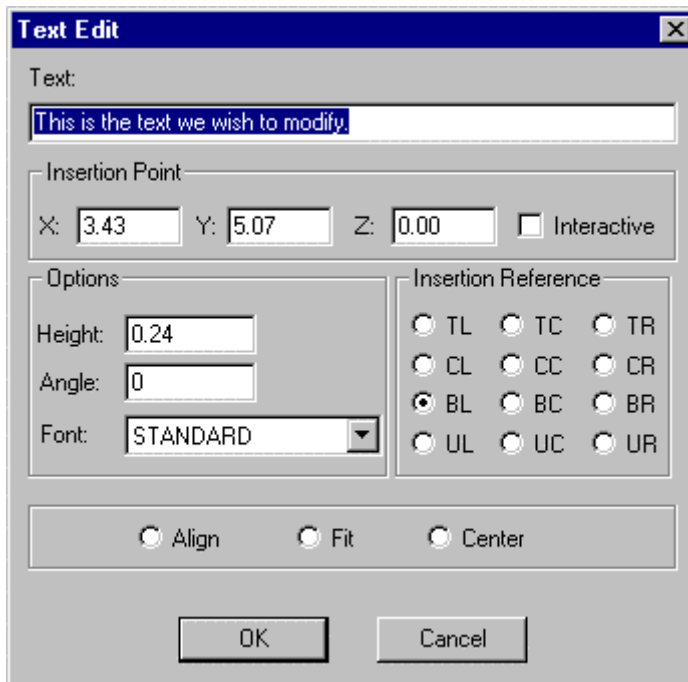
## Alias ED

### Located:

1. **Pull-down Menu:** Detail – Modify Text
2. **Left SideBar**
3. **Palette:** Detail
4. **Or Enter Command or Alias Directly at the Command Line**

You can modify an existing text line in the drawing by selecting **Modify Text...** from the menu *Detail*. Select a text entity in the drawing. The dialog box *Text Edit* is displayed that is similar to the dialog to

create text (with exemption that only one line can be edited).



Simply edit the existing text in the window meet the requirements. The dialog box also presents the parameters attached to the selected text. These parameters can be modified as required, for the replacement text.

If the check box labeled *Interactive* is marked, you will be prompted to relocate the text when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

```
> TEXTEDIT
Select a text:
<Dialog box appears to edit the selected text line>
Insertion point:
```

---

#### See also:

TEXT  
RTEXT  
FONT

Correct Text



**Command: TCCORRECT**

**Alias TR**

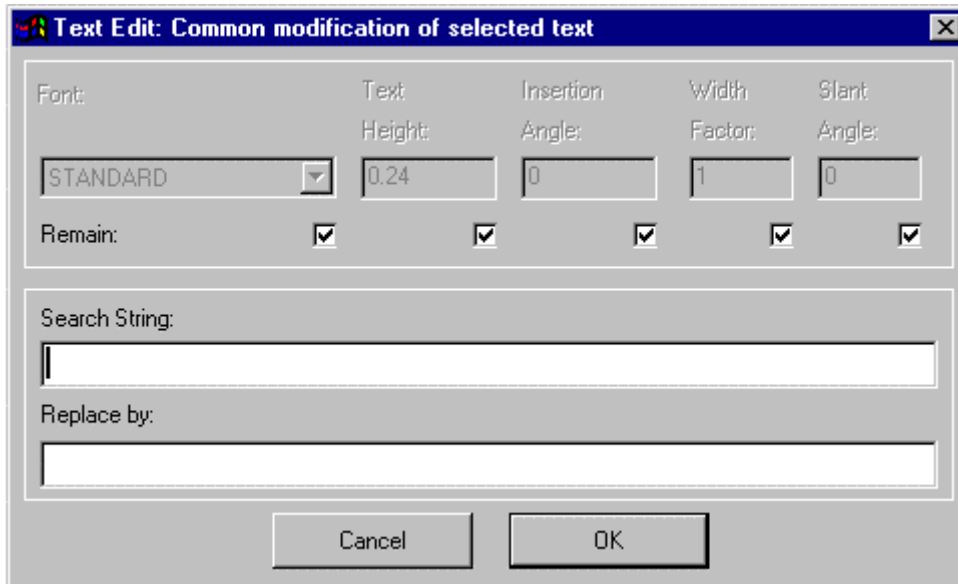
#### Located:

1. **Pull-down Menu: Detail – Correct Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

The function **Correct Text...** allows the user to edit multiple text entities and attribute definitions. It is possible to correct texts or text attributes as well as to change the font or the point size.

Call the function Correct Text by keyboarding TCCORRECT or select Correct Text... from the pull-down menu *Detail*.

First, you are requested to select the text lines or attribute definitions. You may use the usual object select functions. Finish the object selection by ENTER.



As can be seen from the dialog above, this command will replace all instances of an existing text string in the selection set, with a replacement string. Alternately, or in addition, you can change existing text parameters in the selection set by un-checking the appropriate "Remain" box and entering new values.

---

**See also:**

TEXTEDIT

**Font... (Text Style Definition)**



**Command: FONT**

**Alias FT**

**Located:**

- 1. Pull-down Menu: Detail – Fonts**
- 2. Palette: Detail**
- 3. Or Enter Command or Alias Directly at the Command Line**

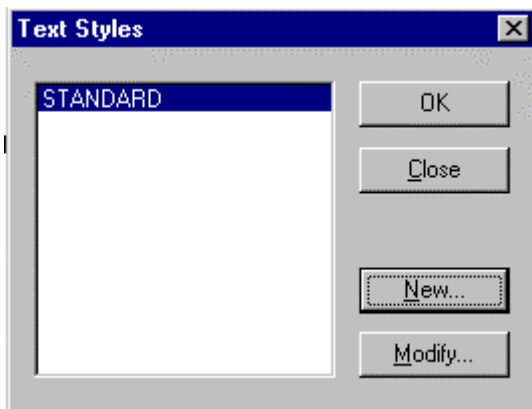
The command FONT loads fonts from the font files and customizes them to the requirements of the drawing.

Fonts define the shapes of the text characters that make up each character set. In AllenCad, you can use:

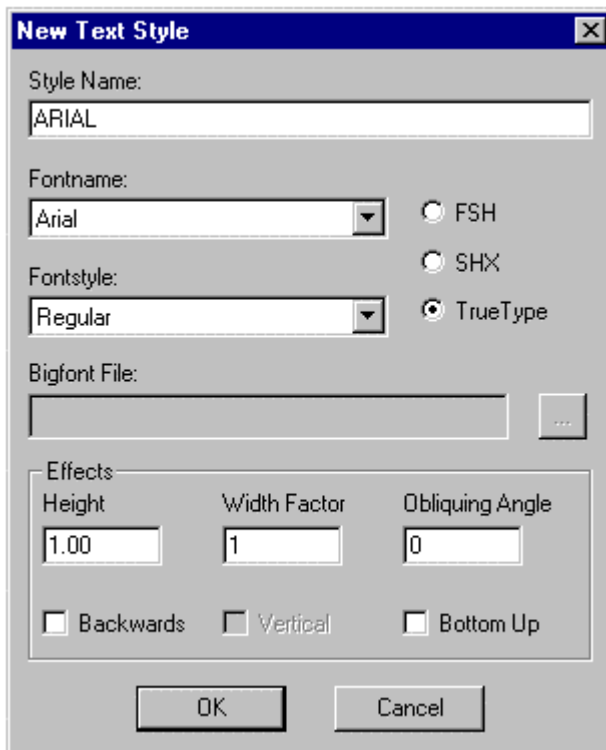
1. AllenCad's own compiled shape (FSH) fonts,
2. AutoCAD's own compiled shape (SHX) fonts
3. True Type fonts.



All installed True -Type fonts, provided with your Windows system, are listed by font family name. SHX fonts and FSH fonts are listed by the name of the file in which the fonts are stored. All fonts to be used in the program must first be assigned to a text style. When you give the FONT command, you are therefore presented with the following dialog:



Here, you create a new text style or choose an existing one to modify, by assigning a different font. With the required style name highlighted, press OK and you will be presented with the font dialog:



In this case, we have chosen to create a new style called Arial and to apply the Arial true type font to this style. To do this, we click on the True Type radio button and select the down arrow on the Fontname box to display the available choices. We scroll to Arial and select it.

Next, we may need to select a fontstyle. Again we click on the down arrow in the fontstyle window and choose regular text, as we do now wish it to be bold, italic or otherwise.

If we wish to use a Bigfont file, we press the browse button to the right of the window and select our

bigfont file from the available choices.

Other text effects are available at the bottom of the dialog. We can assign a text height, width factor or Obliquing (Tilting) Angle. If we want the text to run right to left in a mirror image, we can select the Backwards option. The Bottom Up option will give us text that is upside down. .

**Note:** Some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

Confirm the settings of the properties by selecting OK. Fonts added or changed in this manner can now be used for creating or editing text objects when the modified text style is current.

**Note:** Defined text styles are saved with the drawing file only with a reference to the corresponding font file. They are available any time that the drawing is opened, if the corresponding font file is found in the programs directories.

---

**See also:**

Adding Additional Fonts

Command QFONT

Command TEXT

## Compiling a Source Shape File to a Binary Font File

**Command: FCOMPILE**

**Alias FC**

**Located:**

1. **Pull-down Menu: Detail – Compile Font**
2. **Or Enter Command or Alias Directly at the Command Line**

This program uses binary font files that permit defining text styles in the drawing (see command FONT).

Defined text styles can then be set and used with the commands TEXT, TEXTEDIT, ATTDEF, ATTDEDIT.

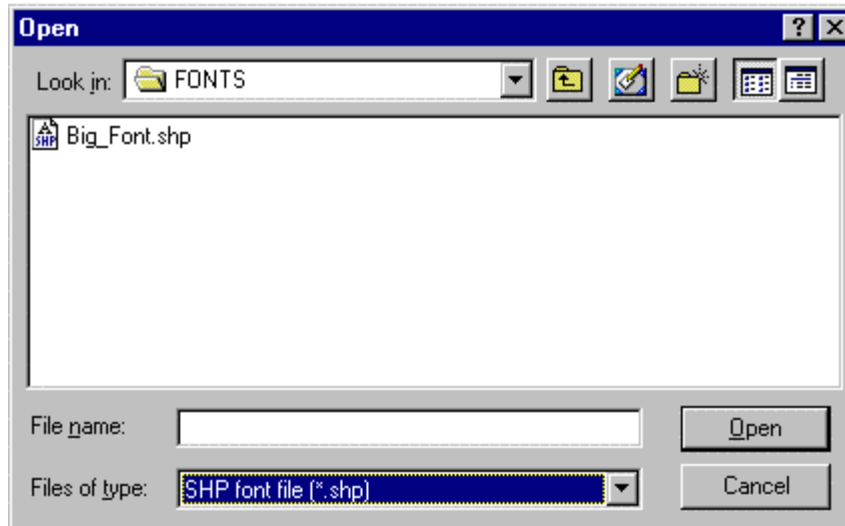
Binary font shape files (*.fsh*) provide less loading time and improved memory usage. A number of these FSH files are delivered with the program.

Application-specific or customized font shapes can be defined in plain text in **.shp** or **.fnt** shape files.

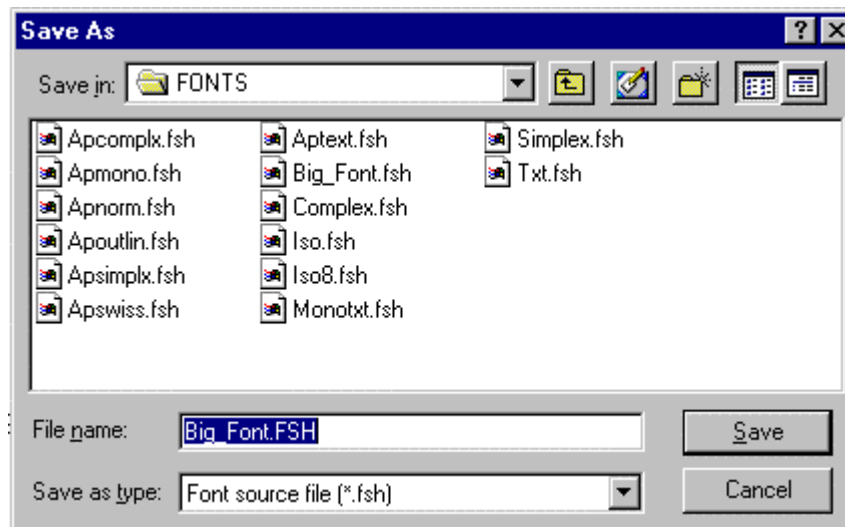
Before they can be used in a drawing, **.shp** or **.fnt** files must first be compiled.

**To convert a source shape file to a binary font file**

1. Call the command FCOMPILE



2. Specify the *.shp* or *.fnt* file name in the file dialog box



3. In the next dialog box, enter the name of the *.fsh* binary font file to be created for later use.

---

**See also:**

FONT  
QFONT

## Command MTEXT



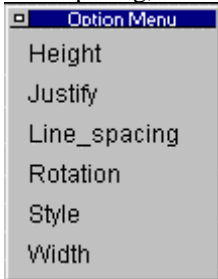
Command MTEXT

Alias MT

**Located:**

1. **Pull-down Menu:** Detail – Multiline Text
2. **Palette:** Detail
3. **Or Enter Command or Alias at the Command Line**

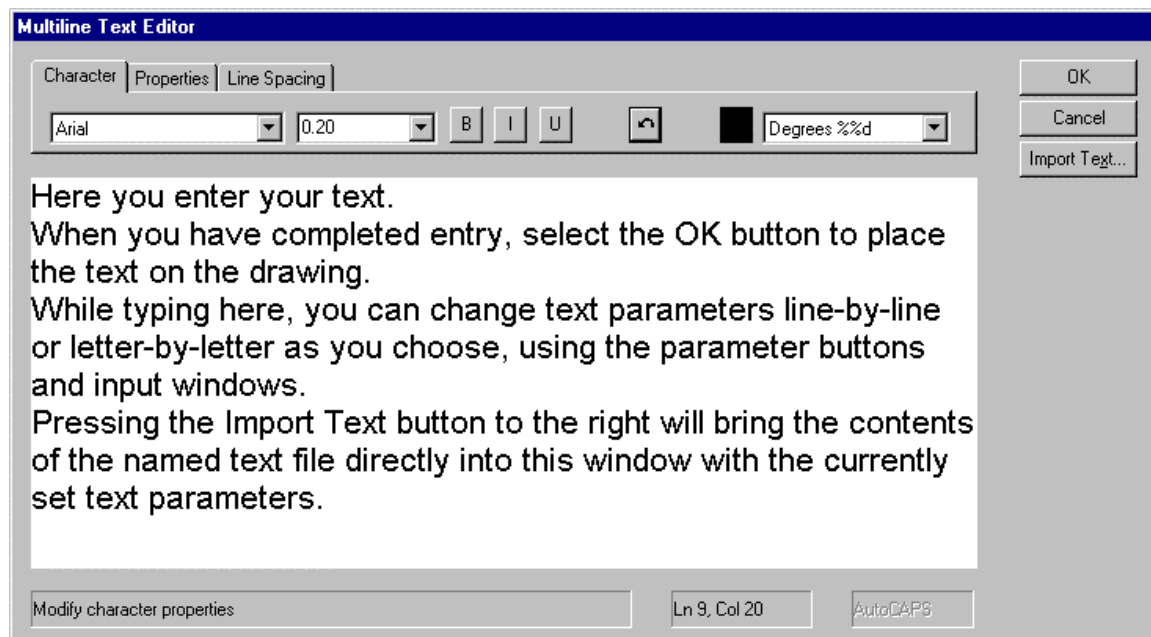
The MTEXT command provides AllenCad with full multiline text functionality. After issuing the command you are asked to specify the location of the new text by picking an upper left and lower right corner on the drawing. This window roughly locates the placement for the text but is essentially used to determine the line length. Text will be wrapped into the next line if the available width is insufficient. While specifying the two window points, you are presented with options to change Height, Justification, Line Spacing, Rotation, Style and Width.



You may choose to set these values here, however these options are also available in the dialog that follows.

Next, you are presented with a windowed dialog box in which you enter your text. Three tabs allow you to exercise control over all text parameters and a **Text File Import function** enables you to import an external text file into the window. These duplicate and add to the earlier options that were presented and you can set your preferences in either manner.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's MTEXT command dialog window, either through picking on the window and using Windows' "Ctrl V " function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry



#### MTEXT Dialog showing Character Options.

**Note:** to allow the text to fill the indicated space in your drawing and to line wrap as needed within that designated width, be sure to continue the line entry beyond the visual extents of the entry window. Pressing the Enter button as we have done in the above example will allow you to see all the text without scrolling left-right, however it may not give the desired effect when this text is transferred to the drawing.

Most of these windows and buttons you are probably fairly familiar with if you do much word processing. Viewed left to right, these windows and buttons:

1. Specifies the required font
2. Toggles between bold and regular type
3. Italicizes text
4. Underlines Text
5. Removes the latest entry or reverses the latest action
6. Indicates the color to be applied to text. Click on this to change the setting
7. The last button allows you to call up special symbols (diameter, degree, plus/minus)



#### MTEXT Dialog showing Property Options.

Here, you can specify a named text Style, set Justification, override text Width originally set using the picked window points, and finally, Rotation, where you can specify an angle or bearing at which to place the entered text.



MTEXT Dialog showing Line Spacing options.

Here you can specify line spacing distance -either a minimum (At Least) or (Exactly)

See Also MTEXTEDIT , QMTEXT

## Command MTEXTEDIT

### Command: MTEXTEDIT



Alias MTE

#### Located:

1. **Pull-down Menu: Detail – Edit Multiline Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

This command predictably is used for editing multi-line text. When the command is issued, you are prompted to select the multi-line text that requires editing. This command will only work on text that has been entered with the MTEXT command. You must use TCORRECT or TEXTEDIT on other text entities.

When you select an MTEXT entity, the full multi-line entity is brought into the MTEXT dialog window, where you can edit any of the parameters that were originally available when the text was first created.

Hatch...



## Command: HATCH

### Alias H

#### Located:

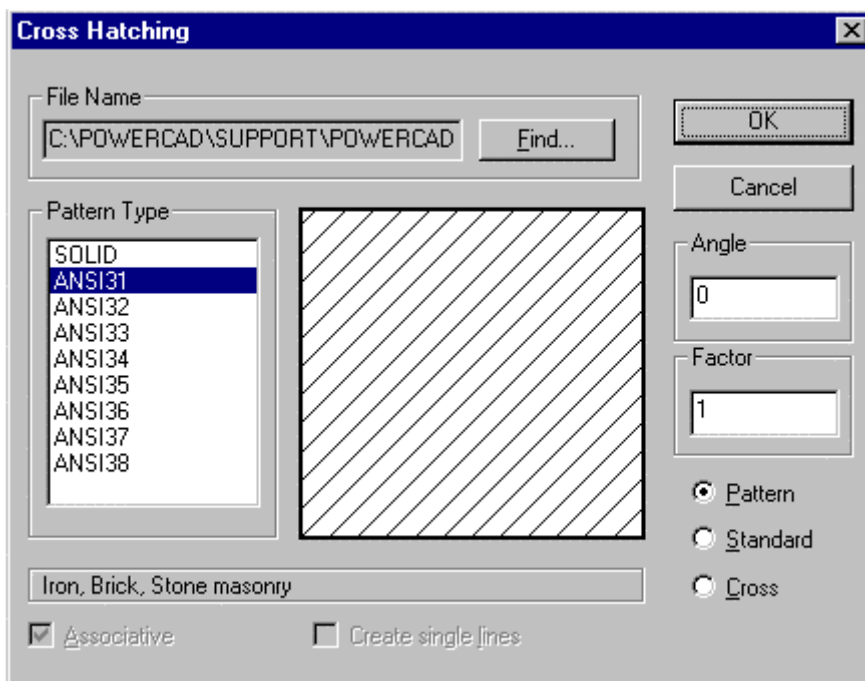
1. Pull-down Menu: Detail – Hatch
2. Left Side Bar
3. Palette: Detail
4. Or Enter Command or Alias Directly at the Command Line

Hatching (or cross-hatching) a drawing adds meaning to the drawing, and helps to differentiate the materials, the areas, or any of a number of other unique attributes. Both the HATCH and the BHATCH commands enable you to fill in areas of your drawing with either a solid color or a distinctive pattern.

#### Command Execution

To execute the command, you are first prompted for the boundary this is typically some closed entity, such as a rectangle, circle or closed polyline. **Note:** If only part of these selected entities forms the boundary, you should instead be using the BHATCH command.

Having selected the boundary, you are next presented with the dialog in which you select the pattern to apply, as well as pattern parameters:



Hatch patterns are stored together in **.pat** files. The file name input at the top of this dialog allows you to select a pattern from any of a number of **.pat** files that you may have available. Having selected the file, all patterns included in that file are listed in the Pattern Type window on the left. Clicking on any of these causes a representation of that pattern to appear in the central window, making selection easier.

The selections to the right on this dialog modify the pattern, in which:

- Angle – Will cause a counter-clockwise rotation of the pattern by the angle entered;

- Factor – Will cause the pattern to become more condensed or expanded, depending upon the entered factor value;
- Pattern will display a non-uniform pattern (if that is available in the particular selection);
- Standard – Will modify the pattern to a uniform spacing and
- Cross – Will create a "criss-cross" pattern effect.

The hatch will be applied immediately after the OK button is pressed.

### Hatch Color

Hatch color is, as with other drawn entities, determined by current layer color setting, or else by a set color that over-rides bylayer color settings. Color fills are achieved by making the selection of "Solid" in the pattern name window.

Further information on hatch pattern definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

---

#### See also:

Boundary Hatching  
 Modify Hatching  
 Polyline Boundary

Create Boundary Polyline for Hatching



**Command: BPOLY**  
**Alias BP**

#### Located:

1. **Pull-down Menu: Draw – Polyline Contour**
2. **Left SideBar**
3. **Palettes: Draw and Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

The command BPOLY creates a boundary polyline from existing objects that form an enclosed area, most often for use with the HATCH or AREA commands.

The original entities can be made up from a combination of lines, arcs, circles, and polylines.

They can be in any arrangement as long as the boundary will make up an enclosed area without looping back upon itself.

Note: The BHATCH command can be used to create its own boundary polyline, while applying a hatch pattern to an enclosed area.

---

#### See also:

Command PCONTOUR

PCONTOUR is the command line version of the BPOLY command.

### Modify Hatching...



**Command: HPEDIT**

**Alias HED**

**Located:**

1. **Pull-down Menu: Detail – Modify Hatching**
2. **Left SideBar**
3. **Palette: Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

You can exchange an exiting associative hatch using this command.

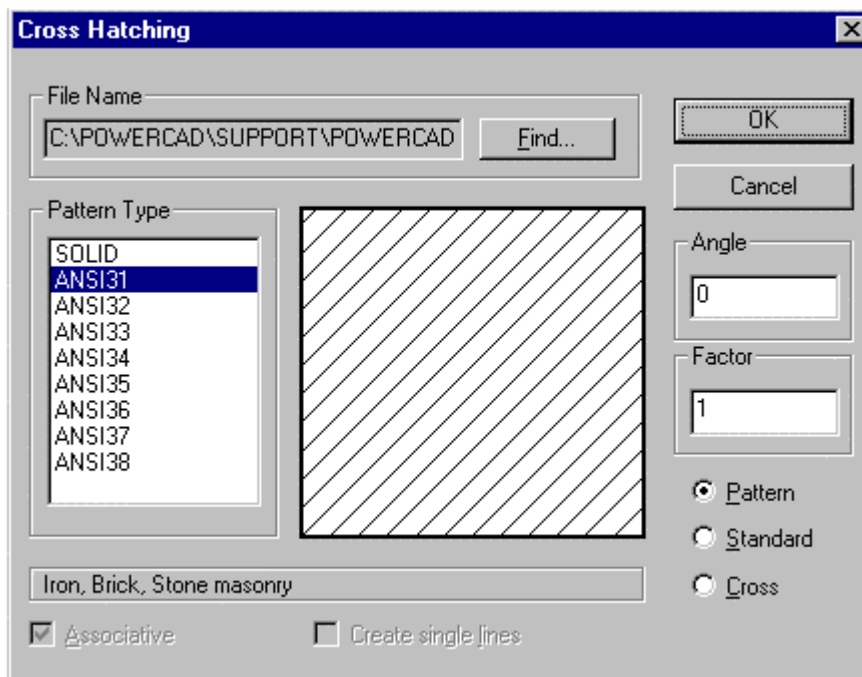
It also allows you to add additional objects to the set of entities to be hatched, as well as change the hatch pattern.

> HPEDIT

Select an associative hatch:

Select additional elements? <N>:

The routine then displays the Hatch Pattern dialog box to select the type of hatching desired.



---

**See also:**

Command QHPEDIT

QHPEDIT is the command line version of the HPEDIT command provided for utilization in Lisp routines, macros, etc.

Command VNOTE



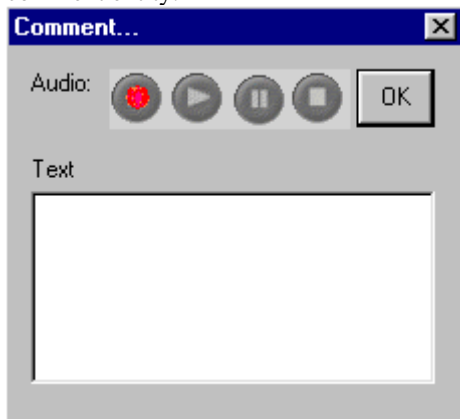


**Command: VNOTE (VoiceNOTE)**  
**Alias VN**

**Located:**

- 5. Draw Menu > MarkUp > Insert/Edit VoiceNote**
- 6. Left Side Bar**
- 7. Palette Draw**
- 8. Enter Command or Command Alias Directly at the Command Line**

VoiceNOTES may be embedded into a drawing by using a feature that attaches audio and text to a special comment entity.



**Adding VoiceNOTES**

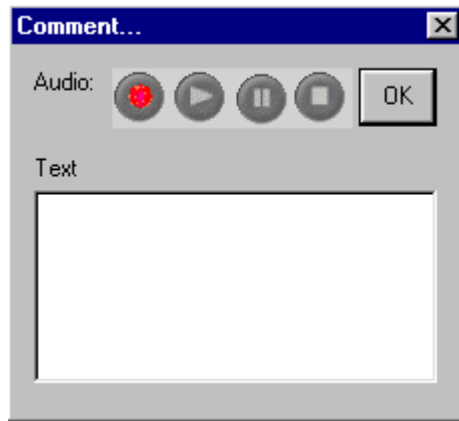
VoiceNOTES can be added to a drawing by selecting Insert/Edit VoiceNOTE from the Draw menu. Select a point in the drawing where you would like to place the VNOTE icon. The dialog box Comment... is then displayed allowing you to enter text and/or to record your voice. Once a VNOTE is entered, select the OK button to place your comment in the drawing.



In the dialog above, simply type your text in the window, or use the recorder buttons to record an audio VoiceNOTE.

**Recording Your Voice**

Note that to record your voice, you will need a microphone, audio hardware and audio software installed and properly configured. To playback your recording, speakers are also required. To record your voice as a VNOTE, either add or retrieve an existing VoicenNOTE icon after issuing the



VNOTE command. This will open the dialog box

In the dialog box Comment... there are four audio controls: record, play, pause and stop. Selecting the record control will record audio until the stop control is pressed. Each new recording completely removes the previous recording. You may playback a recording by selecting the play control. When recording or playing back audio, the pause and stop controls can be used as needed.

### Retrieving or Changing VoiceNOTE Comments

VoiceNOTES can easily be retrieved for reading, listening or also editing. Firstly choose Insert/Edit an Audio Comment from the menu Tools or issue the VNOTE command in any of the above-mentioned locations.

After selecting a VNOTE icon in the drawing, the dialog box Comment...is display with the attached text and/or audio recording. You can read the text message (if there is one) in the dialog window, or you can click on the play button to play an audio recording. Alternately, the text edit box may now be edited or a new speech recording made that overwrites a previous recording. The text or audio recording is updated when the OK button is selected.

### Removing VoiceNOTES

VoiceNOTES can be removed by selecting Delete from the menu Edit.

Select the VNOTE icon containing the comment to be removed. Removing this icon will remove the attached audio and/or text comments.

### Notes:

5. VNOTES are stored in Extended Entity Data format and appended to attributes to the fcomment.flx block in the drawing database and externally in the \support directory.
6. VNOTES are primarily for use when the drawing file is loaded in AllenCad. The drawing file can be saved in DWG or DXF format, without losing the recorded information, however this information is not directly addressable from the AutoCAD program. AutoCAD simply ignores their presence in the drawing file.
7. VNOTES stored externally in the \support directory can be exchanged with any other block. Within the drawing file, they are created with the current layer, color and linetype settings. The scale factor of the inserted block is screensize divided by 9.
8. VNOTES are not limited in size, as additional attributes are added for larger sized recordings..

**Tip:** In complex drawings, placing VoiceNOTES on a unique layer can make them easier to locate, especially when the drawing will be shared with mobile AllenCad CE users.

## Command CALLOUT

### Command CALLOUT

Alias CA

Functional in: AllenCad CE Pro **YES**

## AllenCad CE      YES

### Located:

1. **Draw Menu > Markup> Insert Callout**
2. **Left Side Bar**
3. **Palette Draw**
4. **Enter Command or Alias Directly at the Command Line**

The CALLOUT command offers an easy way to place boxed markup text in your drawing, that has an arrow pointing to a reference object. All of this can be accomplished with a single CALLOUT command.

After issuing the CALLOUT command, you are asked to specify the arrow (point) location and then the other end of the arrow. You are then prompted to insert your text in the dialog box presented, after which the command terminates and the Callout is placed on your drawing.

Callouts are inserted on the current layer in red to conform with redlining convention. Color can be changed as necessary afterwards, using the PROPEDIT command.

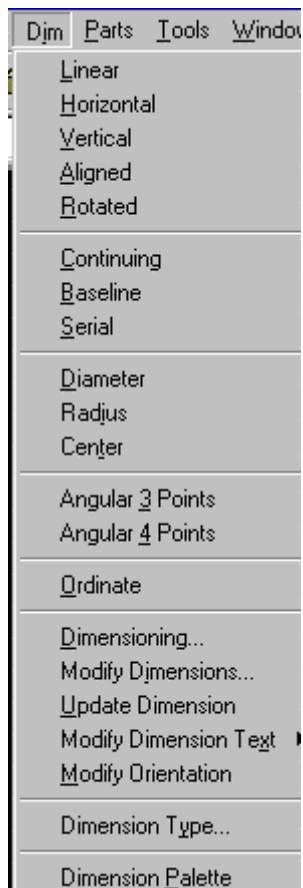
### Detail Palette



This command will place the palette containing Edit command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

### Dim

#### "Dim (Dimension)" Pull-down Menu



The Dimension pull-down menu contains command instructions that add dimensions to drawing elements. For a description of each command, please refer to the specific topics below.

## Linear Dimensioning



**Command: DIMLIN**

**Alias DL**

### **Located:**

1. **Pull-down Menu: Dim – Linear**
2. **Palette: Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

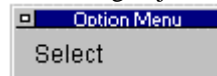


Linear

The DIMLIN command creates a horizontal or vertical dimensioning depending on the position of the dimension line. This command is a combination between both orthogonal modes horizontal and vertical.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points is measured along the X-axis for horizontal and Y-axis for vertical dimensioning and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMLIN
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMLIN
1. Point: Select
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: Return
```

## Horizontal Dimensioning

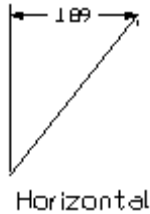


**Command: DIMHOR**

**Alias DH**

### Located:

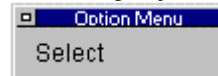
1. **Pull-down Menu: Dim – Horizontal**
2. **Palette: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Draws horizontal dimensions. Horizontal dimensioning is linear dimensioning that both measures and labels the distance between two points parallel to the X -axis.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button . Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the x-axis and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMHOR
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMHOR
1. Point: Select
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: Return
```

## Vertical Dimensioning

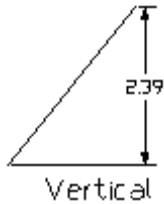


**Command: DIMVER**

**Alias DVR**

**Located:**

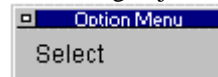
1. **Pull-down Menu: Dim – Vertical**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Vertical dimensioning is linear dimensioning that measures and labels the distance between two points or the linear extension of an object parallel to the Y-axis.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the Y-axis and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMVER
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMHOR
1. Point: SELECT <CR>
select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: Return
```

Command DIMALI



Command DIMALI

Alias DAL

**Located:**

1. **Pull-down Menu: Dim – Aligned**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias at the Command Line**



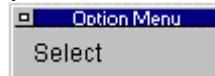
Aligned

Aligned dimensioning is a type of linear dimensioning that measures and labels the absolute distance between two points, regardless of the relative position of the axes. This makes it possible to dimension lines, edges and spaces etc. that are not located on the principal axes of the coordinate system (IE. not parallel to x, y or z-axes).



Aligned

The user will be asked to define two points of measurement (DEFPOINTS). As an alternative the program offers the user the chance to select a specific dimensioning object function. Activate this function by



choosing Select from the options bar. This allows lines, polylines, circles and arcs to be selected. Point to the desired object with the cursor. In the case of lines, polylines and arcs the start and end points of the object will be interpreted automatically and used for the measurement calculations.

The first point of a circle will be defined as a point situated along the perimeter of the circle. The second point will be obtained by drawing a straight line from the first point, through the center point of the circle, to the opposite side of the perimeter of the circle.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line. This can be done either by entering the relevant coordinates, or with the cursor.

The absolute measurement between the two points will be displayed. Pressing the RETURN key will accept the calculated value. The dimension will be inserted into the drawing and shown on a horizontal plain.

> DIMALI

1st Point: P1

2nd Point: P2

Dimension line positioning: P3

Dimension text <2.5000>: RETURN

Or

> DIMALI

st Point: SELECT <CR>

select line, polyline, circle, arc: P1

Dimension line positioning: P2

Dimension text <2.5000>: RETURN

## Rotated Dimensioning



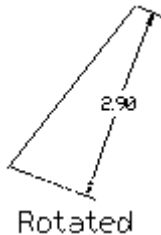
**Command: DIMROT**

**Alias DRO**



### Located:

1. Pull-down Menu: Dim – Rotated
2. Palettes: Create Dimensions and Dim
3. Or Enter Command or Alias Directly at the Command Line



Rotated dimensioning is also a type of linear dimensioning that measures the distance between two points, at a specified angle.

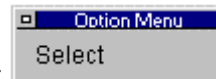
When dimensioning the distance between two points (or an object with linear extension) at an angle greater or less than 90°, the space allocated to the dimension will always be smaller than the absolute dimension measurement. This procedure is equivalent to looking at an object from a specific angle and establishing the visible length from a different viewpoint.

The user will be asked to define the angle at that the dimensioning is to appear in relation to the axial alignment of the coordinate system. This can be done by entering the coordinates at the keyboard, or by using the cursor to click on two points.

The first figure entered defines the angle between the direction of the negative Y-axis and the first dimension line as seen in a clockwise direction. The apex of this angle is determined by the position of the first point of measurement.

If the user defines the angle at that the dimensioning is to take place by selecting two separate points, then a perpendicular line will be drawn at 90° to the extension lines.

After having determined the dimensioning angle, the user will be asked to define two points of measurement (DEFPOINTS). As an alternative, the program offers the user the chance to select a specific



dimensioning object by clicking the Select function from the Options Bar. This will allow lines, polylines, circles and arcs to be selected. Point to the desired object with the selection cursor.

The start and end points of a line, polyline or arced object will be interpreted automatically and used for the measurement calculation.

The first point of a circle will be defined by the program as the next point situated along the circle perimeter. The second point is obtained by drawing a straight line from the first point, through the center intersection of the circle and on to the opposite side of the circle perimeter.

In contrast to objects that have a linear extension, this method will always measure and display the exact diameter of a circle. Only the position of the dimensioning extension lines will be affected when using rotated dimensioning.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line.

The distance between the two points will be displayed at the designated angle orientation. Pressing the RETURN key will accept the calculated value and cause it to be inserted horizontally into the drawing.

```
> DIMROT
Angle of rotation <0>: 60
First Point: P1
```

Second Point: **P2**  
Dimension line positioning: **P3**  
Dimension text <2.50>: **RETURN**

or

> **DIMROT**  
Angle of rotation <0>: **60**  
First Point: **Select**  
Select line, polyline, circle or arc: **P1**  
Dimension line positioning: **P2**  
Dimension text <2.50>: **RETURN**

## Continuing Dimensioning

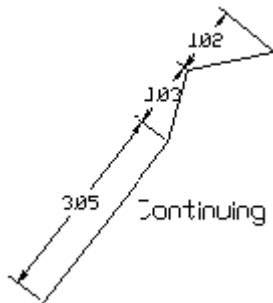


Command: **DIMCON**

Alias **DCN**

### Located:

1. Pull-down Menu: **Dim – Continuing**
2. Palettes: **Create Dimensions and Dim**
3. Or Enter Command or Alias Directly at the Command Line



**Continuing** dimensioning is a variation of linear dimensioning in which existing dimensioning is continued or extended. Here a second dimension is linked to an existing dimension so as to create a dimensioning chain. Continuing dimensions can only be created, when a drawing already contains an existing linear dimension that can be used as a basis for the chain.

A continuing dimension always inherits the same properties as the existing dimensioning. For this reason it is only possible to connect dimensioning of the same type. For instance, a new horizontal dimension can only be added or linked to an existing horizontal dimension.

A message will appear in the text window requesting the selection of an existing dimension. Select the desired dimension, taking care to place the selection cursor as close as possible to the point at that the continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the continuing dimension will be the point that lies closest to the selection point at the time the object is selected

Choose the second point by entering the relevant coordinates, or by selecting with the cursor. Whenever possible, employ the object snap-to function.

The program will display the calculated dimension. This value can either be altered, or confirmed with **RETURN**. The user will not be asked to position the dimensioning line, as this will automatically be placed in line with the existing dimensioning.

> **DIMCON**  
Select a dimension: **P1**

2nd Point: *P2*  
Dimension text <2.50>: *RETURN*

## Baseline Dimensioning

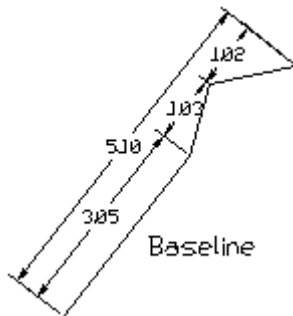


**Command:** DIMBAS

**Alias** DBA

### Located:

1. **Pull-down Menu:** Dim – Baseline
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**



**Baseline** adds a second measurement line to the drawing, parallel to the already existing dimensioning lines. This allows the user to read the total dimension between two end points, extending over a number of individual dimensions within a chain.

This type of dimensioning assumes that a form of linear dimensioning already exists. The appendices dimension will inherit the same properties as the existing measurements.

With the cursor, mark the starting point of the dimensioning chain, taking care to utilize the same direction of measurement as laid down during the creation of the individual dimensions within the chain.

Select the second point to be used for the baseline dimension by entering the coordinates at the keyboard or with the cursor. Whenever possible enlist the help of OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with *RETURN*. The user will not be asked to position the dimensioning line, as this second line will automatically be drawn at a given distance from the existing dimensioning lines.

```
> DIMBAS
Select a dimensioning line: P1
Second Point: P2
Dimensioning text <20.50>: Return
```

## Serial Dimensioning

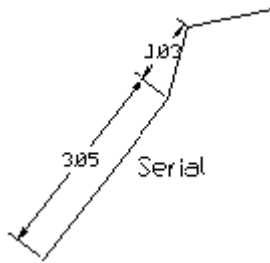


**Command:** DIMSER

**Alias** DSR

### Located:

1. **Pull-down Menu:** Dim – Serial
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**



Serial dimensioning is carried out in the same way as described above for Continuing dimensioning. However, where only one additional dimension can be added to an existing dimension with Continuing dimensioning, Serial dimensioning enables the user to link as many dimensions as required. This assumes that at least one dimension already exists that can be used as a basis for the rest.

It is only possible to add dimensioning of the same type since the additional dimensioning will inherit the properties of the existing dimensioning:

Select the dimensioning that is to act as the basis for the series, taking care to place the selection cursor as close as possible to the point at that the Continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the series dimensioning will be the point that lies closest to the click point at the time of object selection.

In addition to Select and End that are the same for Serial dimensioning as for Continuing, two other selections are possible from within the series dimensioning function bar:

#### **Continue and Baseline.**

1. **Continue** allows the user to link one additional dimension and should be selected when creating serial dimensioning. As this is the default option it only needs to be confirmed with the RETURN key. Alternatively, click the appropriate button as displayed in the options bar.
2. **Baseline** allows the user to create baseline dimensioning, parallel to the existing linear dimensioning. This function is described in the next section. Select the second point by typing in the coordinates or selecting with the cursor. Whenever possible use OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as it will automatically be placed in line with the existing dimensioning.

```
> DIMSER
Select a dimension: P1
Series <continuous> RETURN
2nd Point: P2
Dimension text <2.50>: RETURN
Series <continuous>: RETURN
2nd Point: P2
Dimension text <2.50>: RETURN
...
ESC
```

By re-selecting the **Continue** option, it is possible to repeat the above steps for as many times as dimensions are required. The creation of additional dimensioning can be terminated by pressing the ESC key.

### **Diameter Dimensioning**

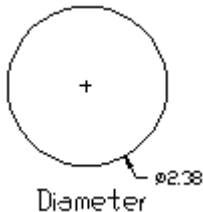


**Command: DIMDIA**

## Alias DDA

### Located:

1. **Pull-down Menu: Dim – Diameter**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Diameter dimensioning is used to display dimensions of circles and arcs. Unlike the horizontal or vertical dimensioning functions, no measurement lines are shown. In this case a single extension line is used to show the relationship between the value and the object itself. The diameter symbol signifies that this is a special dimensioning feature used only in circle and arc dimensioning.

The first prompt will ask for the specific circle or arc. Select with the cursor. Please note that the point at that the user selected the object will be defined as the point at that the dimensioning extension line will connect to the element.

QUADRANT OSNAP will help the user to select specific points on the circle or arc. After selection, the program will display the corresponding diameter measurement. This can be confirmed with the RETURN key or amended as required.

The second prompt will ask for the position of the extension line. Select a coordinate at the required distance from the object

The alignment of the extension line will have been fixed by the position of the two selected points and cannot be altered. The extension line will be drawn along an imaginary straight alignment that extends from the middle point of the object to the measurement point as defined during object selection (DEFPOINT).

The extension line and measurement text will be displayed and a cross (+) will automatically appear at the center of the circle or arc.

```
> DIMDIA
Select circle or arc: P1
Dimensioning text <24.00>: RETURN
Extension length: P2
```

### Radius Dimensioning

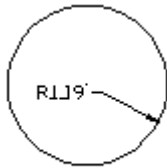


**Command: DIMRAD**

## Alias DRA

### Located:

1. **Pull-down Menu: Dim – Radius**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Radius

Radius dimensioning is carried out in the same way as diameter dimensioning. The only difference is that a radius dimension is calculated instead of a diameter measurement.

```
> DIMRAD
Select circle or arc: P1
Dimensioning text <24.00>: RETURN
Extension length: P2
```

Command DIMCENT

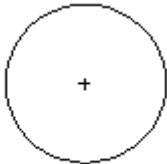


Command: DIMCENT

Alias CM

**Located:**

1. **Pull-down Menu: Dim – Center**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Center

Center marking can only be used in conjunction with circles and arcs. It is used to position a mid point mark (+) at the center of a given object.

The function is carried out in a single step. The user will be asked to select the desired circle or arc. Use the cursor to select the required element.

If a circle or arc is correctly selected the mid point mark will automatically appear. If it is not correctly selected the user will be urged to re-select the object.

```
> DIMCEN
Select circle or arc:
```

Angle Dimensioning (3 Points)



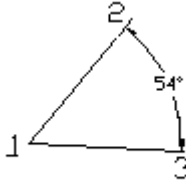
Command: DIMA3P

Alias D3P

**Located:**

1. **Pull-down Menu: Dim – Angular 3 Points**
2. **Palettes: Create Dimensions and Dim**

### 3. Or Enter Command or Alias Directly at the Command Line



Angular- 3-Point

This function uses 3 measurement points that are placed at the apex and legs of an angle. This function not only allows the user to calculate the interior angle between the two legs, but also the exterior angle.

The user will be asked to select the vertex position. Either type in the coordinates, or select with the cursor. It is suggested that the user use the Object Snap options whenever applicable.

Define a point on each of the legs of the angle.

Define the location of the dimensioning arc by selection with the cursor. The dynamic preview function will assist the user in gaining a visual impression of the result. The position chosen for the dimensioning arc will also decide that angle, (interior or exterior) is to be dimensioned.

After confirmation, the program will display the measured value. This can be confirmed with RETURN, or amended as required.

```
> DIMA3P
Identify vertex: end
of object: P1
First point: P2
Second point: P3
Position dimension arc: P4
Dimensioning text <60>: RETURN
```

#### Angle Dimensioning (4 Points)

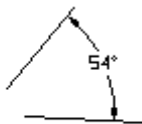


**Command: DIMA4P**

**Alias D4P**

#### Located:

1. **Pull-down Menu: Dim – Angular 4 Points**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Angular 4-Point

The difference between these two angle functions is that the angle 4P dimension is based on four points of measurement. These could be placed at the end points of two lines, or at the tangent of the end points of an arc.

Both the interior and exterior angles can be measured and dimensioned.

The first prompt will ask for the appropriate objects. (lines, polylines, arcs or circles.)

```
> DIMA4P
Select line, polyline or arc: P1
Select line, polyline or arc: P2
Dimensioning text <60°>: RETURN
```

Note: It is not possible to measure the angle between two parallel lines.

### Ordinate Dimensioning

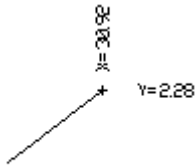


**Command: DIMORD**

**Alias DOD**

#### Located:

1. **Pull-down Menu: Dim – Update Dimension**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



#### Ordinate

Ordinate can be used to determine the X and Y coordinates of a point and enter it into the drawing. These coordinates can either represent a point within the drawing, or be that of a geometrically significant point of a drawing entity.

The X and Y options will be displayed within the options bar. By selecting one of the options the user can determine whether the X or Y coordinate value is to be defined. Click the corresponding button, bearing in mind that the X value is set as the default and will therefore automatically be used if no other option is selected.

The user will then be prompted to select the point to be measured. Type in the coordinate values, or select with the cursor. If the location represents a specific point on an entity, enlist the help of OSNAP.

The program will ask for LEADER LENGTH; this requires that the user enter the EXTENSION LINE position and also determines the physical location of the measurement value text. The extension line originates at the point to be measured and connects to the measurement value text.

Use the cursor to select the position at that the measurement value is to be displayed. This figure can be positioned at any alignment and distance from the dimensioning line. To help, the program draws a line from the selected coordinates to the current cursor position. This illustrates the distance and alignment between the two positions. Select the desired position with the cursor.

If the dimension is not placed horizontally or vertically in relation to the measurement point, then the extension line will appear in orthographic mode and not as a straight line. An example of this is shown in the above diagrams.

The program will display the calculated coordinate. The value can be confirmed by pressing the RETURN key, or amended as required. The user may type in any alphanumeric sequence, thus enabling the point to be titled as required.

```
> DIMORD
```



Coordinate point: **X**  
Coordinate point: **P1**  
Extension length: **P2**  
Dimensioning text <20.00>: **RETURN**

## Command DIMDRAW

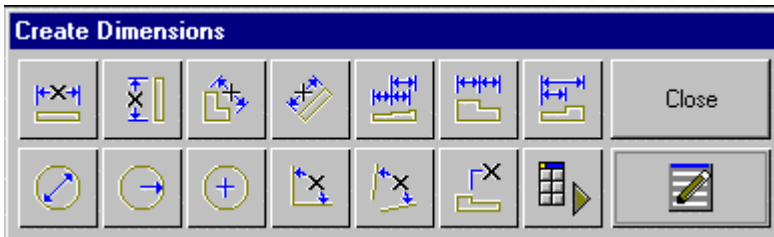


Command DIMDRAW

Alias DDR

### Located:

1. Pull-down Menu: Dim – Dimensioning
2. Left SideBar
3. Palettes: Detail and Dim
4. Or Enter Command or Alias at the Command Line



This command calls the toolbar shown above, which is not unlike the Dimension palette, in function. As with all command icons, holding the cursor above the icon for a second or so, will produce a "tool-tip" that identifies the command function.

Where the Dimension palette remains on the desktop for repeated use, however, the Dimdraw palette is only available for a single command and must be re-called if needed again. Pressing Enter immediately after the palette command has completed, will re-call the Dimdraw palette, however. When placing dimensions on the drawing, you can select the appropriate dimension command by clicking on the associated toolbar icon.

## Command DIMEDIT

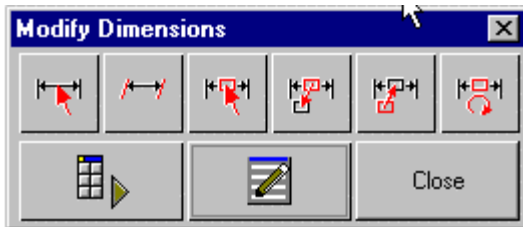


Command DIMEDIT

Alias DED

### Located:

1. Pull-down Menu: Dim – Modify Dimensions
2. Left SideBar
3. Palettes: Modify Dimensions and Dim
4. Or Enter Command or Alias at the Command Line



The DIMEDIT command produces this palette containing icons for commands used to modify dimension entities. The palette remains onscreen for a single selection only and the command is repeated for subsequent modifications.

As with all command icons, holding the cursor above the icon for a second or so, will produce a "tool-tip" that identifies the command function.

## DIMUPD



**Command: DIMUPD**  
**Alias DU**

### Located:

1. **Pull-down Menu: Dim – Diameter**
2. **Palettes: Modify Dimensions and Dim**
3. **Left SideBar**
4. **Or Enter Command or Alias at the Command Line**

After changing the current dimensioning style, all new dimensions will be created in line with the new criteria. Using the Update Dimension function, it is possible to update all of the existing dimensions so that they comply with the new dimensioning style settings.

Using one of the object selection functions found in the options bar, mark or select the dimensions that are to be updated so as to reflect the new dimensioning style.

It is possible to update individual, group or all dimensions contained within a drawing at one time.

Confirm the selection process by pressing RETURN. The marked dimensions will then be updated to conform to the new dimensioning style. The program will display a x dimensions found notice telling the user how many dimensions have been updated.

The following dialog examples demonstrate the selection process and should clarify the use of the DIMUPD command.

> DIMUPD

Select Dimension(s): Fence

First Fence point: P1

Endpoint of the line: P2

Endpoint of the line: P3

RETURN

Select Dimension(s): All

2 Dimensions found

Or

> DIMUPD

Select Dimensions(s): All

Select Dimensions(s): RETURN

3 Dimensions found!

## Command DIMTNEW



**Command: DIMTNEW**

**Alias DN**

### **Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Modify Dim Text Value**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This command allows the user to edit and modify the dimensioning text value.

After calling the function, enter the new dimensioning text value. Select one or several dimensioning objects to be changed.

Press RETURN to close the object selection. The dimensioning text of the selected objects will be modified accordingly.

## Command DIMTMOVE



**Command: DIMTMOVE**

**Alias DTM**

### **Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Move DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to move the dimensioning text of existing dimensioning objects. It doesn't move the associated dimension arrows.

After calling the function, select a dimensioning object. Move the dimensioning text by "pointing" a new insertion point for the text object.

---

### **See also:**

Undo Repositioning

### **Rotate Dimensioning Text**



**Command: DIMTROT**

**Alias DTR**

### **Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Rotate DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

Using this function, it is possible to rotate the dimensioning text of a previously generated dimension.

After calling the function, first enter the rotation angle, then select the one or more dimensioning text objects. Press RETURN to finish the object selection.

The selected dimensioning text will be rotated around its insertion point.

## Undo Repositioning



**Command:** DIMTHOME

**Alias** DTH

### Located:

1. **Pull-down Menu:** Dim – Modify Dimension Text – Realign DimText
2. **Palettes:** Modify Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**

This command undoes the moving of the dimensioning text (done with DIMTMOVE) and returns it to the original position.

After calling the command, select the dimensioning objects that dimensioning text was moved.

Close the object selection by pressing RETURN.

---

### See also:

Move Dimensioning Text

Modify Dimension Orientation



**Command:** DIMOBL

**Alias** DOB

### Located:

1. **Pull-down Menu:** Dim – Modify Orientation
2. **Palettes:** Modify Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to modify the orientation of existing linear dimensions.

Call the command and enter the angle for the dimension orientation. Select one or several dimensions, to be modified in its orientation.

## Dimensioning Styles



**Command:** DIMTYPE

**Alias** DDR

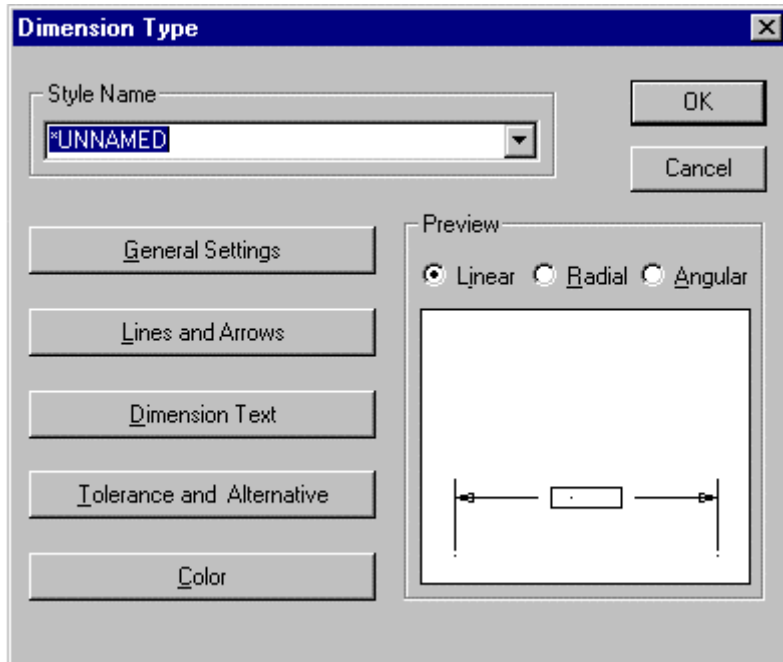
### Located:

1. **Pull-down Menu:** Dim – Dimension Type
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**

Dimensioning format is controlled by the Dimensioning dialog box. Use the style name, input and selection dialog to either store the current settings as a dimensioning style, or to select a pre-defined style as the default.

The buttons described below each open additional dialog windows in which individual dimensioning parameters can be defined.

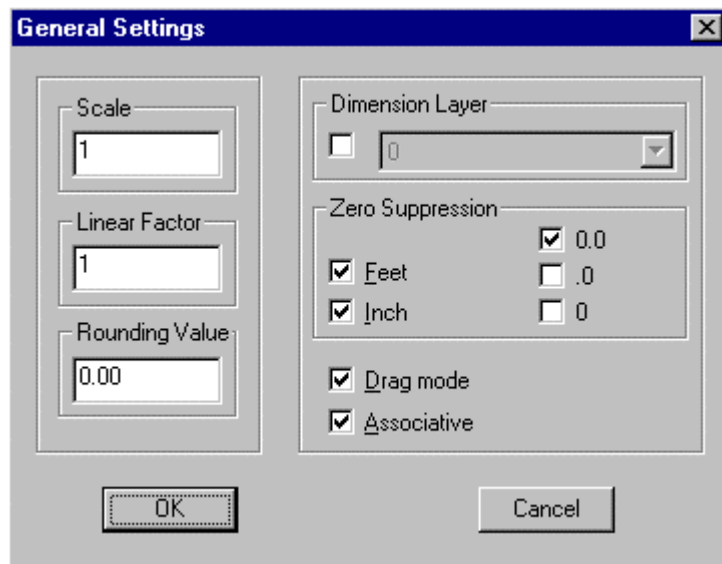
### Dialog Box Options:



When you issue the DIMTYPE command, you are presented with the opening dialog. Here, you see five bars on the left side that lead to specific configuration areas. On the right, you can click on the Linear, Radial and Angular radio buttons, to see a representation of current settings in the window below, for each dimension type.

Clicking on each bar will produce the dialogs detailed below:

#### General Settings



#### Scale

Scale value enlarges or reduces the size of all dimensioning elements (apart from the dimension value itself) and allows the dimensioning elements to be scaled to suit the aspect ratio of the drawing.

#### Linear factor

Linear factor converts the measured drawing units into a dimension figure. The default value is 1.

#### Precision

Precision defines the number of decimal places to which the measurement will be carried.

### Dim. Layer

The layer in which the dimensioning values are stored is identified by the name entered in the dimensioning layer selection field. Select the layer required, or enter a new layer name. It is prerequisite that the option has already been activated by clicking the appropriate control box. A check mark indicates the current status of the option.

### Zero Suppression

The Zero suppression field decides the treatment of the display mode for zeroes.

### Drag mode/ Associative

The last two control boxes activate or deactivate either the "rubber band" mode and / or the associative character of the dimensioning.

### Lines and Arrows

The screenshot shows the 'Lines and Arrows' dialog box. It has a title bar with a close button. The dialog is organized into two main columns. The left column, titled 'Lines', contains three sub-sections. The first, 'Extension Lines', has input fields for 'Distance' (0.06) and 'Extension' (0.18), followed by checkboxes for '1. invisible' and '2. invisible'. The second, 'Dimension Lines', has input fields for 'Increment' (0.38) and 'Extension' (0.00), followed by checkboxes for 'Interior lines' and 'Always interior lines'. The third, 'Center Mark and Lines', has an input field (0.09) and a checkbox for 'Lines'. The right column, titled 'Arrows', has four radio buttons: 'Arrow' (selected), 'Circle', 'Slash', and 'User'. Below these is a 'Blocks' section with a 'Size' input field (0.18) and two radio buttons: 'Same blocks' and 'Seperate blocks'. At the bottom of the dialog are 'OK' and 'Cancel' buttons.

Extension lines are the lines that extend the object dimensions outside the perimeter of the measured object, assuming that dimensioning is to take place outside of these perimeter constraints. These lines, and the nature of the arrowheads to be used, are defined in the Dialog Box *Lines and Arrows*.

### Lines

The fields in this area decide the characteristics of the extension and dimension lines.

#### Extension Lines

This area defines the spacing required for the extension lines.

**Distance** allows the user to choose the space between the object and the point at that the extension lines begin. A value = 0 indicates that the lines will start directly on the object itself.

**Extension** dictates how far the extension lines should extend past the dimensioning line.

The control boxes marked **Invisible** can be used to suppress either one or both of the extension lines in the drawing.

## Dimension Lines

Dimension lines either connect the object edges, or the extension lines of the object that is to be measured. The dimensioning text is written along this line.

**Increment** sets the spacing between successive dimensioning lines that use the same extension lines.

**Extension** allows the user to enter the measurement by that the extension lines should run past the dimensioning lines. This value can only be entered when the **Slash** option has been selected as the dimension arrow type.

The field **Interior Lines** or **Always Interior Lines** decides whether dimensioning lines should be allowed to extend past the extension lines.

## Center Mark and Lines

This area defines the existence and size of a cross-mark indicating a circle center point. It also defines whether or not dashes or strokes should be added to the center point cross or not.

## Arrows

This field allows the user to select the type of arrowhead to be used.

## Blocks

This area describes the length of the arrowhead. Only the size selection of this area is normally active. Block Name is active only when the arrow style selected is User. The size of the Block to be used is defined here.

## Block Name

Active only when the arrow style selected is User, this field decides the Block Name. The name commonly in the field is ARROW.

## Separate Blocks

This field either allows or suppresses the use of divided or separate arrow blocks. When activated, it is also possible to define that elements should be used at either end of the dimensioning line.

## Dimension Text

**Dimension Text**

Text position

☐ Text always inside

☐ Text outside, lines inside

Text always horizontal

☒ inside ☒ outside

Vertical text position

☐ above

☒ relative 0.00

Dimension Text Style

STANDARD

Dimension Text

Height 0.18 Factor 1

Distance to dimension line 0.09 ☐ Frame

Prefix <> Suffix

OK Cancel

These dialog window fields determine the position and look (size, style, spacing) of the dimensioning text.

### Text position, Text always horizontal, and Vertical text position

These fields serve to define the position of the text with respect to the dimension lines.

### Dimension text style, Dim text

These fields define the style and size of the text, and the distance separating text lines.

The prefix and suffix fields enable the user to type in a character string that will appear before or after the dimensioning text. For example, this field may be used to display the unit of measurement.

The < > character decides whether the character string will appear before or after the dimensioning text. For example, **approx. <> ft.** entered in this field would produce the following dimensioning : **approx. 12 ft.**

### Tolerance and Alternative Dimensioning

The screenshot shows a dialog box titled "Tolerances / Alternative Units". It contains several input fields and checkboxes. Under "Tolerance", there are checkboxes for "Variance" and "Limits". Below these are two groups of fields: "Positive Tolerance" and "Negative Tolerance", each with a "Value" field (containing "0") and a "String" field. Under "Alternative Mode", there is a checkbox for "on / off". Below that is the "Alternative Units" section, which includes a "Factor" field (containing "25.40") and a "Decimals" field (containing "2"). At the bottom is an "Alternative String" field. The dialog has "OK" and "Cancel" buttons at the bottom.

The tolerance settings control the tolerance, both the maximum and minimum values, allowed. The control fields activate or deactivate the display of these elements. A check mark in either of the control windows shows the current status of the individual options.

Values for the positive or negative tolerance are entered in the field marked Values. These fields also enable the user to type in a line of text (String).

The **Alternative mode** options control the use of a second, alternative dimensioning method. This mode, when activated, will allow dual dimensioning in metric and imperial values.

### Factor

The conversion factor between the two units of measurement can be entered in the factor field.

### Decimals

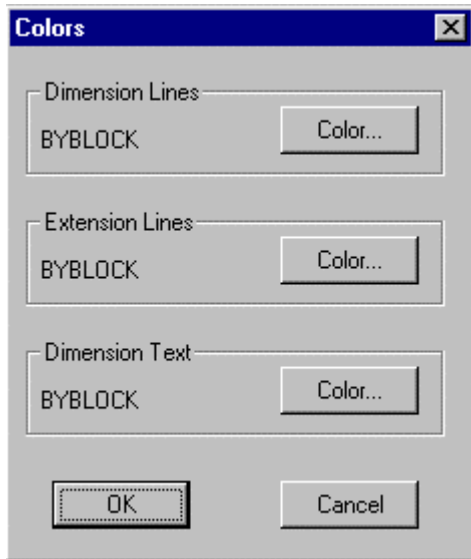
The number of decimal places selected will appear in the decimal display.

### Alternative string

Alternative string should contain a description or title of the second unit of measurement.

### Colors:

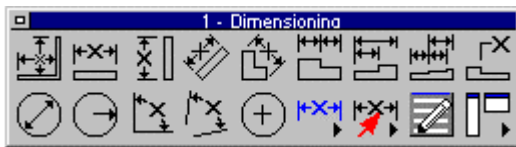




The dialog window allows colors to be assigned to the dimensioning lines, extension lines and dimensioning text (dimension value, alternative dimensioning, tolerances, limit values, unit of measurement etc.)

Select the appropriate element and allocate a color from the displayed selection box. 256 standard colors are available in addition to the BYLAYER and BYBLOCK options.

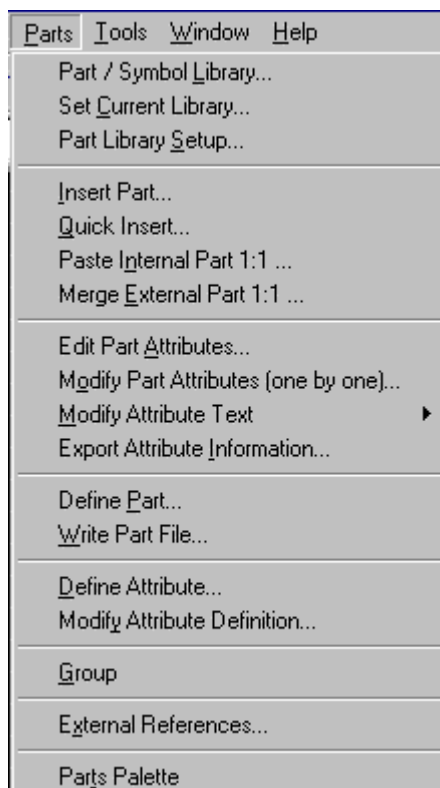
### Dimension Palette



This command will place the palette containing Dimension command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

### Parts

#### "Parts" Pull-down Menu



The Parts pull-down menu contains command instructions that relate to inserting other drawing files (called Parts or Blocks) into your drawing and subsequent Part manipulation. The menu also contains similar commands dealing with external drawing files (External References or Xrefs) that are linked to the current drawing. For a description of each command, please refer to the specific topics below.

## Part / Symbol Library...



**Command: PARTLIB**

**Alias LIB**

### Located:

#### 1. Enter Command or Alias Directly at the Command Line

The command PARTLIB in the menu *Parts* allows the user to manage drawing files such as project drawings, template drawings, part files, or symbol files.

A library is defined by a parameter file (of file type **.plb**) and a set of drawing files, assembled in a directory where the library parameter file is located.

The command PARTLIB called with the name of a parameter file (for example, c:\AllenCad\drawing.plb) will display a dialog that allows you to select a drawing to be inserted. The advantage of the command is that it displays all drawings with a preview bitmap, with verbal information in a list box.

Within the dialog box the user will see the preview of the drawings and information about the drawing.

There are two modes for the list box:

**1.Info Mode** the drawings are displayed together with the verbal information.

2. In **Quick Mode** the drawings are displayed only with their preview bitmap.

The other buttons of the dialog have the following functions:

- **Close:** Closes a dialog without further action
- **File Info:** Opens a dialog to display and to change the file information.
- **Open:** This opens the selected file as separate drawing.
- **Insert:** This button lets the user insert the drawing into your current drawing.

---

#### See also:

Setup and Manage Part Libraries

This section describes how to create PLB files to customize the PARTLIB command to personal needs.

#### Command SETLIB



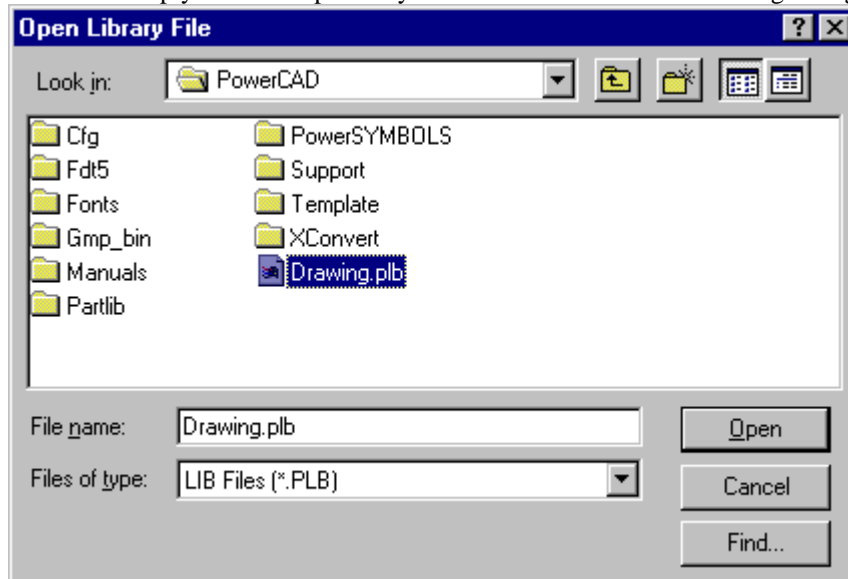
**Command: SETLIB**  
**Alias STB**

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Set Default Part Library**  
**Also - Parts – Set Current Library**
2. **Palette: Parts**
3. **Or Enter Command or Alias at the Command Line**

The LIBRARY command displays the Set Current Part Library / Symbol Library file dialog box. Select a part library (.plb) file.

The **SETLIB** command, on the other hand, sets the default library to be loaded whenever LIBRARY is executed. Simply select the .plb file you want as default in the following dialog:



Alternately, enter the PARTLIB command, which prompts you for the name of a part library at the command prompt. This command is meant for use with macros and FLISP functions. Enter the name of a .plb file, such as:

```
> partlib  
File name: c:\AllenCad\partlib\partlibdrawing.plb
```

Once a part library has been chosen, the Library command continues to use that same library as default. Use the **SETLIB** (Part > Set Current Library) to change the part library.

## Part Library Setup...



**Command: PLBSETUP**  
**Alias PLB**

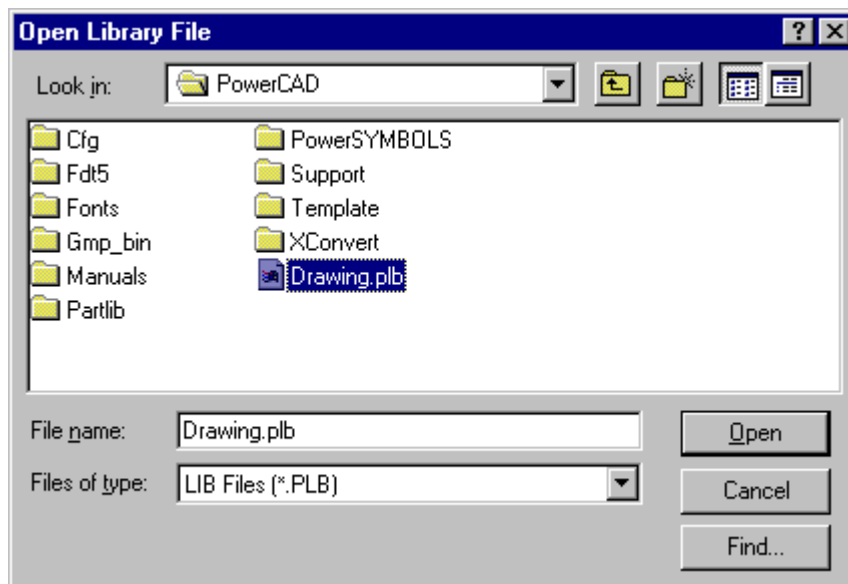
### Located:

1. **Pull-down Menu: Tools – Resource Manager – Part Library Setup**  
**Also - Parts – Part Library Setup**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

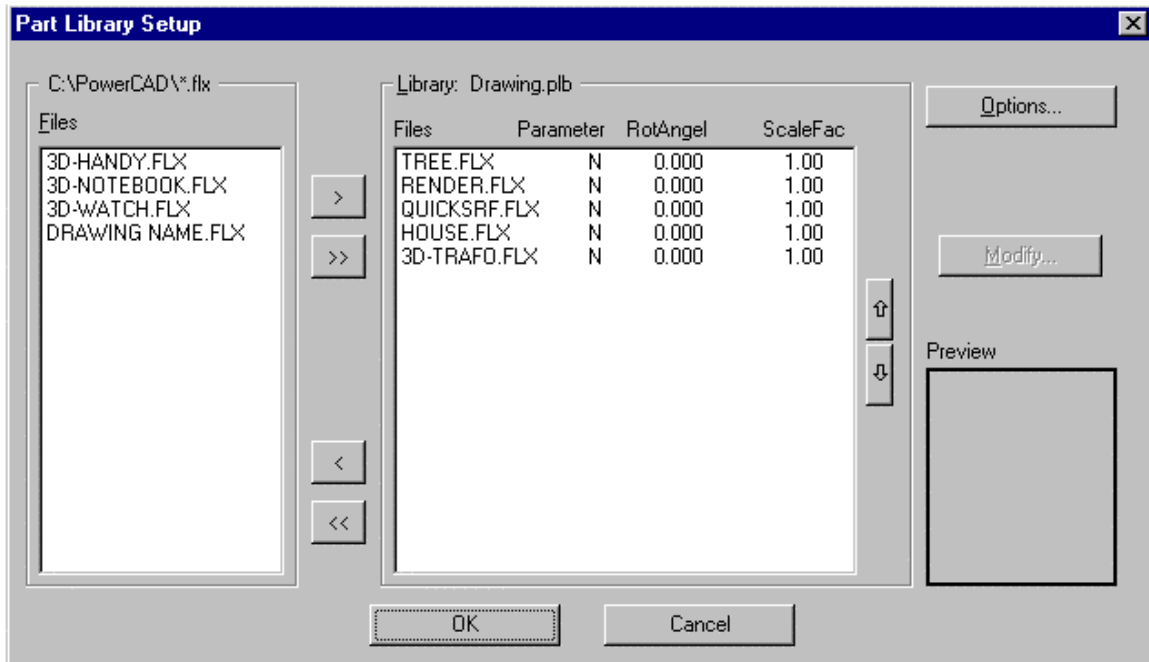
The command PLBSETUP (Part Library Setup) allows the user to setup up new part libraries and/or modify existing part libraries.

The command provides dialog boxes that:

- Select the drawing files to be included in a library;
- Set preferences for part or symbol insertions from the library; and
- Set the parameters that determine the layout and options of the part library Dialog Box.



PLBSETUP creates **.plb** files. These files store the parameters for a single part library. A part library is called by using the command PARTLIB, and then specifying the name of a .PLB file in the above dialog. No knowledge of structure or syntax of the .PLB format is required, when using the PLBSETUP command. Pressing the Open button opens up the next dialog:



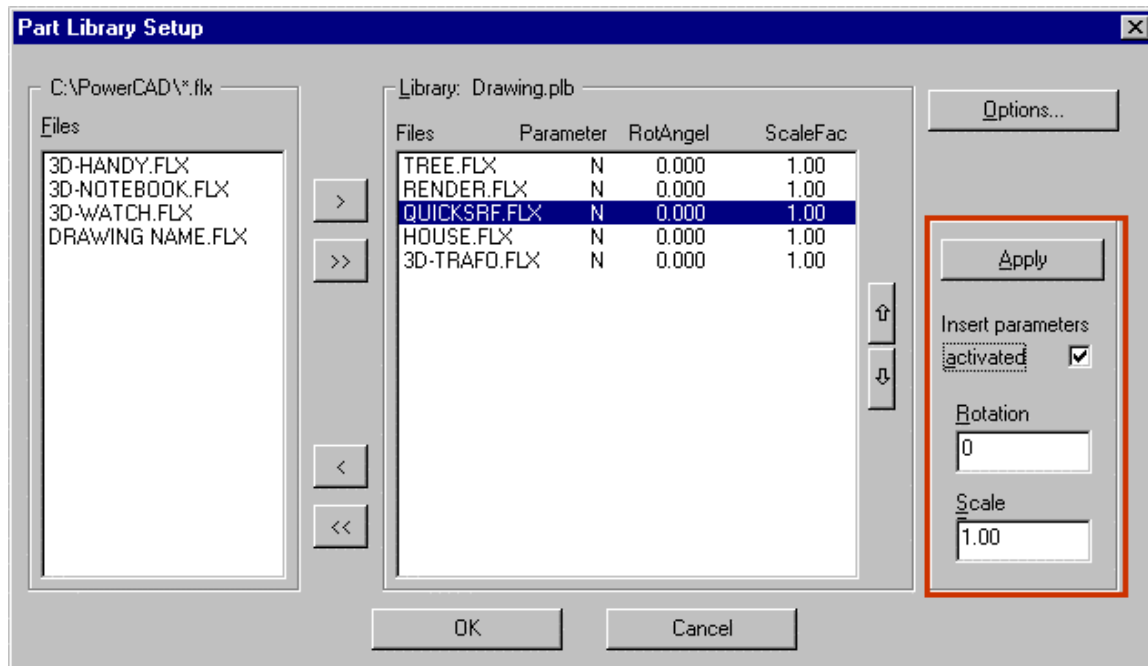
### Specifying the Drawing Files Belonging to a Part Library

The dialog box *Part Library Setup* allows the user to specify the drawing files to be incorporated into the individual library. He or she can **only choose drawing files from the programs Partlib subdirectory**. It is not necessary to include all of the drawings in a folder – simply select those that you want for the specific library that you are creating. Libraries allow the user to maintain all of the drawings used for part or symbol insertions within one directory, and at the same time permits the creation of multiple "thematically separated" libraries.

To add drawings to the current library, highlight the file names in the left side file list of the dialog box and then click to the button > to insert them to the file list making up the library (shown in the right frame). The buttons ( < and > ) permit both addition and removal of drawing files from the list of part library files. When only one specific file is selected from either of the list boxes, the preview image of the drawing will be displayed in the Preview Area. The buttons >> and << allow the user to easily move all file names from one list to the other.

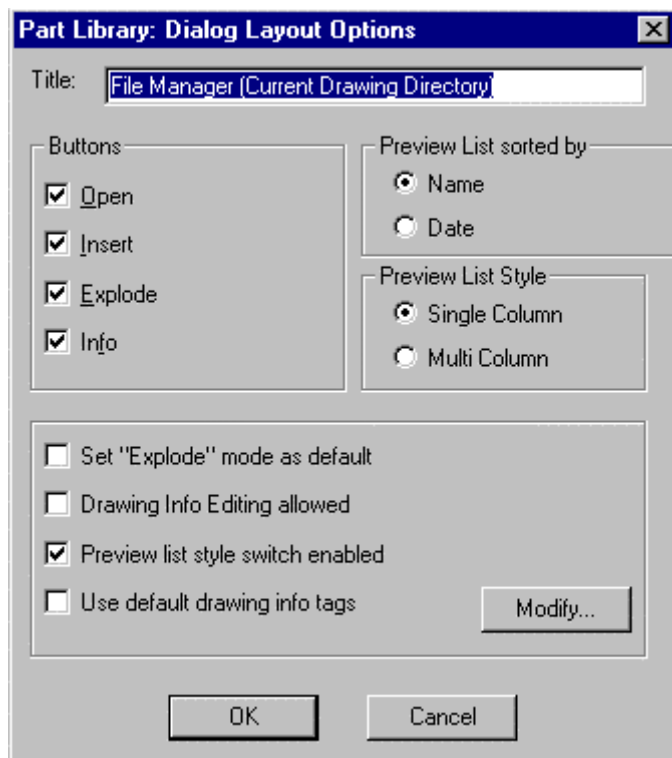
### Insertion Parameters

It is possible to pre-define insertion parameters for the files added to a library. Select one, more than one, or all entries from the list of the library files and then select **Modify**. Next, activate the option **Activate insertion parameters** and enter the default values of the *Scale Factor* and the *Rotation Angle* for insertion in the section outlined in red, below.



### Part Library Dialog Box Layout and Options

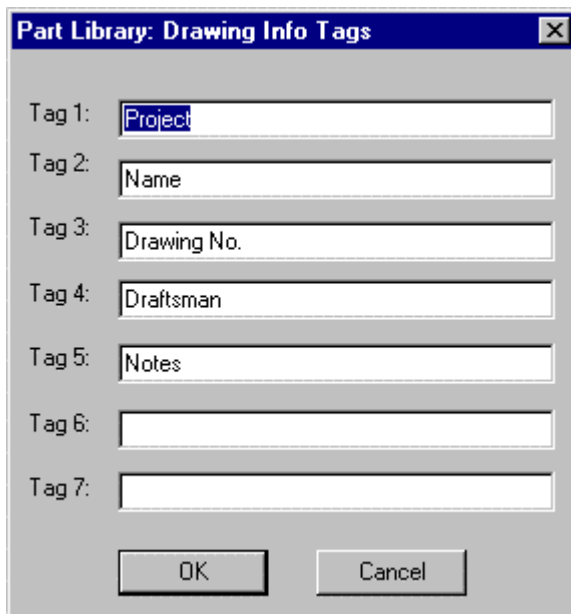
Using the Part Library **Options** dialog box, the user can determine the layout and the available buttons and options of the dialog box that comes up when calling this individual library with the PARTLIB command:



The edit box **Title** allows the user to enter a name for the dialog box of the part library.

### Preferences for the Preview List

- **Preview List Style** lets the user specify whether the part library Dialog Box should start up with a single column or multi column style of the list showing the drawing previews)
- **Sort by...** helps the user determine whether the drawing preview list should be sorted by name (usually determined for part libraries) or by date (usually determined for project libraries).
- **Buttons** assists the user in deciding options provided in the Dialog Box for a specific Part Library.
- **Open** allows the user to open a selected drawing for modification.
- **Insert** allows the user to insert a selected drawing into the current drawing as part.
- **Explode** allows the user to specify whether the part should be inserted as complex object (block), or exploded (broken down to single entities).
- **Info** allows the user to open the Drawing Information sub-dialog of the specific library.
- **Set "Explode" mode as default** sets the preference (activated/deactivated) for the mode of the button Explode.
- **Allow drawing info editing** specifies whether the user may alter drawing file information stored with the drawing database.
- **Enable preview list style switch** determines whether the library Dialog Box should contain the Quick Mode / Extended Mode button that, in turn, allows the user to toggle between single column and multi column style in the file preview list.
- **Use default info tags** determines whether default info tags (Project, Name, Drawing No., Draftsman, Notes) or user defined info tags are used when drawing information is displaced in the Drawing Information Dialog Box of the PARTLIB command.
- **Modify**, in the preferences area, allows the user to determine individual tags for the file info fields used in the specific library (for example; manufacturer, price, etc. in a library managing purchased parts). The dialog that appears with the Modify option is shown below:



---

See also:

Part /Symbol Libraries

### **Insert Part ...**



**Command: INSERT**

**Alias I**

#### **Located:**

1. **Pull-down Menu: Parts – Insert Part**
2. **Palettes: Draw and Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

**Command: QINSERT**

**Alias QI**

#### **Located:**

1. **Pull-down Menu: Parts – Quick Insert Part**
2. **Or Enter Command or Alias Directly at the Command Line**

The command INSERT offers broad support for the user in its Dialog Box. The command QINSERT is the command line version of the command. The command macro QINSERT ? allows fast insertion of internal parts (parts already defined in the drawing).

The command INSERT provides selection areas for:

- Selection of external parts;
- Definition of the insertion point, scale factors, and rotation angle, with future choices of using coordinate/ numeric input or cursor selection;
- Parameter values may be reset; and
- The Part may be "exploded" into its components.

#### **Selecting a Part**

To select the Part to be inserted, type the name of the Part (or select an existing Part) in the field Part Name. Use the buttons External and Internal to select the correct Part listing.

#### **Defining the Insertion Point**

Defining of the Insertion Point either by entering the proper coordinate values, or by selecting Interactive, and "picking" the point with the cursor.

#### **Scale Factor**

The Scale Factor input areas will offer 1.0 as the default XYZ scale factor. This will insert a part at its original size. Remember that a scale factor greater than 1.0 will increase the size of the part, and a factor less than 1.0 will reduce the size of the part to the percentage of the original size specified by the Factor. A factor of 2 will result in the part being twice it's original size, while a factor of 0.5 will be half original size.

By default, AllenCad prompts you for x and y scale factors only. If you need to specify a z scale factor, simply respond to the X scale factor prompt with an xyz response. You will then be prompted for all three scale factors in turn.

The option *Interactive* allows the user to define the size of the Part during the insertion as well. To



insert the Part using the Interactive option, simply define by opposite corners the rectangle into which the Part is to fit.

Since dragmode is active during this operation, the user will have the benefit of seeing the selection results before confirming the selection.

### **Rotation Angle**

The Rotation Angle can be set from the Dialog Box. Enter a value for the Rotation Angle in the proper input window. For counter-clockwise rotation enter a positive value, for clockwise rotation enter a negative value between 0 and 360 degrees

### **Default**

Default allows the user to reset the parameters for the Insertion Point, the Scaling Factor and the Rotation Angle back to the original parameter values.

### **Exploded**

Exploded allows the user to "explode" the part into its component entities. Exploding will cause the loss of the characteristics that were retained as a part.

### **Troubleshooting**

If you find that the blocks you are inserting are not placed in the specified location, check the settings for INSBASE and ELEVATION in both drawing files. These should be set to 0,0,0 and 0 respectively, if you do not want them to affect your block insertion.

## **Command INSERTQ**

### **Command: INSERTQ**

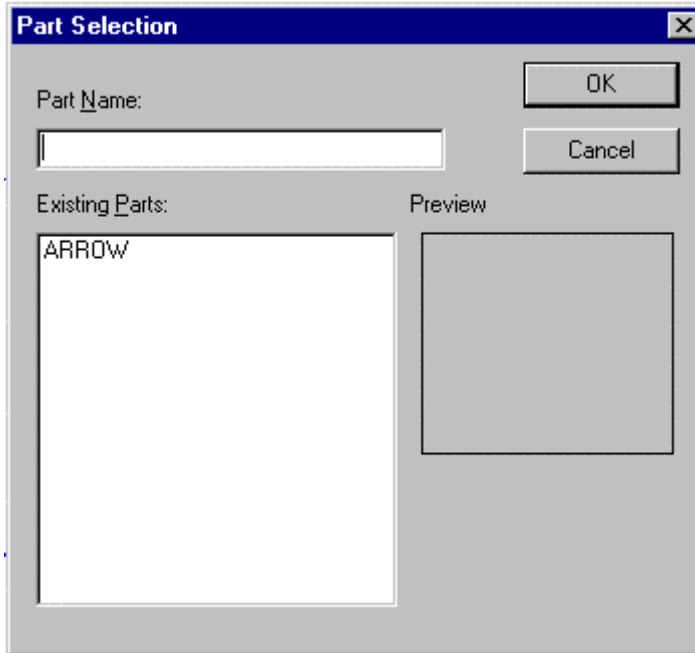
#### **Located:**

- 1. Pull-down Menu: Parts – Paste Internal Part 1:1**
- 2. Or Enter Command or Alias at the Command Line**

The INSERTQ command is used to insert a part into the drawing that has already been defined in the drawing database. In other words, it has been inserted or defined beforehand and is a known entity.

The command can be issued from the "Parts – Paste Internal Part 1:1..." pull-down menu, or entered at the command line, as you prefer.

When executed, the command brings up the following dialog:



Here, you will see in the lower left window, a listing of all predefined parts available for insertion. Clicking on one of them produces an image of the part in the right window, for confirmation. When the part to insert has been correctly identified in the Part Name window, pressing the OK button will begin the insertion routine. You are asked to identify the insertion point, however you are also given a dialog in which you can specify a scale factor or rotation angle other than the defaults as recorded in the Part definition. Unless these options are selected, the insertion point terminates the command with the part being inserted at the default 1:1 scale and in the original orientation.

### **Troubleshooting**

If you find that the blocks you are inserting are not placed in the specified location, check the settings for INSBASE and ELEVATION in both drawing files. These should be set to 0,0,0 and 0 respectively, if you do not want them to affect your block insertion.

## **Command MERGE**

**Command:** MERGE

**Alias** MG

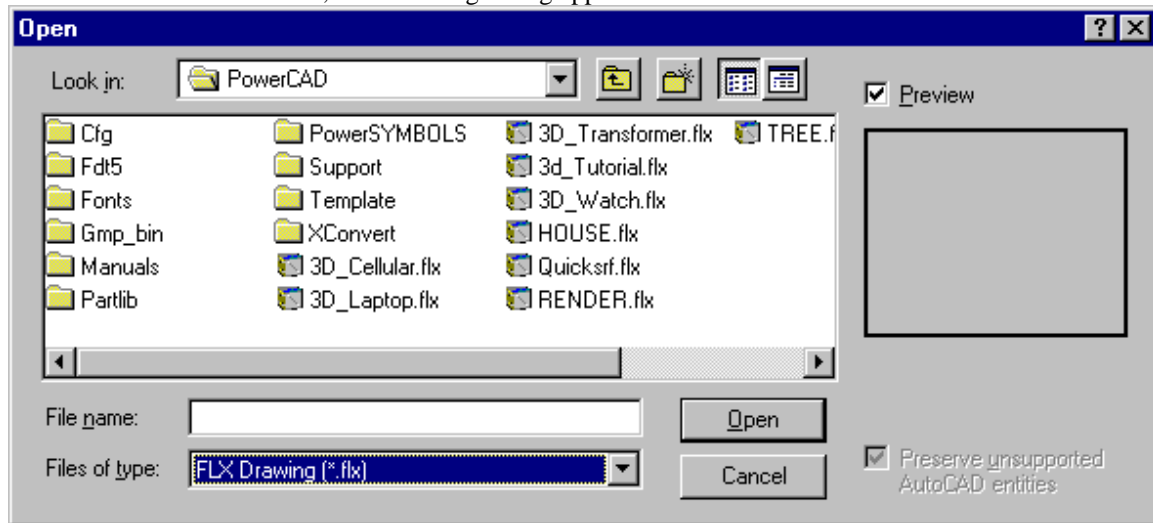
### **Located:**

- 1. Pull-down Menu: Parts – Merge External Part 1:1**
- 2. Or Enter Command or Alias at the Command Line**

The MERGE command is used to import an external .flx drawing file, introducing it as a new Part in the drawing.

The command is located on the Parts pull-down menu as the "Merge External Part 1:1 " selection. It can alternately be entered at the command line, if you choose to.

When the command is issued, the following dialog appears:



With this, you can browse to the location and select the file you want. Note the "Find" button that can be used when you know the name but not the file location. Other buttons are also provided to help with search mobility and file identification.

### Edit Part Attributes...



**Command: ATTXEDIT**

**Alias AX**

#### Located:

1. **Pull-down Menu: Parts – Edit Part Attributes**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

If only the Attribute values shall be changed, please use the command ATTXEDIT or call the function **Edit Part Attributes...** from the menu *Part*.

After the calling of the command you will be requested in the command line area to select the part of that the Attributes shall be edited. For this it is enough to click the desired part with the cursor at any spot. You can always select only one part.

After the selection of the part the program calls a dialog box.

In the window the Requests (left) and the current Values (right) of the Attributes included in the selected part are listed. Select the Attribute value to be changed. The selected value is transferred to the editing field (below) and can be edited.

---

#### See also:

Modify Part Attributes

QATTEDIT

## Modify Single Part Attributes (one by one) ...



**Command:** ATTEDIT

**Alias** ATT

### **Located:**

1. **Pull-down Menu:** Parts – Modify Part Attributes (1 by 1)
2. **Palette:** Parts
3. **Or Enter Command or Alias Directly at the Command Line**

Enter the command ATTEDIT in the command line area or select **Modify Attributes...** from the *Part* menu.

The first prompt will request that the part be selected. Make this selection either by naming the part, or by selecting the part with the cursor.

After the part selection, a dialog box will appear. The editing process from this point is very similar to the procedure followed in the creation of an Attribute, except that the name is now displayed, and cannot be changed.

If a part contains several Attributes, individual Attribute values can be displayed in the input field *Value* using these external. Clicking the right double arrow will display the next Attribute, while clicking the left double arrow will display the previous Attribute is displayed. That Attribute can at this moment be edited results from the name display above the input field.

---

### **See also:**

Edit Part Attribute

## **Attribute Modification**

**Command:** ATTVALUE

**Alias** AV

### **Located:**

1. **Pull-down Menu:** Parts – Modify Attribute Text – Modify Attribute Text Value
2. **Or Enter Command or Alias at the Command Line**

**Command:** ATTMOVE

**Alias** ATM

### **Located:**

1. **Pull-down Menu:** Parts – Modify Attribute Text – Move Attribute Text
2. **Or Enter Command or Alias at the Command Line**

**Command:** ATTROT

**Alias** ATR

### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Rotate Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

### **Edit Attribute Text Value**

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu *Modify Attribute Text*. ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

### **Move Attribute Text**

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

### **Rotate Attribute Text**

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

### **Attribute Modification**

**Command: ATTVALUE**

**Alias AV**

#### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Modify Attribute Text Value**
2. **Or Enter Command or Alias at the Command Line**

**Command: ATTMOVE**

**Alias ATM**

#### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Move Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

**Command: ATTROT**

**Alias ATR**

**Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Rotate Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

**Edit Attribute Text Value**

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu *Modify Attribute Text*. ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

**Move Attribute Text**

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

**Rotate Attribute Text**

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

**Attribute Modification**

**Command: ATTVALUE**

**Alias AV**

**Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Modify Attribute Text Value**
2. **Or Enter Command or Alias at the Command Line**

**Command: ATTMOVE**

**Alias ATM**

**Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Move Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

**Command: ATTROT**

## Alias ATR

### Located:

1. **Pull-down Menu: Parts – Modify Attribute Text – Rotate Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

### Edit Attribute Text Value

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu *Modify Attribute Text*. ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE  
Select attribute of part: P1  
New value for attribute <47>: 48
```

### Move Attribute Text

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE  
Select attribute of part: P1  
Target point: P2
```

### Rotate Attribute Text

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT  
Select attribute of part: P1  
New angle <0>: 90
```

### Export Attribute Information



**Command: ATTEXP**

## Alias ATX

### Located:

1. **Pull-down Menu: Parts – Export Attribute Information**
2. **Palette: Parts**
3. **Or Enter Command or Alias at the Command Line**

Writes attribute information to a file.

With this function, the user can extract attributes for parts or entities from the user drawing and save these in a CDF-format file. This is important when transferring and analyzing information from the drawing to or from programs for further processing

Proceed as follows

Enter the command **ATTEXP** or enter the function by selecting the menu point **Export Attribute Information...** in the pull-down menu Part.

Select the objects whose attributes are to be analyzed and the information that to be exported. The usual object select functions in the Options Bar are available.

### Select attributes

The left-most table lists the attributes that have been selected for export. An *N* indicates that it is a numeric value, and a *C* indicates that a character string is to be exported.

The next number displays the length of the field input for the attribute. For numeric values, the number of the decimal digits is also displayed.

Switches for Adding and Deleting attributes to the selection list are placed underneath the selection table. To delete attributes from the list, mark one or more lines in the list and select the switch Delete.

### Select Part Properties for Export

The user can select that part properties are to be included in the CDF-file in a manner similar to adding attributes.

Select the switch Adding / Changing below the display and selection field for the parts and properties.

All of the properties that can be exported will be shown in the selection area. Select a property by selecting the marked field. Enter the desired number of characters for the field width and, if necessary, edit the number of decimal digits.

### Change Field Separators

In the CDF file, all fields are separated by *Field separators*. Non-numeric fields are included in *text separators*. These values can also be changed in this box.

### View and Write CDF Files

When the user has selected all of the information desired, determine the name of the output file. A list will appear in the field CDF file. The switch '...' on the right side next to the input field for the file name displays a file selection box.

The switches Display and OK allow the user to search the drawing for the parts with the selected attributes, and the output in the CDF format is prepared. If the user selects the switch Display, a file will not be written, but will be displayed in a tabular Dialog Window.

From this window, individual lines, or all lines, can be copied to intermediate storage (Clipboard) for processing by other Windows programs.

Define Parts...



**Command: PARTDEF**

**Alias PDF**

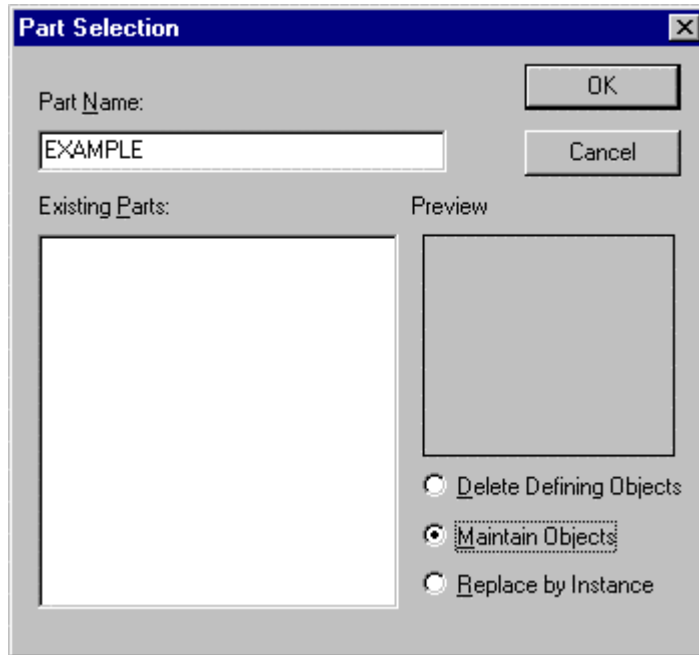
**Located:**

1. **Pull-down Menu: Parts – Define Part**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

The PARTDEF command defines a new internal part. Select the function **Define Part...** from the menu *Part*, or enter the command PARTDEF via the keyboard. Once defined, the internal part definition will allow for multiple part insertions *within the current drawing*.

When issuing the command, you are presented with the following dialog:





### Part Name

Names of parts can contain up to 31 characters. In addition to letters and numbers the special characters \$, -, and \_ are allowed. The box is not case-sensitive.

Enter the name of the part in the input field of the Dialog Box. Existing Parts are listed in the selection window below. If one of those parts is required, select the part from the Selection Window using the cursor.

If there are parts of the same name already in the selection box, a message will appear so stating. The message will give the user the option of choosing a new name, or of re-defining the existing part.

The user may also, in this Dialog Box, elect to delete the part in the original drawing. The appropriate box should be checked to exercise this function.

After this choice has been made, verify the selections of the Dialog Box by OK. The Box will close, and the part will have been created.

### Existing Parts

If there are any existing part definitions in the drawing, they will appear in the window on the left side of the dialog. They can be selected here for inclusion in the new part definition, if required. The **Preview Window** to the right will show a thumbnail picture of the part selected in the window on the left.

### Other Dialog Options

The remaining three options determine what action the program will take with the objects being included in the part definition.

- **Delete Defining Objects** – deletes the objects after including them in the part definition
- **Maintain Objects** – Leaves them in the drawing as individual objects as well as including them in the definition.
- **Replace by Instance** – Substitutes the new Part in place of the individual defining objects.

This completes dialog input, where pressing OK moves you on to the final command line entries that complete the PARTDEF command.

You are next asked to define: **Insertion point**

Each part must have a defined insertion point.

The **insertion point** is a reference point that enables the user to position the part properly during insertion. The insertion point of a part is also the base point for changing the scale, or for use as a rotation point during the insertion.

Any point on - or even off of - the part may be selected as the insertion point. Typically, users will select the left lower edge, the center of an object, or some other discreet point of the geometry as the insertion point. It is a good idea to use the Object Snap options for this selection, as it insures precision.

### Object selection

Next, the user must select the object or objects that will make up this part. These will join any existing part definitions selected in the dialog, in the new definition. To do this, the Option Bar or Dialog furnishes the standard object selection options. After selection is complete, confirm by pressing Enter. At this point, definition is made and the command terminates. The part is now ready for insertion as required.

## Write Part File ...



**Command: PARTEXP**

**Alias PX**

**Located:**

1. **Pull-down Menu: Parts – Write Part File**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

**Write Part File...** works identically to the function **Part Definition...** (Command PARTDEF) but is used to export parts as external drawing files.

In practice, it is used to create new drawing files (.flx, .dwg or .dxf) consisting of only selected drawing entities, regardless of their intended use. It is similar to the SAVEAS command, except that Saveas will include all current drawing elements in the new file.

Both PARTDEF and PARTEXP functions create part definitions within the current drawing. The proposed part name, the part insertion point, an internal part definition and/or selected drawing entities are input during the export process.

## Define Attribute...



**Command: ATTDEF**

**Alias AD**

**Located:**

1. **Pull-down Menu: Parts – Define Attribute**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

Attributes are, in general, text objects that are related to, or joined to, Parts (Blocks). They are treated together with the Part as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single Attribute with a Part, or several.

Using Attributes, the parts can be further defined by textual information. This information may be used to

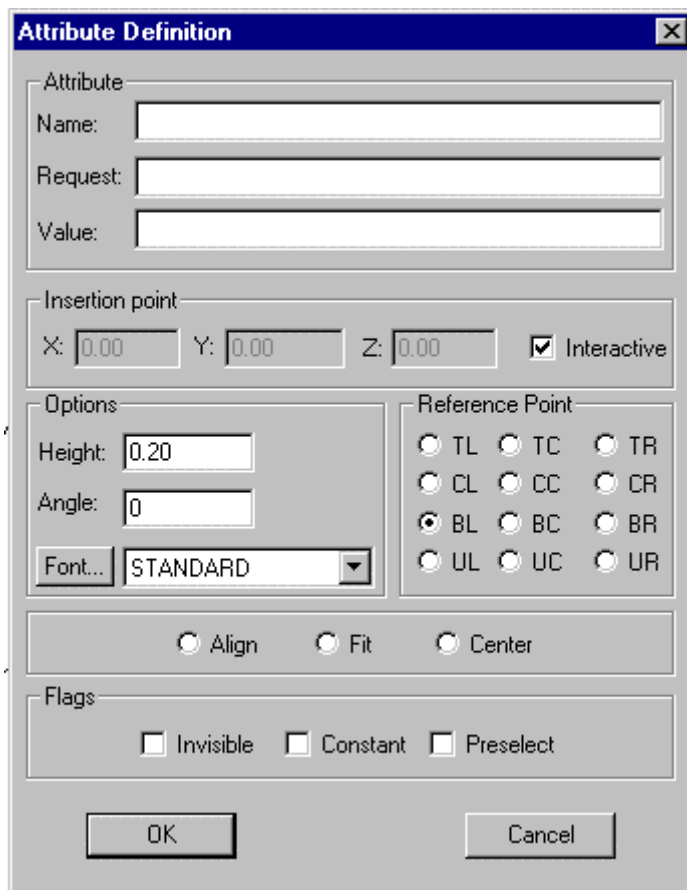
label Parts in the drawing, or may be used to contribute to a data base program for analysis later in the engineering process.

An Attribute consists mainly of three components; its name; its request; and the value. There are also a series of options that can be elected during the creation of an Attribute.

- The **Name** is the designation of the Attribute under that it is integrated in a part. Especially if a part contains more than one Attribute, the name is the most important criteria for the effective usage of the Attribute.
- The **Request** is a component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally, the request contains the invitation to enter a variable Attribute value.
- The **Value** is that portion of the Attribute that is written, together with the Part, into the drawing. It can be constant or variable, visible or transparent.

The following parameters for an Attribute may be defined during the creation of the Attribute, and edited later in the design process.

- Insertion point;
- Font options, specifically Font name, font height and font angle;
- Reference point for insertion;
- Visibility, whether visible or hidden; and
- Attribute value, whether constant or variable.



The image shows a software dialog box titled "Attribute Definition". It contains several sections for configuring an attribute:

- Attribute**: Three text input fields for "Name:", "Request:", and "Value:".
- Insertion point**: Three numeric input fields for "X:", "Y:", and "Z:", each with a value of "0.00". There is a checked checkbox labeled "Interactive".
- Options**: A "Height:" input field with "0.20", an "Angle:" input field with "0", and a "Font..." button next to a dropdown menu currently showing "STANDARD".
- Reference Point**: A grid of nine radio buttons labeled TL, TC, TR, CL, CC, CR, BL, BC, BR, UL, UC, and UR. The "BL" option is selected.
- Alignment**: Three radio buttons labeled "Align", "Fit", and "Center".
- Flags**: Three checkboxes labeled "Invisible", "Constant", and "Preselect", all of which are currently unchecked.
- Buttons**: "OK" and "Cancel" buttons at the bottom.

**Dialog Options**

## Attribute Name

Enter the name of the Attribute in the input field. This is the name that the Attribute will carry as it is joined to the Part, so it is important to use as descriptive a name as possible. When a part contains several Attributes, the name is an important criteria for effective use of the Attribute. An Attribute name can consist of up to 31 characters, including the special characters \_ (Under line) and \$ (Dollar).

## Request

The Request is a textual component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally the Request contains an invitation to enter a variable Attribute value.

Request can consist of up to 256 possible characters. Blanks and special characters are allowed.

The user may enter in this input field, for instance, "Enter norm term". This would be a variable. A typical answer to Request would be to enter information like "Schedule 40 steel pipe".

The answer to Request is interpreted as an Attribute value and as such inserted in the drawing (see below).

Occasionally the entering of a request can be omitted. This would be the case especially when the part is using a constant Attribute value (see below).

## Value

The Value is that component of the Attribute that is written, with the Part, into the drawing. It serves as a designation of the Part and contains information to be analyzed. Its content is determined either by the input in the input window Value or by the answer to Request (see above). An Attribute Value can consist of up to 256 possible characters. Blanks and special characters are allowed.

## Insertion Point

There are two options in defining this point in the Dialog Box; the entry of absolute coordinate values in the X, Y, and Z axes, or Interactive.

Interactive allows the user to define the Insertion Point by the cursor. By filling the control box marked Interactive, the user can exercise this option. The advantage to this option is that Dragmode is active as well, offering a preview of the positioning.

## Text Options

With the help of the text options the user may determine the height (font size), the rotation and the Font style of the Attribute text.

**New text styles** can be directly created from the **ATTDEF** command

The **FONT** command can be issued by depressing the font button on the text dialog

Define the font size by the input field Height. Enter the desired text height in drawing units, not in text points.

Via the input field Angle, specify the insertion angle of the Attribute value. For an counter-clockwise rotation enter a positive value, for a clockwise rotation enter a negative value, between 0 and 360 degrees.

A selection window is available to define the Font style. A list of the available fonts is displayed. Select the desired Font style.

## Reference Point

The Reference Point determines the position and alignment of the text object in relation to the Insertion Point of the Attribute. To this end, a Reference Point is defined on the text object. During insertion, this point will be coordinated with the Insertion Point.

If the reference point, for instance, is the lower left corner of the text object, then the text will run from

the Insertion Point to the right, and the lower edge of the text will be located at the height of the Insertion Point.

In total there are twelve Reference Points available. Select one of them with the cursor. The Reference Points represent the initials of the position on the text object. BL, for instance, represents Bottom Left, while UR represents Upper Right. Center positions are available as well.

### **Align**

Text objects may be rotated or scaled during the process of insertion by using the function Align. Determination of rotation angle, as well as a scaling factor, is made by indicating two points in the drawing using the cursor.

Activate the option Align by filling the circle marked "Align". Activating this option will remove the options Insertion Point, Height, Angle, and Reference Point. Align requires that the user pick two points on the drawing. The text object will be placed left justified on this baseline.

Confirm the use of the option Align after the input of the string by clicking OK.

Then simply follow the prompts for a first point, then a second point. The first point is interpreted as the Insertion Point, the second point sets the angle of insertion. The second point will also determine a scaling factor that will, in turn, determine the text height.

### **Fit**

Fit enables the user to rotate and expand a text object during the insertion. The text object is not scaled, so the text height remains unchanged, but the text string is expanded or compressed as required by the Fit. The procedure for this option is:

Activate Fit by clicking the panel with this name at the lower left side of the dialog window Text. The options Insertion Point, Angle and Reference Point will not be available. The parameters required for Fit are two points that must be defined during the inserting of the text object. The reference point of the text object is automatically positioned left justified on the basis line;

Answer the prompt for "first point". This point is interpreted by the program as the Insertion Point. It is also the point of rotation for the rotation angle and the first point for the expansion factor; then

Indicate the second point. This will determine the rotation angle and expansion factor, and set the text height.

### **Center**

Center allows fast insertion of a text object with the vertical and horizontal alignment centered. Text height and rotation angle may be specified in the normal fashion.

### **Flags**

#### **Visibility**

Invisible is a switch panel that controls the visibility of the Attribute Value. A cross in the control box displays active status of the option.

If Invisible is activated, the Attribute Value will not be displayed during the insertion of the Attribute in the drawing. this is the option to choose if the Attribute contains information to be analyzed later that does not have to be displayed on the drawing.

It is also convenient to use this option when there are so many Attributes that to show them all would make the drawing to "busy", confusing the drawing.

#### **Constant**

This switch panel is used for switching between constant and variable Attribute values (see above). A cross in the control box displays the active status of the option.

With Constant activated, the text entered in the Value field will be written as a constant Attribute Value

in the drawing. There will be no opportunity to insert any variable data.

With Constant de-activated, the request at the command line to insert variable Attribute Values will appear.

### **Preselect**

The Preselect mode allows the user to create Attributes that accept their default values.

With Preselect active, Attribute Values are not requested. The Preselect mode is the most flexible mode, since the mode allows the user to edit pre-selected Attribute Values after insertion by using the editing commands.

Verify the entries by OK switch or by pressing RETURN. The Attribute will be inserted as specified.

**See also:** QATTDEF

Edit Attribute Definition...

**Command:** ATTDEDIT

**Alias** ATD

#### **Located:**

- 1. Pull-down Menu: Parts – Modify Attribute Definition**
- 2. Or Enter Command or Alias Directly at the Command Line**

You can modify an existing attribute definition by using the ATTDEDIT command. Select an attribute definition in the drawing. Immediately a dialog box is displayed that is similar to the dialog to create attribute definitions.

If the check box labeled Interactive is marked, you will be prompted to relocate the attribute definition when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

> ATTDEDIT

Select a attribute definition:

*<Dialog box appears to edit attribute definition parameters>*

Insertion point:

Hint: ATTDEDIT makes it easy to create several similar attribute definitions. Just copy a master attribute definition, and then edit the name and request parameters with the ATTDEDIT dialog.

---

**See also:**

ATTDEF

Grouping Entities



**Command:** GROUP

**Alias** G

#### **Located:**

- 1. Pull-down Menu: Parts – Group**
- 2. Palette: Parts**
- 3. Palette Edit**
- 4. Or Enter Command or Command Alias Directly at the Command Line**

Drawing entities can be combined in named Groups. Essentially, groups are a named selection set, and as

such, the user is able to manipulate all entities within the Group in a single operation (Moving, Rotating, Mirroring, or Scaling, for example) just by specifying the group name. Single entities of a Group may still be *edited* individually (for example, intersecting or stretching). Entities may be removed or added to a Group at any time. An object may be a member of more than *one* Group. *Groups* may be contained in *other* Groups. Group definitions are stored in a distinct table of the drawing database. Groups are maintained in this fashion from one drawing session to the other.

### Differences Between ‘Parts’ and ‘Groups’

A Part (a block) has its own insertion point. A Group has none. A Part can occur multiple times within the drawing. If a block is redefined, all of its occurrences are updated. Groups are not treated like that. Groups can, however, be copied. A Group that originates from another Group that has been copied is a separate union of entities. Each Group has a unique name.

**Note:** If an entity that is member of a Group is included in a part definition, the entity is deleted from the drawing and removed from the Group.

### The Command GROUP

To group entities, or to manage and manipulate Groups, use the command GROUP. The options within the command are:

<b>Create</b>	Brings entities into a newly named Group
<b>Explode</b>	Abolishes a Group definition
<b>Rename</b>	Alters the name of an existing Group, especially of anonymous groups
<b>Add</b>	Adds drawing objects to an existing Group
<b>Remove</b>	Removes drawing objects from a Group
<b>Selectable</b>	Determines whether a Group is selectable or not

### Creating Groups

To create a group, first determine a unique name. The name will be required to identify the group later for a number of command options. In addition, specify whether the group should be selectable or not (see below).

#### To define a new group,

1. Call the command GROUP with option Create,
2. Specify a unique group name (max. 31 characters),
3. If desired, enter a description (max. 64 characters),
4. Determine whether the group should be selectable or not (the default is Yes), and
5. Select the drawing objects that should belong to the named group.

To create a group using Command Line:

```
> GROUP
Select option: Create
Group name: G1
Description: Screw
Should the group be selectable (Yes/No)? Yes
Select objects: Select the entities that should belong to the group
```

### Exploding Groups

Group definitions may be abolished by using option **Explode** of the GROUP command. When a group

definition is dissolved the corresponding entry in the group table is deleted in the drawing database.

The only way to abolish a group definition is by abolishing its name.

To explode a group,

1. Choose the command GROUP with its option *Explode*,
2. At the prompt "Group name: " enter the name of the group, to be exploded.

Because a single entity may belong to multiple groups and groups may be part of other groups, the **Explode** option does not allow the user to pick an entity within a group to explode the group.

### Renaming Groups

The name of a group may be changed by using the GROUP command **Rename**. It is useful to employ this method when "anonymous" groups that have been created by copy commands exist. The system will automatically apply anonymous names to copies group (\*\*A1, \*\*A2, etc.) that exist through the use of one of the commands like COPY, MIRROR, or ARRAY.

To rename a group,

1. Call the command GROUP with its option Rename;
2. At the prompt "Current group name: " enter the name of the group (for example "\*\*A0");
3. At the prompt "New group name: " enter the name of the specified group (for example "Screw\_Copy1")

### Add / Remove Entities

Two other options of the command GROUP are provided to allow the user to integrate additional entities into an existing group or to remove entities that are currently in the group.

After a group name is specified, the entities that belong to the group are highlighted.

► To add entities to an existing group, simply 'pick' the entities that are to be added (they will *not* be highlighted).

► To remove entities from an existing group, 'pick' the highlighted entities that are to be removed

**Note:** Should *all* entities be deleted from a group, remember that *the group will still exist as long as it has a name!*

### Working with Groups

Editing commands will treat all entities in a group as a single object. Commands like MOVE, ROTATE, or COPY will act on the entire group.

If the user should select an object for one of these commands at the prompt that selection will lead to the automatic selection of all group entities.

A typical prompt of a command like this would be:

Select objects:

The rule is: In commands that allow **multiple selection**, identifying a *single* entity of a group is a selection of *all* entities belonging to the group(s). When the user is *prompted* to select multiple objects, he or she may choose the object selection option *Group* from the options bar, and enter a group name to select all of the entities belonging to the group. Groups, therefore, can also be named selection sets.

Other MODIFY commands (Offset, Lengthen, Intersect, Trim, etc.) request a single entity in one or more steps (usually to modify the geometry of drawing entities). A typical prompt for a command like this would be:

Select entity (Line, Circle, Arc, 2D-Polyline):

In this case **single selection** is being used. Any time the user is prompted to select a single entity, the



group does not matter. These commands modify the geometry of single selected entities even though those entities belong to a group. The modified entities will remain members of the group. This feature makes allows great flexibility in working with GROUPS and provides significant advantages for the user compared to using BLOCKS.

The following table summarizes the editing commands and describes how entities that are member of a group will be processed when using each command.

#### DELETE

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. When a single entity of a group is deleted, the entity is removed from the group definition.

#### MOVE, ROTATE, SCALE, FLIP

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used.

#### MIRROR, COPY, ARRAY

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. The system will apply anonymous names to copied groups ("\*A1", "\*A2", etc.) that are created by the use of these commands.

#### STRETCH

Those entities belonging to a group that have been selected using the object selection modes *Crossing* or *Cpolygon* will be stretched.

#### TRIM, EXPAND

The prompt to "Select cutting edges" or "...boundary edges" will include all entities of a group, unless the Object Selection Mode *Single* has been used.

#### OFFSET, FILLET, CHAMFER, LENGTHEN, INTERSECT

These commands always prompt the user to select a *single* entity to be modified. That an entity belongs to a group does not matter in these operations. Single entities altered by these commands will remain in the group. If *new* entities *originate from these commands* they will *not* be automatically included in the group.

#### DELPARTIAL, REJOIN

When an entity (line, circle, arc or 2D-Polyline) is broken into two or more entities by partial deletion, only one of the entities will remain in the group. When an entity that has been broken into two or more pieces is rejoined, the newly created entity is no longer member of any group, even if one or more of the pieces was part of a group.

#### PARTDEF, HATCH, HATCHEDIT

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. This also applies to the command HATCHEDIT when the user is prompted to select additional objects: to be included to the set of entities for associative hatch editing.

#### TEXTEDIT

The text entity retains its association to its group if the command TEXTEDIT is used to modified text.

## Externally Referenced Drawings



**Command: XLINK**

## **Alias XL**

### **Located:**

- 1. Pull-down Menu: Parts – External References**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Externally Referenced Drawings (XREF's) serve especially to

1. Create assembly drawings from different files
2. Insert detail drawings into a finished one
3. Temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is realized within a running project where a workgroup uses XREF's to create several drawings and details in a network.

Another advantage is that within the current drawing, only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file. This keeps the file size small and as such it is easier to work with. In this way, very complex drawings can be created from a group of relatively small files. In these instances, XREF's have an advantage over the use of blocks.

### **XLINK command: Control externally referenced drawings**

The XLINK command allows you to create a link to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

### **Differences to Part Insertions**

Although the treatment of linked drawings is basically similar to those of part insertions (block insertions), some fundamental differences include:

An Xref establishes a link to another drawing file, but it does not become a permanent part of the current drawing.

### **Dependent Symbols**

When attaching an external file, those symbol tables are loaded into current drawing that are required to display the referenced coincident to its original: layers, linetypes, text styles, dimension styles, and block definitions.

The program utilizes the following scheme to name dependent symbols in the current drawing:

Logical\_Xref\_Name| Dependent\_Symbol\_Name

For example, the layer names of an externally referenced drawing attached with the logical name SHAFT are listed as:

SHAFT|CONSTRUCTION

SHAFT|DETAILS

SHAFT|DIMENSION

Other dependent symbol table entries like linetypes or blocks will carry the same naming convention.

Note that you cannot redefine or rename dependent symbols. Also, dependent blocks cannot be inserted and a dependent layer cannot be made the current layer.

But the dependent symbol conventions allow you to control the visibility, the colors, and the linetypes of externally referenced drawings. The layer dialog box displays the names of the dependent layers and you can apply the options On/Off, Thaw/Freeze, Linetype, and Color on them.

The logical name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

To keep the visibility and display settings over subsequent sessions, make sure to set the system variable **VISRETAIN** to 1. By default VISRETAIN is set to a value of 0 that means that visibility and display modifications made to the dependent layers are not maintained when you leave the current drawing.

### **The Dialog Box *Externally Referenced Drawings***

The dialog box displays a list box and a column of option buttons.

The list box displays the following information on externally referenced drawing files that have been inserted into the current drawing

1. The logical name assigned to it when the file was attached, and
2. The path and filename of the referenced drawing

The options available to manage externally referenced drawings are provided in a column of buttons:

<b>Attach</b>	Insert an external drawing file into the drawing as block
<b>Reload</b>	Update externally referenced drawing to display its current state
<b>Path</b>	Relocate/redirect or alter source of the referenced file
<b>Bind</b>	Transform an externally referenced drawing to an ordinary block in the drawing
<b>Detach</b>	Delete an externally referenced drawing from the current drawing and erase the dependent symbols from the drawing database tables

These options are described in detail in the following sections.

#### **Attach**

The *Attach* option allows you to insert an external drawing as a block to the current drawing.

#### **To attach (set up a link and insert the contents) to an external drawing file:**

1. The standard file dialog box allows you to specify the drawing you want to attach to the current drawing.
2. Within the dialog box *Attach External Drawing*, a unique Logical Name (block name) needs to be determined for the associated referenced file. The program suggests using the name of the file. However, in many cases it is recommended to use a short logical name, because the logical name will be used as prefix for the names of dependant symbol names for layers, linetypes, etc.
3. The other options of the dialog box let you specify preferences for the insertion and are identical to those found in the dialog box of the INSERT command (insertion point, scale factor, and rotation angle), as described previously.

The drawing attached is based on the most recently saved version of the drawing file.

#### **Reload**

The option *Reload* allows you to update and external reference any time during a current drawing session. Any Reload reflects any subsequent modification made to the external file.

Note that when you open a drawing that contains externally referenced drawings, these are displayed automatically in their current state. When working on a project in a workgroup over a network, it might be necessary to update one or more external references while you are working on a current drawing. The program will reload the specified drawing(s) in the shape as they have been saved most recently.

#### **To reload externally referenced drawings in a current drawing session,**

1. select the external reference to be updated from the list box
2. click the *Reload* button

#### **Path (Relocate or alternate source of the referenced file)**

Occasionally the storage location of an externally referenced drawing might have been changed or the referenced file has been renamed. Also, it might happen that, when sending the project files including referenced drawings to someone else, the path to the referenced files needs to be updated.

In such a situation the option *Path* of the dialog box allows you to re-establish the path to the referenced drawing, to reload a renamed referenced file, or to replace a referenced drawing by another file.

**To edit or relocate the path to a previously linked reference drawing,**

1. select the external reference to be relocated from the list box
2. click the *Path* button

From now on, the link to the referenced file will be maintained as specified and the data of the referenced file will be updated from that source.

**Bind**

In certain situations it is useful to make an externally referenced drawing a permanent part of a drawing, for example when you need to send the drawing to someone else or if it has been finalized and is going to be archived. Binding the referenced drawing to the current drawing has the advantage that there is no need to gather an assembly of drawings within an archive or mail.

The option *Bind* of the XLINK command allows you to transform an externally referenced drawing to an ordinary block in the drawing. Once you bind an externally referenced drawing, the link to it is broken and you will no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definition, text styles, and dimension styles into the current drawing. Within each corresponding dependent symbol name, the vertical bar symbol (|) is replaced by a new three character symbol \$n\$, where n is a number that is usually 0. For example, the layer name HOUSE|BASEMENT will be altered to HOUSE\$0\$BASEMENT.

**To bind one or more external references to the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Bind* button

**Detach**

Externally referenced drawings that are not needed any longer in the current drawing can be removed from the base drawing with the *Detach* option. This will remove all of its dependent layer, linetype, and block references as well.

Note: Although you can erase an externally referenced drawing with the DELETE command, it is recommended to use the Detach option of the XLINK command, as this operation will also remove the Block definition from the drawing database.

**To remove one or more external references from the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Detach* button

**Open**

As the Multiple Document Interface allows you to display up to four drawings on the desktop, the option *Open* of the dialog box allows you to open an externally referenced drawing simultaneously on the CAD desktop. This allows you to modify a referenced drawing, for example, if you have detected discrepancies or inconsistencies in the linked drawing that needs to be fixed.

**To open a referenced drawing in another drawing window (viewport),**

1. select the external reference you want to open from the list box
2. click to the *Open* button

Once you have modified a referenced drawing, save the drawing (and, if the modification has been finished, close it). To display the revised external reference in the master drawing, call the command XLINK and reload the corresponding file (see above: *Reload*).

### Insert

The option *Insert* allows you to insert another copy of the Xref chosen from the list box into the current drawing. You may then define the insertion point, the scale factor and the rotation angle for that additional instance of the external drawing in the current drawing.

You can move, scale, rotate, and copy externally referenced drawings. But, you cannot explode an external reference.

**Note:** You should not remove an Xref from the current drawing with the DELETE command, because it does not purge the Xref block definition from the drawing database and the associated symbols will be maintained. To remove an external reference correctly from the current drawing, it is recommended you use the Detach option of the XLINK command.

The Insert option, however, can serve to reinsert an external reference that accidentally has been deleted.

---

### See also:

XINSERT

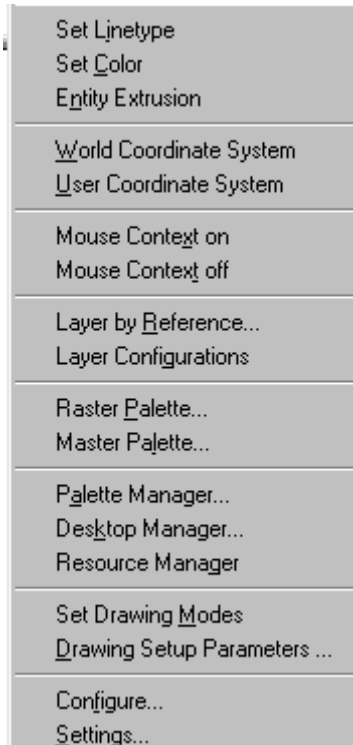
Parts Palette



This command will place the palette containing Parts command icons on the desktop for easy access. The palette can be dragged and dropped as required, to place it in the most convenient location. By holding the cursor over an icon, the operator can view the popup ToolTip that identifies the associated command. After use, the palette may be closed by clicking on its close button beside the Palette header.

Tools

"Tools" Pull-down Menu



The Tools pull-down menu contains command instructions that control Layers and command option prompts. For a description of each command, please refer to the specific topics below.

## Layer Manager



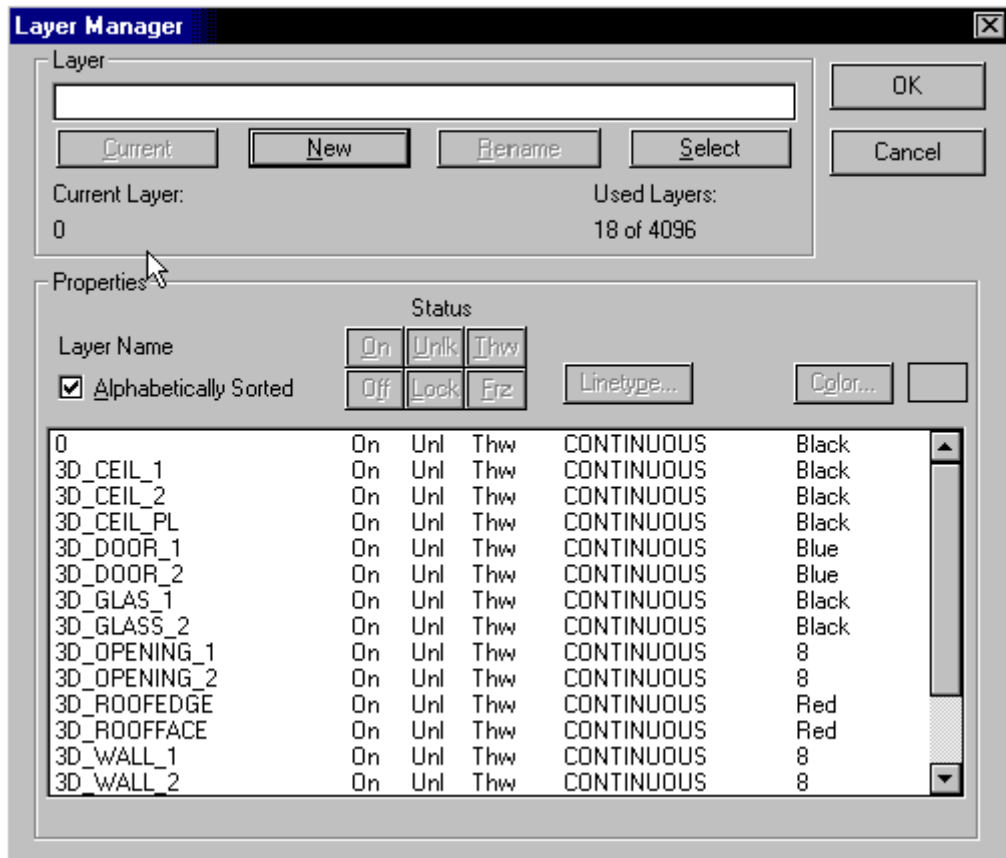
**Command: LAYER**

**Alias LA**

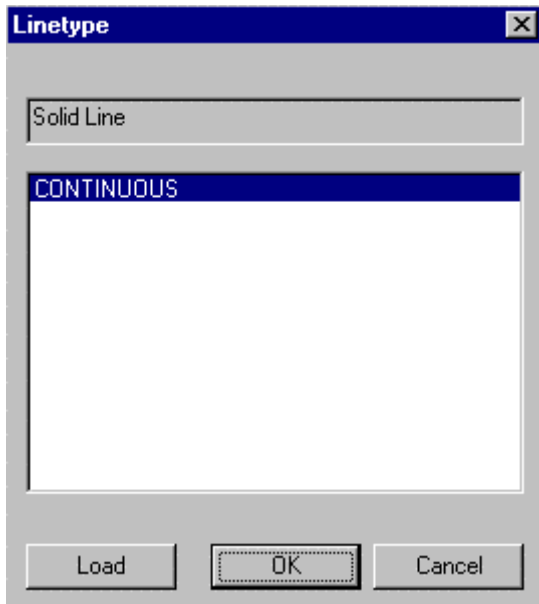
### Located:

1. **Pull-down Menu: Tools – Layer**
2. **Properties Toolbar**
3. **Or Enter Command or Alias at the Command Line**

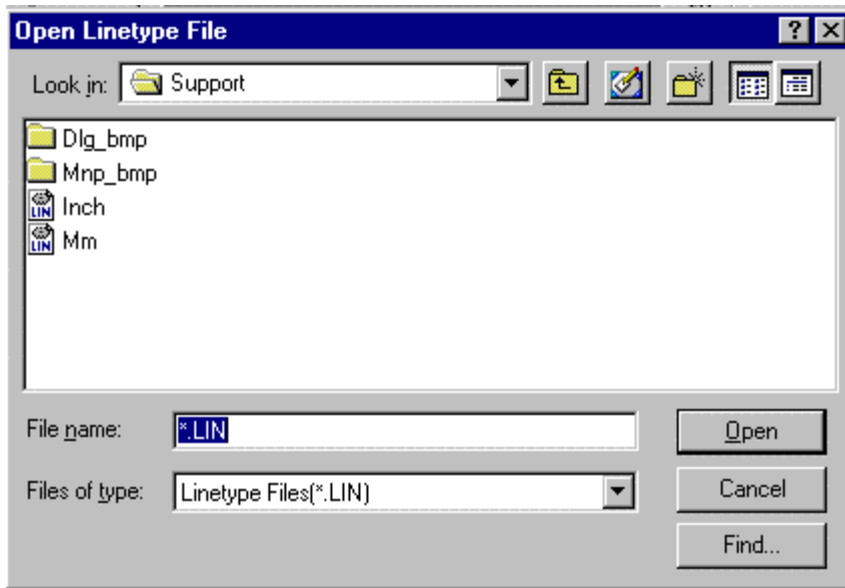
Creating new layers, choosing an active layer and controlling all previously described layer characteristics are accomplished through a dialog box. This dialog box is called *Layer Manager*.



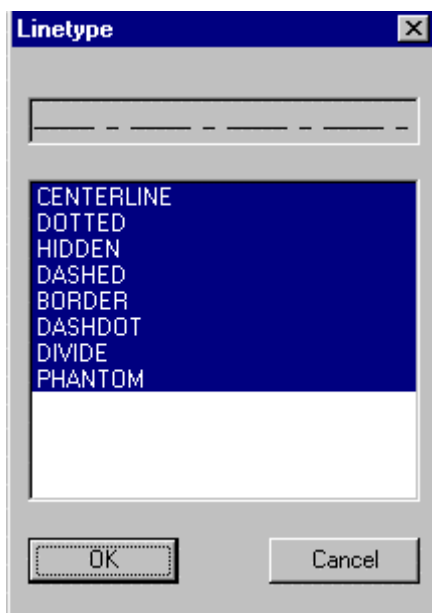
Line types may be loaded directly from the linetype dialog, in the Layer Manager dialog, by pressing the **Load** button



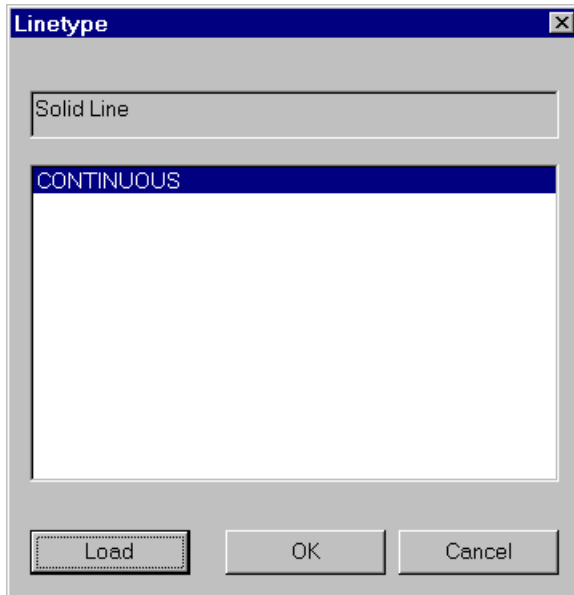
You will then be asked to select the linetype file (.lin) containing the linetype you want to open:



At this point, you are presented with the dialog below, showing all of the linetypes contained in the linetype file that you selected. As you select each linetype in turn, a representation of its appearance is presented in the window at the top of the dialog. Choose the linetype that you want loaded and press OK.







In the **Layer Manager** the default button is the **New** button. When you type in a layer name and press the return key, a new layer will be created and you remain in the layer manager.

### Current Layer and Used Layers

In the upper rows of the *Layer Manager* Dialog Box is a field for the name of the active (current) layer (left window) and a field for the number of defined layers in this drawing.

### Layer

Later rows contain combined view and entry fields for layers that still have to be worked on, and a selection box that allows the user to choose between creation of new layers, renaming layers and choosing an active layer.

### Properties

There are fields for the layer state and for choosing line types and layer color.

Directly beneath that, in the dialog box, all the existing layers of the drawing are shown. Any of these may be selected and activated with the cursor. The layer color and linetype are also shown there.

The meaning of the abbreviations used are shown here.

Abbreviation	Meaning
On	Visibility on - The layer becomes visible
Off	Visibility off - The layer becomes invisible
Lock	Status free, drawing elements can be edited
Unlk	Status closed, drawing elements cannot be edited
Thw	Layer thawed, visible and can be edited
Frz	Layer frozen, invisible and cannot be edited, will not be regenerated

### Quick Layer Setting...

**Command:** SETLAYER

**Alias** STL

#### Located:

1. Pull-down Menu: Tools – Set Layer
2. Properties Bar

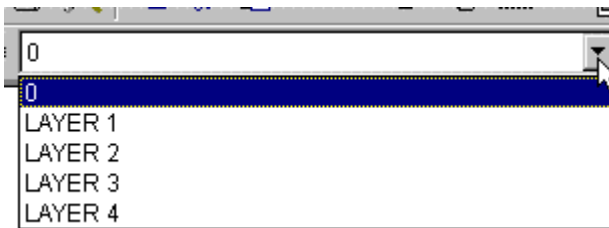
### 3. Or Enter Command or Alias Directly at the Command Line

This function allows the user to switch the active layer. It is not possible to create a new layer or to change any characteristics of the existing layer through the selection of this button. These changes must be made in the fashion described previously in Layer Control.

The letters in front of the layer names show the status of the characteristics visibility, access, and thawing/freezing. They have the following meaning:

- ON = On (Visibility)  
OFF = OFF (Visibility)
- U = Unlocked (Access)  
L = Locked (Access)  
T = Thawed (Thawing/Freezing)

When changing layers on the Property Bar, click on the down arrow to the right of the layer window, to produce a drop-down window showing all drawing layers.



You can then select the layer you want to be current from this list. After selection, the drop-down window closes and the layer selected should appear in the collapsed window. This indicates that it is now the current layer.

**Note:** A frozen layer cannot become the active layer, so that choice will not appear for the current layer. Highlight a layer in the list with the cursor and verify your choice by clicking on OK.

---

#### *See also:*

Layer Manager

Load Linetype(s)...

#### **Command: LINETYPE**

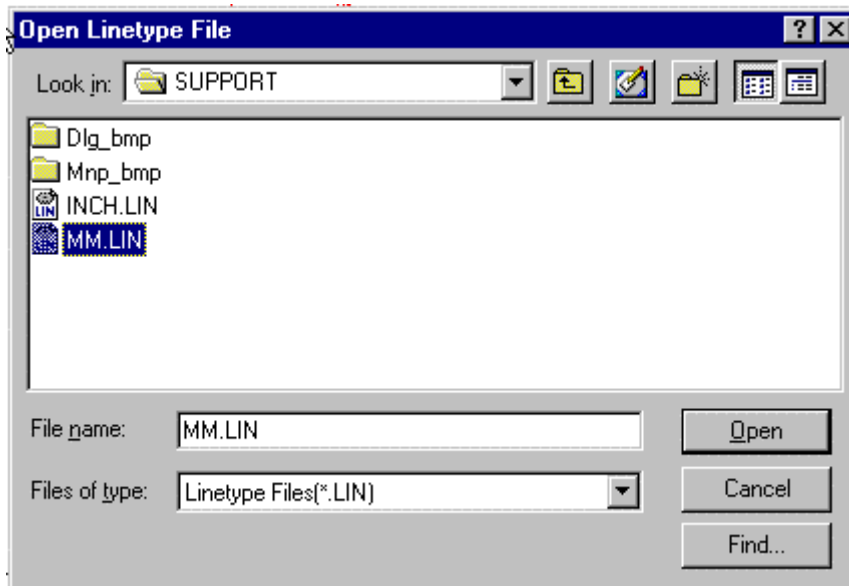
#### **Alias LT**

#### **Located:**

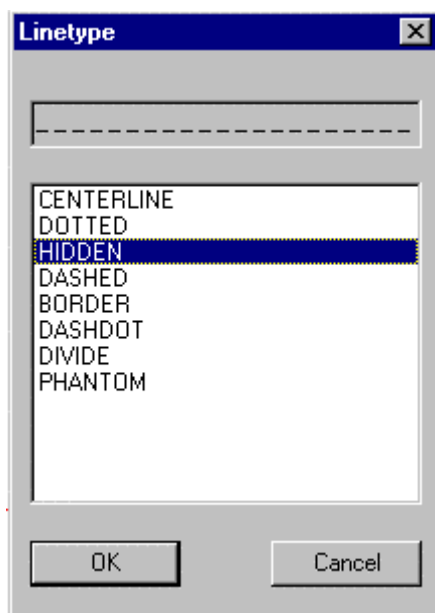
1. **Pull-down Menu: Tools – Load Linetypes**
2. **Or Enter Command or Alias Directly at the Command Line**

The LINETYPE command allows the user to load additional linetypes for use in the drawing. These linetypes will join those originally loaded with the drawing when it was opened, including those opened in previous drawing sessions.

Linetypes are stored in .lin files and these files are selected for linetype extraction when using the command, as in the commands first dialog (below):



Once you select and confirm the .LIN file to use, the following dialog opens, and displays the linetypes that are available in that file.



As you click on each one, a representation of the linetype appears in the window at the top of the dialog. This aids in linetype selection. Multiple linetypes can be loaded at once through the use of the Shift or Ctrl key while picking individual linetypes on the list.

Linetypes that are loaded, will be saved along with the drawing so as to be available at a later time.

Once loaded, linetypes can be assigned as default to layers (Command LAYER), can be assigned to existing linear entities (Command PROPEdit), or set as default linetype for subsequent dline drawing (Command SETLINETYPE).

Further information on linetype definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

See Also: Command **LOADLTYPE**

## **CELTYPE**

### **Variable CELTYPE**

#### **Located:**

- 3. Pull-down Menu: Tools – Set Linetype - Set Layer's Linetype**
- 4. Or Enter Command or Alias at the Command Line**

This variable specifies the current linetype when drawing entities. The argument of the variable contains the name of the linetype.

#### **Properties**

<i>Type</i>	String
<i>Default</i>	"BYLAYER"
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### **Commands**

'SETLINETYPE. See also command **LINETYPE**.

Change the Linetype...

#### **Command: SETLINETYPE**

#### **Alias SLT**

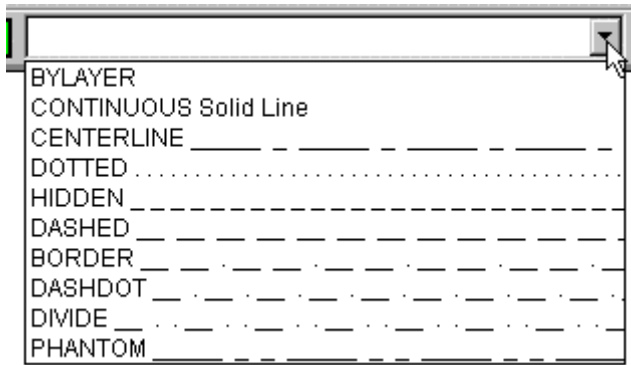
#### **Located:**

- 1. Properties Bar**
- 2. Pull-down Menu - Tools - Setlinetype - Set Explicit Linetype**
- 3. Entered Directly at the Command Line**

For reasons of clarity, the user may want to draw different objects with different linetypes than the one marked BYLAYER. This command sets the default linetype to be used for subsequent line drawing, overriding bylayer settings. Using SETLINETYPE – Bylayer returns the program to layer-assigned linetype control.

Remember that a new linetype has to be loaded before it is available for use, and that linetype should be designated before drawing a new object.

To set a current layer (other than bylayer) on the Properties Bar, click on the down arrow to the right of the linetype window to see a drop-down menu that lists all loaded linetypes.



Click on the linetype you want to make it current. When you wish to return to layer control over linetype, simply repeat the procedure and select "bylayer" from the list.

---

*See also:*

Load Linetypes...

## Quick Color Setting...

**Command:** SETCOLOR

**Alias** STC

**Located:**

1. **Pull-down Menu:** Tools – Set Color – Set Layer's Color
2. **Or Enter Command or Alias Directly at the Command Line**

With the SETCOLOR command the user can specify a line color that over-rides the default layer color, or that returns color control to the layers color setting. This command resets the CECOLOR variable to the selected setting.

## CECOLOR

**Variable** CECOLOR

**Located:**

1. **Pull-down Menu:** Tools – Set Color - Set Explicit Color
2. **Or Enter Command or Alias at the Command Line**

This variable determines the current color when entities are drawn. This color assignment over-rides layer color assignments. To return to Layer control of color, use the CECOLOR – "bylayer" option.

### Properties

<i>Type</i>	Integer
<i>Default</i>	"BYLAYER" (= 256)
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Command

'SETCOLOR

THICKNESS

**System Variable - Thickness**

## Alias TH

### Located:

1. **Pull-down Menu: Tools – Entity Extrusion - Set Thickness or No Thickness**
2. **Or Enter Command or Alias at the Command Line**

Sets the value for the default thickness (extrusion). Specifying the value 0.00 means that no thickness is applied.

### Properties

<i>Type</i>	Real
<i>Range</i>	Any positive or negative number
<i>Default</i>	0.00
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### See also

System variable ELEVATION

## THICKNESS

### System Variable - Thickness

## Alias TH

### Located:

1. **Pull-down Menu: Tools – Entity Extrusion - Set Thickness or No Thickness**
2. **Or Enter Command or Alias at the Command Line**

Sets the value for the default thickness (extrusion). Specifying the value 0.00 means that no thickness is applied.

### Properties

<i>Type</i>	Real
<i>Range</i>	Any positive or negative number
<i>Default</i>	0.00
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### See also

System variable ELEVATION

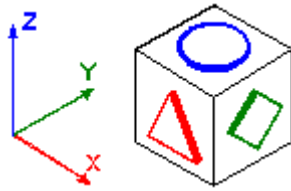
## Coordinate systems

The coordinate system of this program permits the unambiguous determination of every single point on a drawing surface or a drawing object, through coordinates. In simple terms, coordinates precisely determine every aspect of the drawing. These coordinates originate in the coordinate system. The program uses a Cartesian coordinate system consisting of three coordinate axes (essentially length, width and height (elevation) or more technically, (x,y,z coordinates). All axis have the same measurement units and each point in the drawing has a unique (x,y,z) location.

All coordinate systems have an "origin". This is the point in 3-dimensional space where all three values are zero. To determine the coordinates of a point, the distances of this point from the origin are measured along the x, y and z axes. These establish the x,y,z values for the point.

In some situations, you may find that points may need to be assigned coordinates based on more than one coordinate system. This may be because you may have information for your drawing from different sources, where different coordinate values have been used, or where you may need to convert your drawing information into another coordinate system for some other purpose. AllenCad allows for this by permitting you to create different coordinate system definitions and making it possible for you to switch from one coordinate system to another, at will. By default, the first entities that you draw establish what is

referred to as the "World" coordinate system. After that, any new coordinate system that is needed, is given a separate User Coordinate System (UCS) designation.



## World Coordinate System

The world coordinate system is a permanently installed Cartesian coordinate system that is used as a standard for all drawings. The x-axis measures the horizontal distance, the y-axis the vertical distance from the origin. The z-axis is orthogonal to the x- and y-plane and measures the distance of a point to this plane.

The world coordinate system cannot be changed by the user within a particular drawing.

## User Coordinate Systems

A user coordinate system can be freely defined by the user, having the world coordinate system as a basis.

The following changes can be made in a user coordinate system:

The origin can be moved to any point in the world coordinate system.

The user coordinate system can be angled or turned around one or more axis in relation to the world coordinate system.

Using the user coordinate system, for example, it is possible to change the location of the drawing plane in such a way that it always lies parallel to the objects that are to be drawn, making the construction of 3-dimensional objects much easier.

User coordinate systems are an advanced drafting concept and AllenCad subscribes to the same conventions used in many other programs in this regard. More in-depth information from other sources in this regard is generally applicable to AllenCad concepts, as a result.

## User Coordinate Systems



**Command: UCS**

**Alias UC**

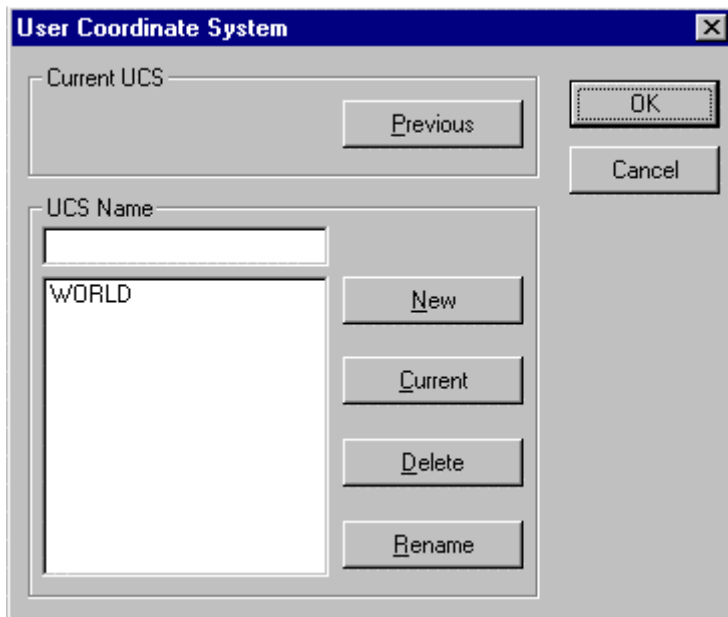
**Located:**

1. **Pull-down Menu: Tools – World Coordinate System and**  
**Tools - User Coordinate System – UCS Manager (UCS\_C) and**  
**Tools - User Coordinate System – UCS Origin (UCS) and**  
**Tools - User Coordinate System – Rotate Around Z Axis (UCS\_R-Z)**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

When the UCS command is issued, an option menu is presented that lead directly to individual UCS functions.



The UCS Manager function is available by clicking on the Control option in the menu above. The UCS Manager controls the setup and definition of User Coordinate Systems.



It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate systems will be saved along with the drawing and can be used again when opening the drawing later on.

### Active coordinate system

From all the user coordinate systems in one drawing, only one can be active at any given time.

Choosing a coordinate system to activate is done in a dialog box. Using the same dialog box it is possible to rename, create or delete already existing user coordinate systems.

### Create new user coordinate system

User coordinate systems are always created in two steps. First the parameters of the user coordinate system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system, enter the command UCS. Through the message UCS **origin <0,0,0>** in the text window you are asked to enter the coordinates of the origin of the user coordinate system. The coordinates **<0,0,0,>** are offered as a help. If you don't want to change the origin of the coordinate system



then verify this choice through enter. Otherwise please enter new coordinates and verify this entry also with return.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented..

Control      World      Z-direction      3point      View      Rotation

The option **Control** calls up a dialog box that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog box will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option **World** chooses the world coordinate system to be the active coordinate system. This allows you to go back to the world coordinate system quickly without having to open the dialog box of the command control.

### Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .
2. The direction of the axis of the coordinate system can be determined through fixing the z-axis.

To do that, choose a point through that the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
> UCS
UCS origin <0,0,0>: Z-axis
UCS origin <0,0,0>: Enter
Point on the positive z-axis: P1
```

### 3Point

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane, nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the objectcatch function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through that the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so that they are oriented at the plane of the drawing object at that the user coordinate system should be adjusted. For example, choose the corner points of this plane with the objectcatch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
> UCS
UCS origin <0,0,0>: 3Point
UCS origin <0,0,0>: P1
Point on the positive x-axis: P2
Point on the positive xy-plane: P3
```

### View

This option adjusts the user coordinate system parallel to the active viewing plane, the screen Through

this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

### **Rotation**

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly

To call up the option rotation, please click onto its field among the options.

With one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around two or three axes, please repeat the following steps for each axis.

1. First determine the axis around that the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z) that will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. Entering or pointing an angle does this.

The angle of the rotation can be determined either by entering it manually or by the pointing of two points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
> UCS
UCS origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z axis <0>: 30
```

### **Controlling the UCS**

Of course it would not be very user-friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore the program offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager that is found when using the standard menu in the pull-down menu view

Also it is possible to open a dialog window by entering the command UCS and choosing the option *Control*.

### **Create a new UCS**

The creation of a user coordinate system is done in two steps

1. Arrange the coordinate system with the previously described options z-direction, 3point, view and/or rotate.
2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate system has been adjusted.

The dialog box has an option that is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and \_ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name, click onto the button **New** and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember that all changes made in the user coordinate system dialog box are registered only after clicking OK as you leave the box.

### **Renaming a user coordinate system**

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click onto the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.

Please remember that all changes made in the user coordinate system dialog box are registered only when you click on OK as you leave the box.

### **Erasing a user coordinate system**

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember that the world coordinate system cannot be deleted. Please remember that all changes made in the user coordinate system dialog box are registered only when you click OK as you leave the box.

### **Changing the active coordinate system**

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.


The **active coordinate system** is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).

1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate system.
3. The field "active user coordinate system" shows the name of the active coordinate system. Please check there that the name of your chosen coordinate system appears there.

Changes to the active coordinate system are registered only when you click OK as you leave the dialog box.

### **Change to the previous coordinate system**

An option exists in the dialog window "User Coordinate System" to change to a previous coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Note: While a brief overview of UCS concepts has been provided here, users are encouraged to consult the **User Manual** , as well as other CAD documentation for a more in-depth exploration of the subject.

## User Coordinate Systems



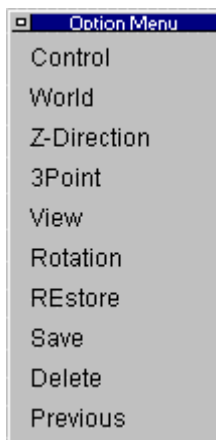
**Command:** UCS

**Alias** UC

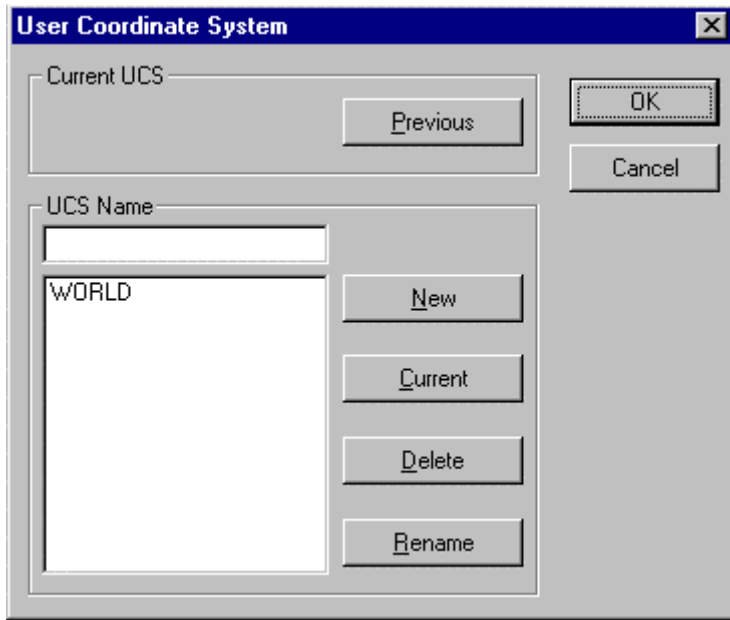
### **Located:**

1. **Pull-down Menu:** Tools – World Coordinate System and  
Tools - User Coordinate System – UCS Manager (UCS\_C) and  
Tools - User Coordinate System – UCS Origin (UCS) and  
Tools - User Coordinate System – Rotate Around Z Axis (UCS\_R-Z)
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

When the UCS command is issued, an option menu is presented that lead directly to individual UCS functions.



The UCS Manager function is available by clicking on the Control option in the menu above. The UCS Manager controls the setup and definition of User Coordinate Systems.



It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate systems will be saved along with the drawing and can be used again when opening the drawing later on.

### Active coordinate system

From all the user coordinate systems in one drawing, only one can be active at any given time.

Choosing a coordinate system to activate is done in a dialog box. Using the same dialog box it is possible to rename, create or delete already existing user coordinate systems.

### Create new user coordinate system

User coordinate systems are always created in two steps. First the parameters of the user coordinate system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system, enter the command UCS. Through the message UCS **origin <0,0,0>** in the text window you are asked to enter the coordinates of the origin of the user coordinate system. The coordinates **<0,0,0,>** are offered as a help. If you don't want to change the origin of the coordinate system then verify this choice through enter. Otherwise please enter new coordinates and verify this entry also with return.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented..

Control    World    Z-direction    3point    View    Rotation

The option **Control** calls up a dialog box that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog box will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option **World** chooses the world coordinate system to be the active coordinate system. This allows you to go back to the world coordinate system quickly without having to open the dialog box of the command control.

### Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .

2. The direction of the axis of the coordinate system can be determined through fixing the z-axis. To do that, choose a point through that the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
> UCS
UCS origin <0,0,0>: Z-axis
UCS origin <0,0,0>: Enter
Point on the positive z-axis: P1
```

### **3Point**

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane, nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the objectcatch function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through that the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so that they are oriented at the plane of the drawing object at that the user coordinate system should be adjusted. For example, choose the corner points of this plane with the objectcatch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
> UCS
UCS origin <0,0,0>: 3Point
UCS origin <0,0,0>: P1
Point on the positive x-axis: P2
Point on the positive xy-plane: P3
```

### **View**

This option adjusts the user coordinate system parallel to the active viewing plane, the screen. Through this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

### **Rotation**

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly.

To call up the option rotation, please click onto its field among the options.

With one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around two or three axes, please repeat the following steps for each axis.

1. First determine the axis around that the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z) that will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. Entering or pointing an angle does this.

The angle of the rotation can be determined either by entering it manually or by the pointing of two

points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
> UCS
UCS origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z axis <0>: 30
```

## Controlling the UCS

Of course it would not be very user-friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore the program offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager that is found when using the standard menu in the pull-down menu view

Also it is possible to open a dialog window by entering the command UCS and choosing the option *Control*.

## Create a new UCS

The creation of a user coordinate system is done in two steps

1. Arrange the coordinate system with the previously described options z-direction, 3point, view and/or rotate.
2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate system has been adjusted.

The dialog box has an option that is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and \_ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name, click onto the button **New** and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember that all changes made in the user coordinate system dialog box are registered only after clicking OK as you leave the box.

## Renaming a user coordinate system

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click onto the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.

Please remember that all changes made in the user coordinate system dialog box are registered only when you click on OK as you leave the box.

### Erasing a user coordinate system

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember that the world coordinate system cannot be deleted. Please remember that all changes made in the user coordinate system dialog box are registered only when you click OK as you leave the box.

### Changing the active coordinate system

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.


The **active coordinate system** is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).

1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate system.
3. The field "active user coordinate system" shows the name of the active coordinate system. Please check there that the name of your chosen coordinate system appears there.

Changes to the active coordinate system are registered only when you click OK as you leave the dialog box.

### Change to the previous coordinate system

An option exists in the dialog window "User Coordinate System" to change to a previous coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Note: While a brief overview of UCS concepts has been provided here, users are encouraged to consult the **User Manual** , as well as other CAD documentation for a more in-depth exploration of the subject.

## User Coordinate Systems



**Command:** UCS

**Alias** UC

### Located:

1. **Pull-down Menu:** Tools – World Coordinate System and  
Tools - User Coordinate System – UCS Manager (UCS\_C) and  
Tools - User Coordinate System – UCS Origin (UCS) and  
Tools - User Coordinate System – Rotate Around Z Axis (UCS\_R-Z)
2. **Left SideBar**
3. **Palette:** View

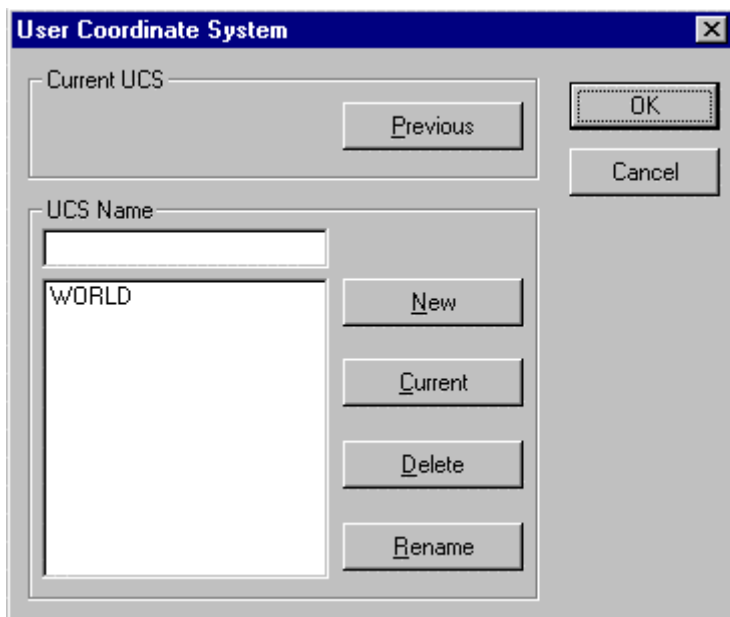


#### 4. Or Enter Command or Alias at the Command Line

When the UCS command is issued, an option menu is presented that lead directly to individual UCS functions.



The UCS Manager function is available by clicking on the Control option in the menu above. The UCS Manager controls the setup and definition of User Coordinate Systems.



It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate systems will be saved along with the drawing and can be used again when opening the drawing later on.

##### **Active coordinate system**

From all the user coordinate systems in one drawing, only one can be active at any given time.

Choosing a coordinate system to activate is done in a dialog box. Using the same dialog box it is possible to rename, create or delete already existing user coordinate systems.

##### **Create new user coordinate system**

User coordinate systems are always created in two steps. First the parameters of the user coordinate

system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system, enter the command UCS. Through the message UCS **origin** <0,0,0> in the text window you are asked to enter the coordinates of the origin of the user coordinate system. The coordinates <0,0,0> are offered as a help. If you don't want to change the origin of the coordinate system then verify this choice through enter. Otherwise please enter new coordinates and verify this entry also with return.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented..

Control      World      Z-direction      3point      View      Rotation

The option **Control** calls up a dialog box that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog box will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option **World** chooses the world coordinate system to be the active coordinate system. This allows you to go back to the world coordinate system quickly without having to open the dialog box of the command control.

### Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .
2. The direction of the axis of the coordinate system can be determined through fixing the z-axis.

To do that, choose a point through that the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
> UCS
UCS origin <0,0,0>: Z-axis
UCS origin <0,0,0>: Enter
Point on the positive z-axis: P1
```

### 3Point

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane, nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the objectcatch function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through that the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so that they are oriented at the plane of the drawing object at that the user coordinate system should be adjusted. For example, choose the corner points of this plane with the objectcatch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
> UCS
```

UCS origin <0,0,0>: 3Point  
UCS origin <0,0,0>: P1  
Point on the positive x-axis: P2  
Point on the positive xy-plane: P3

### View

This option adjusts the user coordinate system parallel to the active viewing plane, the screen. Through this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

### Rotation

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly.

To call up the option rotation, please click onto its field among the options.

With one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around two or three axes, please repeat the following steps for each axis.

1. First determine the axis around that the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z) that will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. Entering or pointing an angle does this.

The angle of the rotation can be determined either by entering it manually or by the pointing of two points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
> UCS  
UCS origin <0,0,0>: Rotate  
Rotate around x-, y-, or z-axis: Z  
Rotation around Z axis <0>: 30
```

### Controlling the UCS

Of course it would not be very user-friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore the program offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager that is found when using the standard menu in the pull-down menu view.

Also it is possible to open a dialog window by entering the command UCS and choosing the option *Control*.

### Create a new UCS

The creation of a user coordinate system is done in two steps.

1. Arrange the coordinate system with the previously described options z-direction, 3point, view and/or rotate.
2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate

system has been adjusted.

The dialog box has an option that is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and \_ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name, click onto the button **New** and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember that all changes made in the user coordinate system dialog box are registered only after clicking OK as you leave the box.

### **Renaming a user coordinate system**

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click onto the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.

Please remember that all changes made in the user coordinate system dialog box are registered only when you click on OK as you leave the box.

### **Erasing a user coordinate system**

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember that the world coordinate system cannot be deleted. Please remember that all changes made in the user coordinate system dialog box are registered only when you click OK as you leave the box.

### **Changing the active coordinate system**

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.


The **active coordinate system** is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).

1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate system.
3. The field "active user coordinate system" shows the name of the active coordinate system. Please check there that the name of your chosen coordinate system appears there.

Changes to the active coordinate system are registered only when you click OK as you leave the dialog box.

### **Change to the previous coordinate system**

An option exists in the dialog window "User Coordinate System" to change to a previous coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Note: While a brief overview of UCS concepts has been provided here, users are encouraged to consult the **User Manual** , as well as other CAD documentation for a more in-depth exploration of the subject.

## User Coordinate Systems



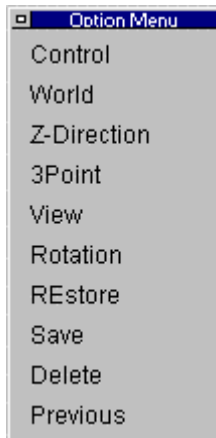
**Command:** UCS

**Alias** UC

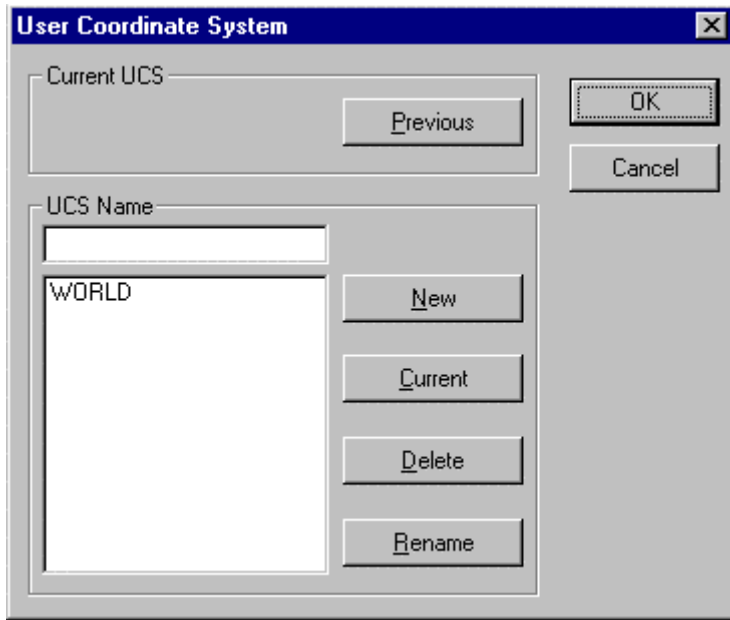
### **Located:**

1. **Pull-down Menu: Tools – World Coordinate System and**  
**Tools - User Coordinate System – UCS Manager (UCS\_C) and**  
**Tools - User Coordinate System – UCS Origin (UCS) and**  
**Tools - User Coordinate System – Rotate Around Z Axis (UCS\_R-Z)**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

When the UCS command is issued, an option menu is presented that lead directly to individual UCS functions.



The UCS Manager function is available by clicking on the Control option in the menu above. The UCS Manager controls the setup and definition of User Coordinate Systems.



It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate systems will be saved along with the drawing and can be used again when opening the drawing later on.

### Active coordinate system

From all the user coordinate systems in one drawing, only one can be active at any given time.

Choosing a coordinate system to activate is done in a dialog box. Using the same dialog box it is possible to rename, create or delete already existing user coordinate systems.

### Create new user coordinate system

User coordinate systems are always created in two steps. First the parameters of the user coordinate system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system, enter the command UCS. Through the message UCS **origin** <0,0,0> in the text window you are asked to enter the coordinates of the origin of the user coordinate system. The coordinates <0,0,0,> are offered as a help. If you don't want to change the origin of the coordinate system then verify this choice through enter. Otherwise please enter new coordinates and verify this entry also with return.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented..

Control    World    Z-direction    3point    View    Rotation

The option **Control** calls up a dialog box that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog box will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option **World** chooses the world coordinate system to be the active coordinate system. This allows you to go back to the world coordinate system quickly without having to open the dialog box of the command control.

### Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .

2. The direction of the axis of the coordinate system can be determined through fixing the z-axis. To do that, choose a point through that the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
> UCS
UCS origin <0,0,0>: Z-axis
UCS origin <0,0,0>: Enter
Point on the positive z-axis: P1
```

### **3Point**

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane, nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the objectcatch function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through that the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so that they are oriented at the plane of the drawing object at that the user coordinate system should be adjusted. For example, choose the corner points of this plane with the objectcatch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
> UCS
UCS origin <0,0,0>: 3Point
UCS origin <0,0,0>: P1
Point on the positive x-axis: P2
Point on the positive xy-plane: P3
```

### **View**

This option adjusts the user coordinate system parallel to the active viewing plane, the screen. Through this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

### **Rotation**

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly.

To call up the option rotation, please click onto its field among the options.

With one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around two or three axes, please repeat the following steps for each axis.

1. First determine the axis around that the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z) that will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. Entering or pointing an angle does this.

The angle of the rotation can be determined either by entering it manually or by the pointing of two

points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
> UCS
UCS origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z axis <0>: 30
```

## Controlling the UCS

Of course it would not be very user-friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore the program offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager that is found when using the standard menu in the pull-down menu view

Also it is possible to open a dialog window by entering the command UCS and choosing the option *Control*.

## Create a new UCS

The creation of a user coordinate system is done in two steps

1. Arrange the coordinate system with the previously described options z-direction, 3point, view and/or rotate.
2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate system has been adjusted.

The dialog box has an option that is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and \_ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name, click onto the button **New** and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember that all changes made in the user coordinate system dialog box are registered only after clicking OK as you leave the box.

## Renaming a user coordinate system

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click onto the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.



Please remember that all changes made in the user coordinate system dialog box are registered only when you click on OK as you leave the box.

### **Erasing a user coordinate system**

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember that the world coordinate system cannot be deleted. Please remember that all changes made in the user coordinate system dialog box are registered only when you click OK as you leave the box.

### **Changing the active coordinate system**

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.


The **active coordinate system** is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).

1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate system.
3. The field "active user coordinate system" shows the name of the active coordinate system. Please check there that the name of your chosen coordinate system appears there.

Changes to the active coordinate system are registered only when you click OK as you leave the dialog box.

### **Change to the previous coordinate system**

An option exists in the dialog window "User Coordinate System" to change to a previous coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Note: While a brief overview of UCS concepts has been provided here, users are encouraged to consult the **User Manual** , as well as other CAD documentation for a more in-depth exploration of the subject.

### **Set Plan View to Construction Plane**



**Command: PLANVIEW**

**Alias PV**

#### **Located:**

1. **Pull-down Menu: Tools – Set Plan View to Construction Plane**
2. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
3. **Left SideBar**
4. **Palette: View**
5. **Or Enter Command or Alias at the Command Line**

The PLANVIEW command sets the view planar to the current User Coordinate System (UCS). Normally, this command is used to return from an isometric 3-dimensional view of the drawing, to the standard (from above) view of the drawing.

---

**See also:**

PLAN

3DVIEW

UCS

UCSFOLLOW

**Variable UCSFOLLOW**

**Alias UF**

**Located:**

1. **Pull-down Menu: Tools – Let View Follow UCS**
2. **Or Enter Command or Command Alias Directly at the Command Line**

Determines, if the alteration of the user coordinates system (UCS) causes an automatic change to the plan view of the new UCS.

**Values**

0	Off: A new UCS does not change the view
1	On: Plan view follows automatically to an UCS alteration

**Properties**

<i>Type</i>	Integer (0   1)
<i>Default</i>	0
<i>Status</i>	Read / Write

**Command**

VIEWPAR

Mouse Context On/Off

**Located:**

1. **Pull-down Menu: Tools – Mouse Context On or Mouse Context Off**

This option, when activated ('ON'), will automatically show any modifications available to any object via an option dialog box, by simply left-mouse clicking on it. It allows you to simply modify objects without searching for the applicable commands.

## Set Layer by Reference

**Command: LAYREF**

**Alias LRF**

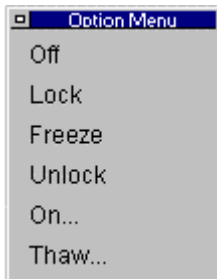
**Located:**

1. **Pull-down Menu: Tools – Layer By Reference**
2. **Or Enter Command or Alias at the Command Line**

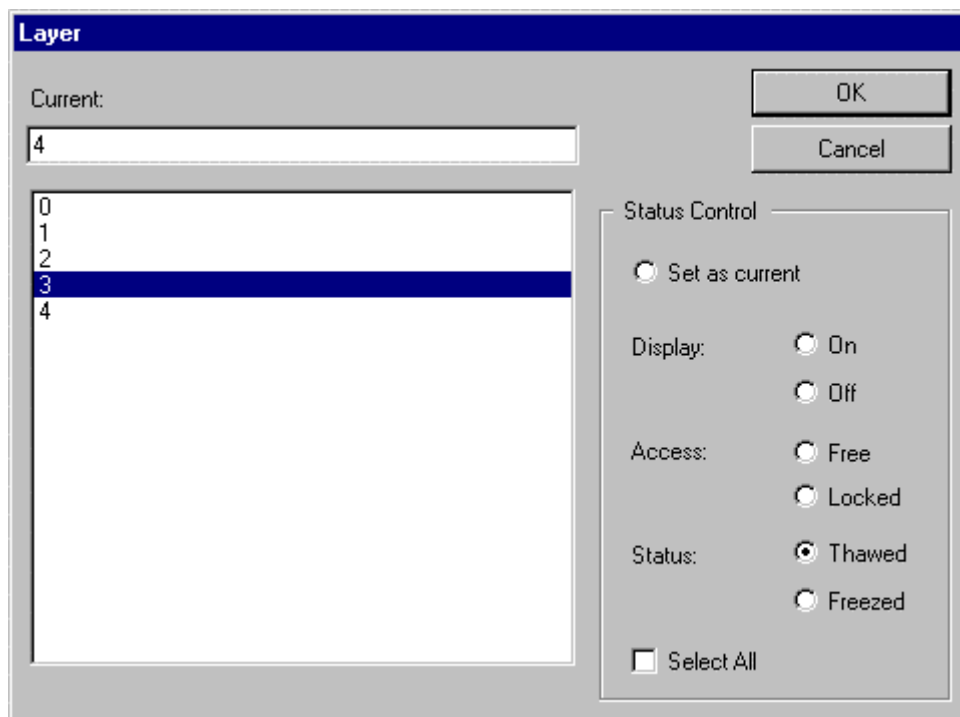
The LAYREF utility provides an alternate means of layer control that is, in some situations, more appropriate and direct. The user can set, turn off, freeze, lock, or unlock a layer by picking a reference entity.

Select the function **Set Layer by Reference** from the Tools pull-down menu or call it by typing LAYREF.

If you have Option Menus or Toolbar set, you will be presented with the following command options:



You simply select the option that you want to perform and then select an object onscreen to indicate the layer that you wish the action to be taken upon. The last two options (On and Thaw) however, would have no visible entities onscreen, so instead, you are presented with the following dialog:



Here, you simply select the layer you want to change the status of (Layer 3 is shown selected in the window), and you simply select the action you want to perform on that layer. As you can see, you can set the layer as current and also switch display, access and status settings.

**Notes:**

1. This latter dialog can be called with any of the original options, simply by pressing enter or right-clicking with the mouse, instead of picking an entity onscreen.
2. The option SET is understood, if you do not choose an option and simply pick an entity after giving this command. The current layer will then become that of the entity selected.

## Command LAYCONF

**Command:** LAYCONF

**Alias** LC

### Located:

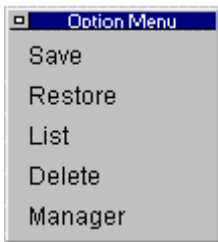
1. **Pull-down Menu:** Tools – Layer Configurations
2. **Or Enter Command or Alias at the Command Line**

The LAYCONF command is used to save and restore layer settings in and from a named status file.

As you work in the program, you will probably add new layers and change layer colors or other parameters using the Layer Manager. The LAYCONF command allows you to save layer settings at any time, in a named setting configuration file, and also lets you return to those saved settings at any later point by specifying the saved configuration file.

Note that the status of newer layers, that are not specified in the named configuration file are not changed in any way by a return to a previous layer status – only layer parameters for layers specified in the saved configuration are changed.

The command LAYCONF is typed in at the command line, at which time you see the option dialog:



You also see a corresponding prompt at the command line:

```
> LAYCONF
```

```
Select option: Save/Restore/List/Delete/Manager:
```

At this point, you can make your selection by picking the option on the dialog with the mouse, or by typing in the first letter of the option you want at the command line.

The **Save** option will save current layer status in the filename that you specify. The default is "LC2" for the first saved file, "LC3" for the second, and so on. You can accept these default filenames by pressing the Enter key, or else you can change the name to another and confirm your choice by pressing Enter.

The **Restore** option is used when you want to revert to a previously saved configuration. You will be prompted for the filename, after which the layer status is changed back. Remember that if you think you will want to return at any time to the status you have prior to the reversion, that you should save that status first in a different configuration file.

The **List** option simply returns a list of all stored configuration files. This can be useful if you have forgotten the filename that you specified earlier when you saved.

The **Delete** option is used to delete a configuration file that you no longer need.

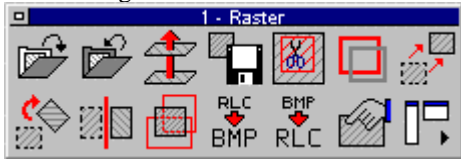
The **Manager** option simply calls the Layer Manager, which allows you to change current layer settings and add new layers.

## Raster Palette

### Located:

#### 3. Pull-down Menu: Tools – Raster Palette

When the raster palette icon is selected on the horizontal toolbar, the Raster Control palette is called onto the drawing editor.



This palette contains icons that control raster image loading, editing and saving functions that have been built into the program.

## Master Palette

### Located:

#### 2. Pull-down Menu: Tools – Master Palette



The Master Palette

The Master Palette selection on the Tools pull-down menu contains pre-configured command buttons for most of the commonly used AllenCad commands. Due to the palettes large size, it would be impractical to

use the palette while editing your drawing, however, it has been provided as a source, should you choose to customize your Horizontal Toolbar or the Left Sidebar on the AllenCad desktop.

By following the instructions in "Reassigning Toolbar Commands/Icons", it is easy to copy these buttons to locations on these toolbars that work best for you.

## Palette Manager

**Command PALMAN**

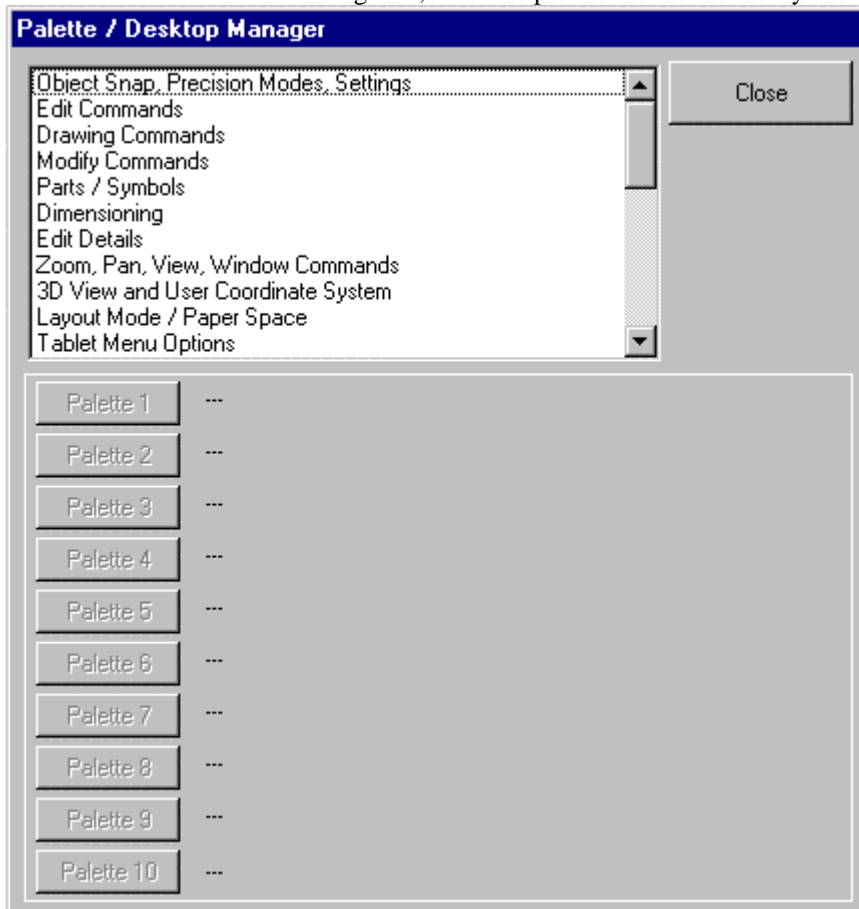
**Alias PM**

**Located:**

- 1. Pull-down Menu: Tools – Palette Manager**
- 2. Or Entered Directly at the Command Line**

Most users will elect to use the palette icons on the Toolbar to activate the various palettes, however others may prefer to select *Palette Manager* from the menu *File* and choose the action that they want in a more interactive fashion.

In the selection area of the Dialog Box, all of the palettes that are currently available are shown:



Select the desired palette. Underneath the selection area are three switches with the designations "Palette 1", "Palette 2", and so forth, to "Palette 10". By selecting one of the ten switches, the user can assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID-number already opened, this palette will be closed and replaced by the new palette having that ID-number. When the palette is opened, the selecting window will close. If no palette is to be opened or replaced the user may close the window by selecting *Cancel*.

If the user wants to include customized palettes in the Palette Manager she/ he should proceed as follows:

Edit the file <Menu name>.pal that is situated in the \SUPPORT directory. For instance, if the user menu is called *Pcad.mnu*, the user must edit the file *Pcad.pal*. The structure of the file is:

Palette description 1  
palette1.mnp  
Palette description 2  
palette2.mnp  
and so forth.

The palette description will be found in the list shown in the Palette Manager Dialog Box, and the line following will always name the palette file. The user can complete or delete the entries as he or she wishes.

## Desktop Configuration

### Command DESKCONF

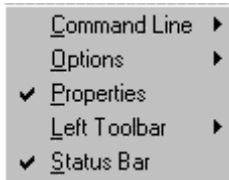
#### Alias DSK

#### Located:

1. Pull-down Menu: Tools – Desktop Manager
2. Or Enter Command or Alias Directly at the Command Line

## Modifying the Desktop Layout

The DESKCONF command's pop-up menu enables you to re-arrange the desktop to suit your needs. This pop-up menu allows you to set various layout options for the programs user interface, including the display of the Command Window, Options Menu or Options Bar, Property Bar, Left Sidebar, and Status Line.



#### The Desktop Layout pop-up menu

**Caution!** The following sections describe how to turn off desktop elements. The user should exercise caution in doing so, however, as these actions will remove visual references to command inputs. If all desktop elements are turned off and the user wishes to reinstate them, the DESKCONF command will need to be retyped and executed with the relevant options selected. Pressing F2 will call the text window where the user can see the command prompt, as needed. Likewise, online Help can be called as needed, by pressing the F1 key.

#### Turning desktop items on and off:

To achieve more space to display drawings on the desktop, or as an advanced user, who is familiar with the names and options of the commands, you may want to turn off certain items of the desktop. You can turn off and on (at runtime) the display of the command line, the Options Menu or context toolbar, the property bar, the left toolbar, and the status bar as the figure above shows.

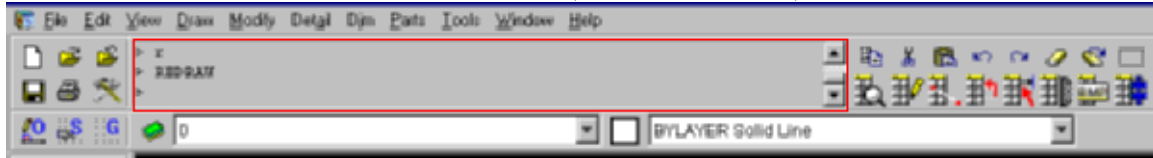
#### How to open the Desktop Configuration pop-up menu:

You can call the *Desktop Configuration* menu by typing the DESKCONF command or selecting the Desktop Configuration selection on the Tools pull-down menu.

You can also open the *Desktop Configuration* pop-up menu by clicking the right mouse button in a "non-functional" area of the program's desktop. Such area is either the background area upon where the drawing viewports are displayed or areas in the property bar, Options Bar, or toolbars that are not covered by buttons or drop-down lists. If necessary, reduce the size of the current drawing(s), cascade the drawing(s), or minimize the drawing(s) to get some free space of the drawing area background. Right-click to this area to open the *Desktop Configuration* menu. You exit this pop-up menu by left clicking to some other area or

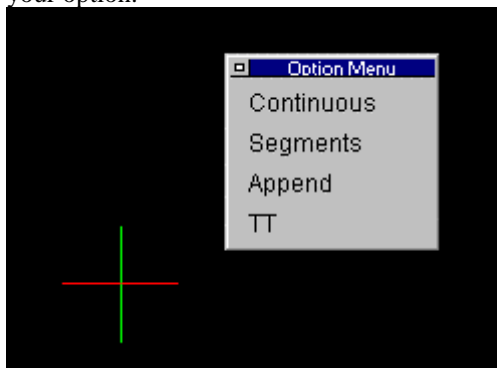
component on the desktop.

The **Command Line** can either be placed at the top of the screen (below the menu bar as pictured below), or in the default location at the bottom of the screen (above the status line).



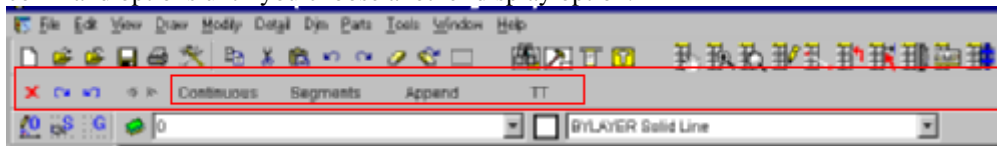
Where you place the bar is strictly a matter of personal preference, as there are no specific advantages in choosing either location. If you are an advanced user, who is familiar with the program's command set, you might even turn the command line off to provide a larger drawing edit window. In this instance, you can always check for program messages or inputs by toggling the text window on/off using the F2 key.

The items of the **Options** sub-menu allow you to determine whether you want to be prompted for command options via an *Options Dialog Menu* or within a *Context Toolbar*. By default, command options are displayed in a pop-up menu that appears in a dialog window in the edit window. These command option dialogs (as pictured below) will disappear or change to offer new options, once you have selected your option.



An Options Dialog Menu - The benefits to this option are that it places the options right on the editor where you are working, and that it doesn't reduce the overall size of your drawing edit window. The drawback is that you sometimes may have to move (drag and drop) the dialog if it is in your way as you draw.

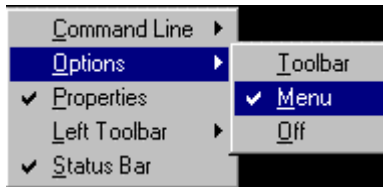
If you choose to display options on an *Options Bar*, you will see a new toolbar inserted between the horizontal toolbar and the Properties Bar, as pictured below. This toolbar will remain open to display command options until you choose another display option.



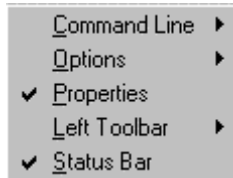
The Options Bar - Note the options presented on the bar.

Note: You can recall the display of the context pop-up menu by selecting the option *Options>Menu* from the *Desktop Configuration* pop-up menu.

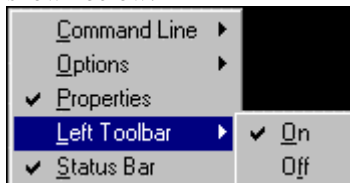




Both the **Properties Bar** and the **Status Bar** can be toggled off and on by clicking on the selection in the Desktop Configuration dialog. When toggled on, the selection will appear with a check mark as shown below:



The **Left Sidebar** can be switched off and on by clicking on the on or off options that appear in a secondary window when you click on the Left Toolbar option in the Desktop configuration dialog, as shown below:



See Also: Emulating the AutoCAD Desktop

## Template Drawings

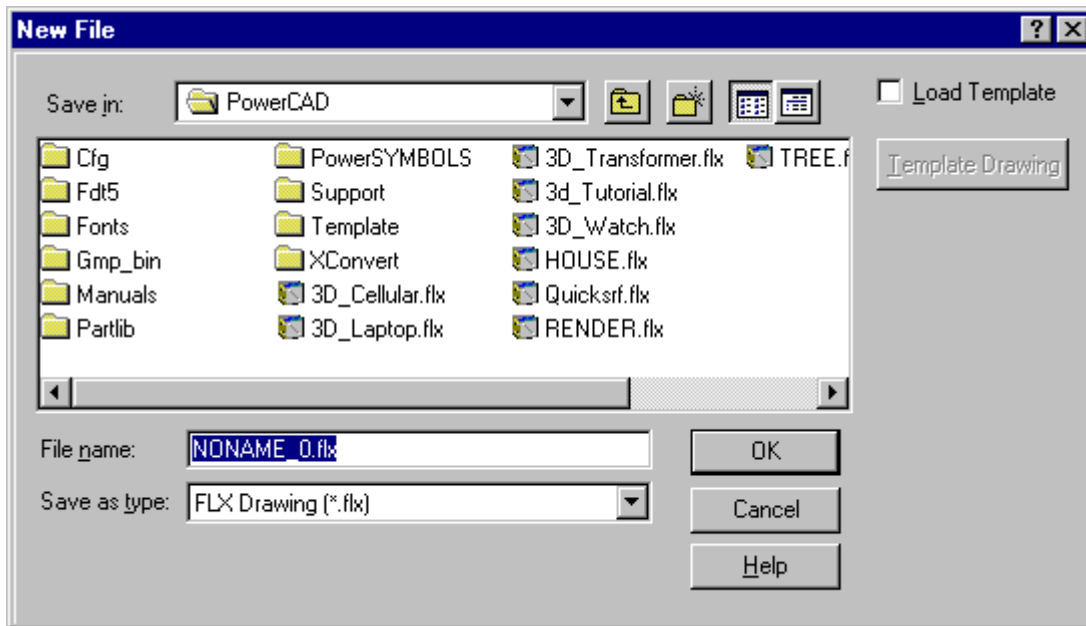
### Command TEMPLATE

#### Alias TP

#### Located:

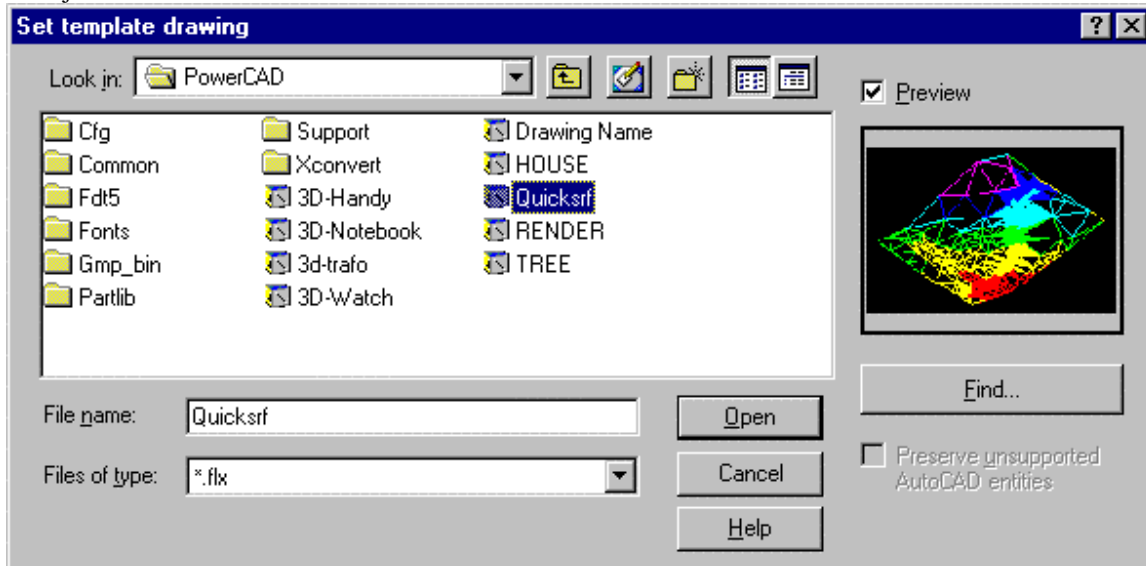
1. Pull-down Menu: Tools – Resource Manager – Set Template Drawing
2. Or Entered Directly at the Command Line

When opening a new drawing you are given the option of using pre-defined templates.



This is the standard file OPEN dialog, where you name your new drawing or choose to over-write an existing drawing with a new blank file. In the top right corner, you see the option to Load Template. If selected, this will load the default template drawing into your new file. If you wish to load another drawing as template, (with Load Template selected) select the Template Drawing button and browse to select another drawing file as template.

To set a new default template drawing, however, you must use the TEMPLATE command from within the program. All subsequent new drawing files will have the option of using this new template that you have just made default.



When you issue the TEMPLATE command, you will be presented with the dialog shown above. You simply navigate to the drawing you want as template, select it and press the Open button. That drawing will then be the new default template drawing until changed, using this same procedure.

#### Notes On Templates:

1. Templates are a form of specimen drawing that either act as a basis for your finished work, or set up the required drawing environment.
2. The main advantage of using templates is that all elements and settings of an already existing drawing can be adopted and as such do not need to be re-entered.

This is particularly useful for parameters such as drawing boundaries, frames, guidelines, measurement settings, views etc. In order to facilitate this, a number of local variables containing stored settings are loaded along with the template. These settings are then used to display the drawing, drawing elements and objects.

For a listing of commands that can be used in template drawing setup, please refer to Drawing Setup. Alternatively, if you have a drawing file that has all your standard setup elements in it, you can simply delete any drawing elements you do not wish to have in your template drawing and SAVEAS a new drawing file, to be used as the template.

Further information on the use of template drawings can be found in Chapter 4 of the Customization and Programmer's Guide.

## Command UNITSETUP

**Command:** UNITSETUP

**Alias** US

**Located:**

**Located:**

1. **Pull-down Menu: Tools – Default Unit Setup Precision**
2. **Or Enter Command or Alias at the Command Line**

This command is used in exactly the same way as the command SETUP. It is used to set or change the drawing unit display parameters for the current drawing. Please click on this link for a detailed description of its use.

## Part Library Setup...



**Command:** PLBSETUP

**Alias** PLB

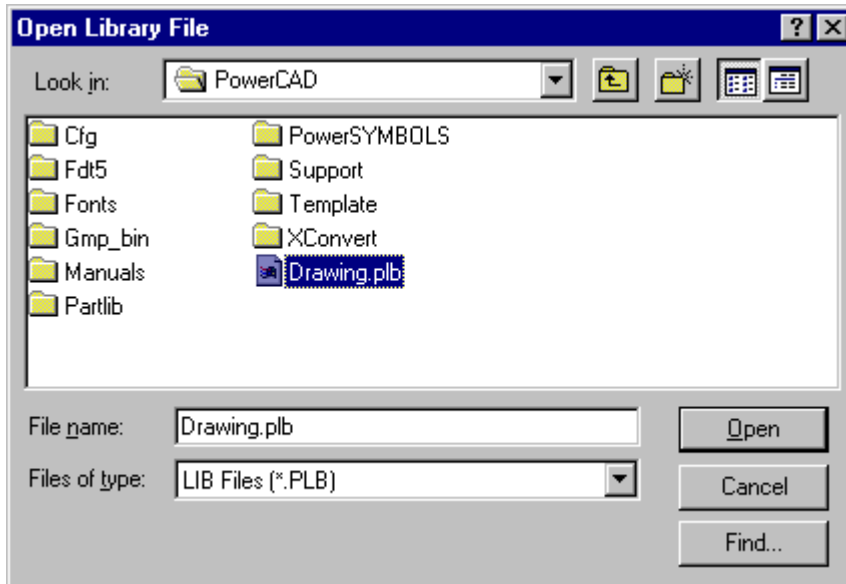
**Located:**

1. **Pull-down Menu: Tools – Resource Manager – Part Library Setup**  
**Also - Parts – Part Library Setup**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

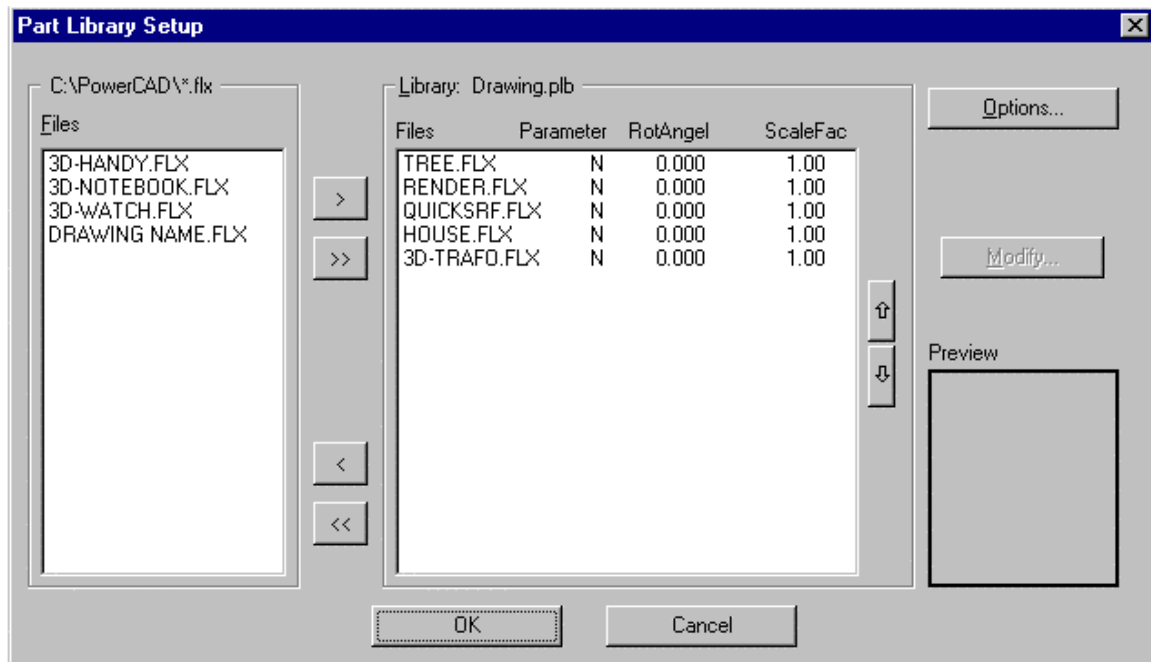
The command PLBSETUP (Part Library Setup) allows the user to setup up new part libraries and/or modify existing part libraries.

The command provides dialog boxes that:

- Select the drawing files to be included in a library;
- Set preferences for part or symbol insertions from the library; and
- Set the parameters that determine the layout and options of the part library Dialog Box.



PLBSETUP creates **.plb** files. These files store the parameters for a single part library. A part library is called by using the command PARTLIB, and then specifying the name of a .PLB file in the above dialog. No knowledge of structure or syntax of the .PLB format is required, when using the PLBSETUP command. Pressing the Open button opens up the next dialog:



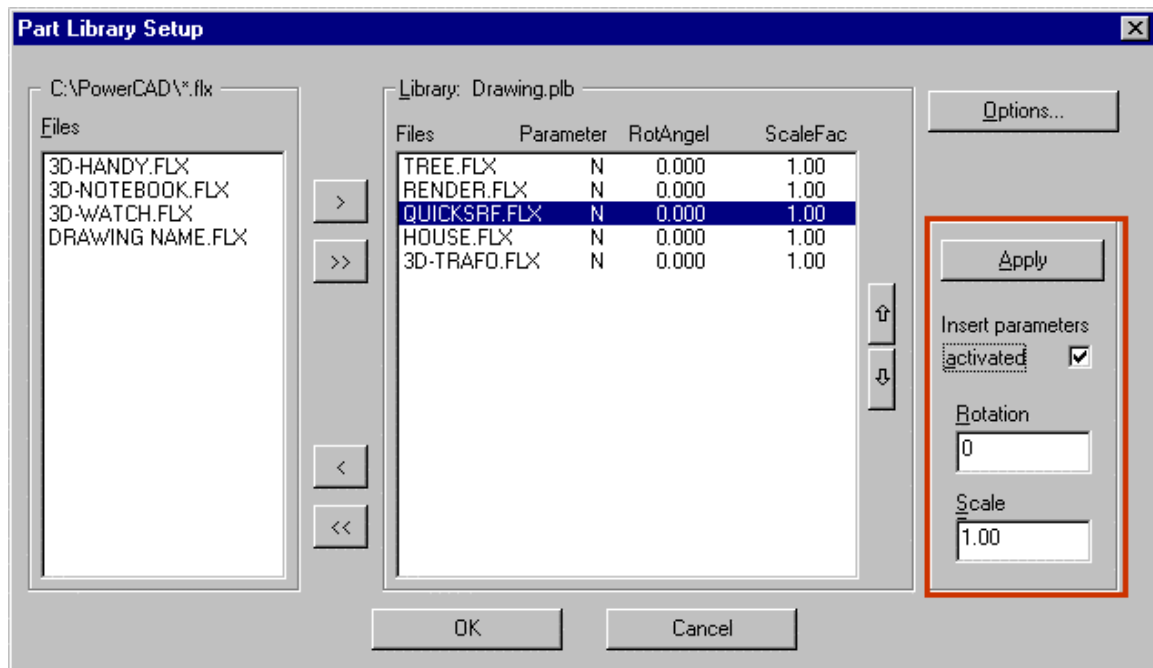
### Specifying the Drawing Files Belonging to a Part Library

The dialog box *Part Library Setup* allows the user to specify the drawing files to be incorporated into the individual library. He or she can **only choose drawing files from the programs Partlib subdirectory**. It is not necessary to include all of the drawings in a folder – simply select those that you want for the specific library that you are creating. Libraries allow the user to maintain all of the drawings used for part or symbol insertions within one directory, and at the same time permits the creation of multiple "thematically separated" libraries.

To add drawings to the current library, highlight the file names in the left side file list of the dialog box and then click to the button > to insert them to the file list making up the library (shown in the right frame). The buttons ( < and > ) permit both addition and removal of drawing files from the list of part library files. When only one specific file is selected from either of the list boxes, the preview image of the drawing will be displayed in the Preview Area. The buttons >> and << allow the user to easily move all file names from one list to the other.

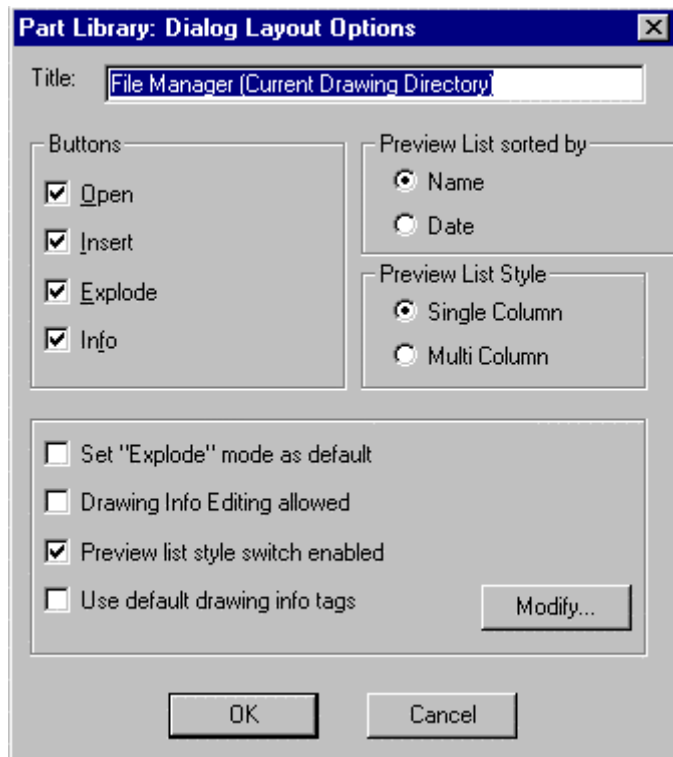
### Insertion Parameters

It is possible to pre-define insertion parameters for the files added to a library. Select one, more than one, or all entries from the list of the library files and then select **Modify**. Next, activate the option **Activate insertion parameters** and enter the default values of the *Scale Factor* and the *Rotation Angle* for insertion in the section outlined in red, below.



### Part Library Dialog Box Layout and Options

Using the Part Library **Options** dialog box, the user can determine the layout and the available buttons and options of the dialog box that comes up when calling this individual library with the PARTLIB command:

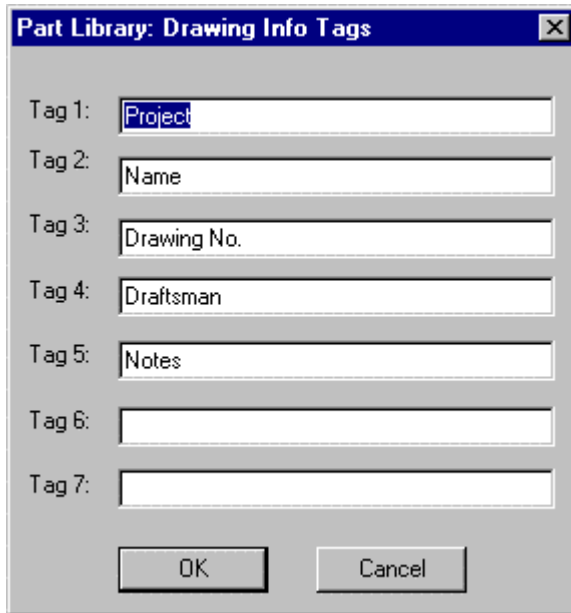


The edit box **Title** allows the user to enter a name for the dialog box of the part library.

#### Preferences for the Preview List

- **Preview List Style** lets the user specify whether the part library Dialog Box should start up with a single column or multi column style of the list showing the drawing previews)
- **Sort by...** helps the user determine whether the drawing preview list should be sorted by name (usually determined for part libraries) or by date (usually determined for project libraries).
- **Buttons** assists the user in deciding options provided in the Dialog Box for a specific Part Library.
- **Open** allows the user to open a selected drawing for modification.
- **Insert** allows the user to insert a selected drawing into the current drawing as part.
- **Explode** allows the user to specify whether the part should be inserted as complex object (block), or exploded (broken down to single entities).
- **Info** allows the user to open the Drawing Information sub-dialog of the specific library.
- **Set "Explode" mode as default** sets the preference (activated/deactivated) for the mode of the button Explode.
- **Allow drawing info editing** specifies whether the user may alter drawing file information stored with the drawing database.
- **Enable preview list style switch** determines whether the library Dialog Box should contain the Quick Mode / Extended Mode button that, in turn, allows the user to toggle between single column and multi column style in the file preview list.
- **Use default info tags** determines whether default info tags (Project, Name, Drawing No., Draftsman, Notes) or user defined info tags are used when drawing information is displaced in the Drawing Information Dialog Box of the PARTLIB command.
- **Modify**, in the preferences area, allows the user to determine individual tags for the file info fields

used in the specific library (for example; manufacturer, price, etc. in a library managing purchased parts). The dialog that appears with the Modify option is shown below:



The dialog box is titled "Part Library: Drawing Info Tags" and contains seven text input fields labeled Tag 1 through Tag 7. Tag 1 contains the text "Project", Tag 2 contains "Name", Tag 3 contains "Drawing No.", Tag 4 contains "Draftsman", and Tag 5 contains "Notes". Tags 6 and 7 are empty. At the bottom of the dialog are "OK" and "Cancel" buttons.

---

**See also:**

Part /Symbol Libraries

Command SETLIB



**Command: SETLIB**

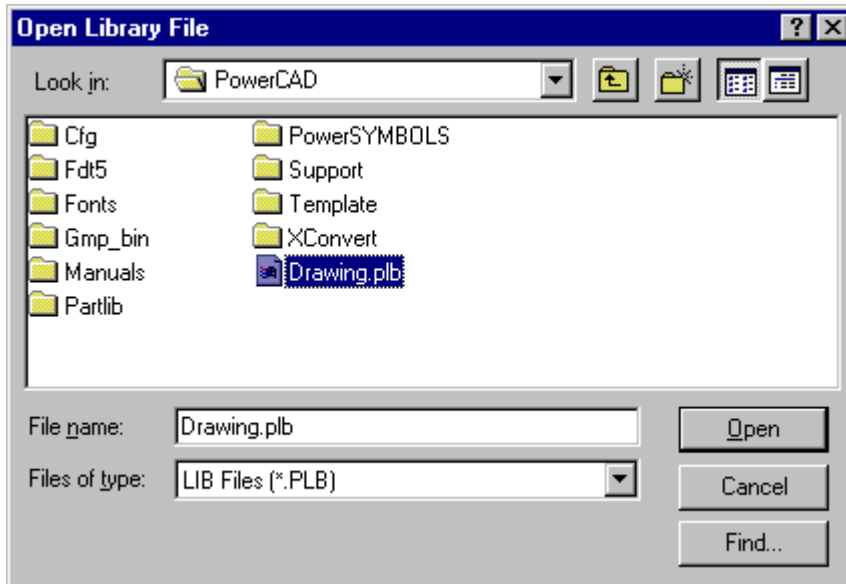
**Alias STB**

**Located:**

- 1. Pull-down Menu: Tools – Resource Manager – Set Default Part Library**  
**Also - Parts – Set Current Library**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias at the Command Line**

The LIBRARY command displays the Set Current Part Library / Symbol Library file dialog box. Select a part library (.plb) file.

The **SETLIB** command, on the other hand, sets the default library to be loaded whenever LIBRARY is executed. Simply select the .plb file you want as default in the following dialog:



Alternately, enter the PARTLIB command, which prompts you for the name of a part library at the command prompt. This command is meant for use with macros and FLISP functions. Enter the name of a .plb file, such as:

> partlib

File name: c:\AllenCad\partlib\partlibdrawing.plb

Once a part library has been chosen, the Library command continues to use that same library as default. Use the **SETLIB** (Part > Set Current Library) to change the part library.

### Load Palettes...



**Command: 'PALETTE'**

**Alias PLT**

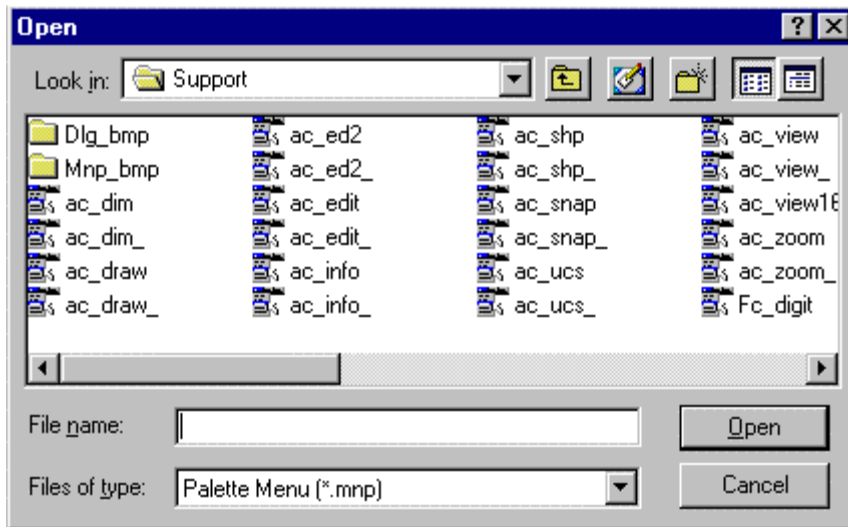
#### **Located:**

- 1. Pull-down Menu: Tools – Resource Manager – Palette...**
- 2. Or Enter Command or Alias at the Command Line**

*Load Palette...* serves to open and place a Palette onto the program desktop. From the Open File dialog box that is opened the user can select the desired palette file to be loaded.

When you issue the command, a dialog box is presented, allowing you to select the palette file (\*.mnp) that corresponds to the palette you want to load. Simply select the file, and the palette will be placed on the desktop.





Every palette opens in its own window. In the title bar of the palette (toolbox) the user will find the name of the palette as well as an ID-number.

Palettes may also be opened through individual selections located on the pull-down menus, or using the palette icons located on the toolbar, as pictured in the image below:



Further information on creating and editing palettes can be found in Chapter 7 of the **Customization and Programmer's Guide**

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#### See also:

- Exchange Palettes
- Close Palettes

## Load Menu



### Command: 'MENU

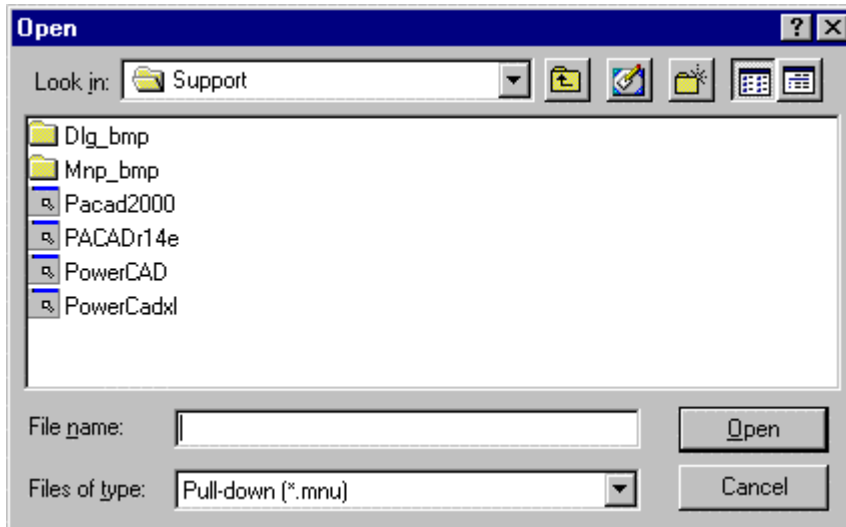
Alias MU

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Pull-down Menu**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The program is designed so that it is possible to work with different pull-down menus. AllenCad comes complete with both it's own AllenCad menu, as well as AutoCAD and FelixCAD emulation menus. Although the AllenCad menu offers the greatest functionality, users more familiar with other CAD programs may prefer to use the emulation menus provided.

The pull-down menus are stored in menu files with the extension **.mnu**. To switch to using another stored menu file, simply execute the MENU command and select the file from the listing.



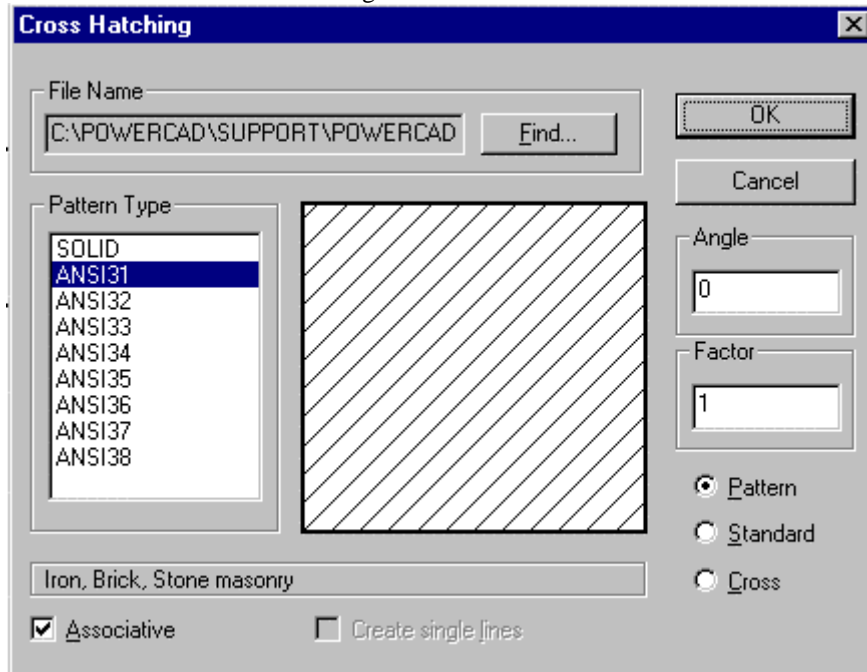
## Icon Menu

### Command ICONMENU

#### Located:

#### 1. Pull-down Menu: Tools – Resource Manager - Icon Menu...

The Icon menu command is a programming tool that presents the user with selections that are supported by images. Icon menus use groups of images as graphic labels. Icon menus also have a list box containing text labels associated to each image. This combination allows the user to select options



Icon menus are a valuable visual tool, as in the example above, when selections of hatch patterns or text styles should be customized. Also, icon menus are can be useful in creating symbol library selection menus.

With this command, a dialog presenting an assortment of graphical options can be created. A slider bar is added automatically to the dialog if the number of images exceeds display limits.

You can customize the number of columns and rows to be displayed for each icon menu.

## Desktop and Menu Editor

### Desktop and Menu Editor – Command DLGEDIT

#### Alias DM

##### Located:

1. **Pull-down Menu:** Tools – Resource Manager – Run Desktop and Menu Editor
2. **Or Entered Directly at the Command Line**

The Desktop and Menu Editor (DME) design and lay out your menus and palettes in a visual environment. Knowledge of programming is not required.

Please refer to the Command Button Chart for input information when modifying palettes.

For more information on the DME click here: DME Help 

## Image Library Tool

### Command IMAGLIB

#### Located:


1. **Pull-down Menu:** Tools – Resource Manager – Run Image Library Tool
2. **Or Entered Directly at the Command Line**

The Image Library Tool assembles and organizes images that can be used in several ways. Icons and other screen grabs incorporated in image libraries can be displayed in icon menus, toolbars, palettes, and application dialog boxes. Image libraries created and maintained with the ImageLib program are stored in files with extension *.ilb*.

AllenCad's image libraries in are similar in nature to AutoCAD's slide libraries. You can import entire slide libraries (*.slb*) from AutoCAD into a AllenCad image library. But AllenCad's Image Library Tool provides features not available in AutoCAD's slide libraries:

- Beneath slides, you can store bitmaps (*.bmp*) and Windows Metafiles (*.wmf*) in image libraries.
- You can use icons and images from your libraries in icon menus, dialog boxes, palettes, and toolboxes.
- The ImageLib program is a Windows program with easy-to-use import and export of image files.

To start the ImageLib program, select File > Resource Manager > Run Image Library Tool from the AllenCad menu bar.

For more help with the Image Library click here: IMAGELIB HELP 

## Command BMPEDIT

### Command: BMPEDIT



#### Alias BPE

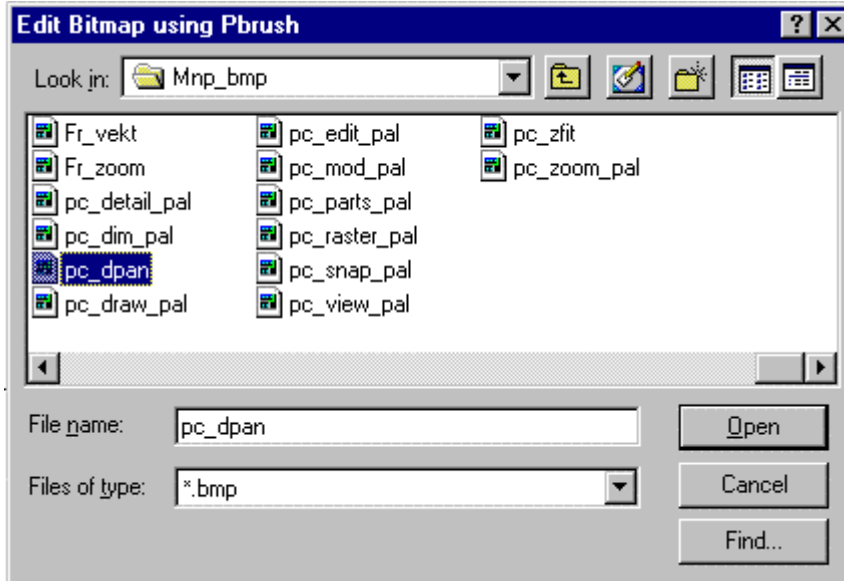
##### Located:

1. **Pull-down Menu:** Tools – Resource Manager – Run Bitmap Editor
2. **Palette:** Raster

### 3. Or Enter Command or Alias at the Command Line

This command allows the user to edit an external Bitmap graphic (.bmp), using the Windows Paintbrush program. In this way, you can make necessary changes to the graphic before importing it into AllenCad, without ending your AllenCad session.

This command can be run from the "Resource Manager - Run Bitmap Editor" selection on the Tools pull-down menu or it can be typed at the command line. The following dialog appears onscreen once the command is issued, allowing you to browse and locate the bitmap file that you need to edit.



The bitmap is opened in the Paintbrush program once the selection has been made. Please consult Windows' documentation for instructions on the use of the Paintbrush program.

## Command APPLOAD

**Command: APPLOAD**

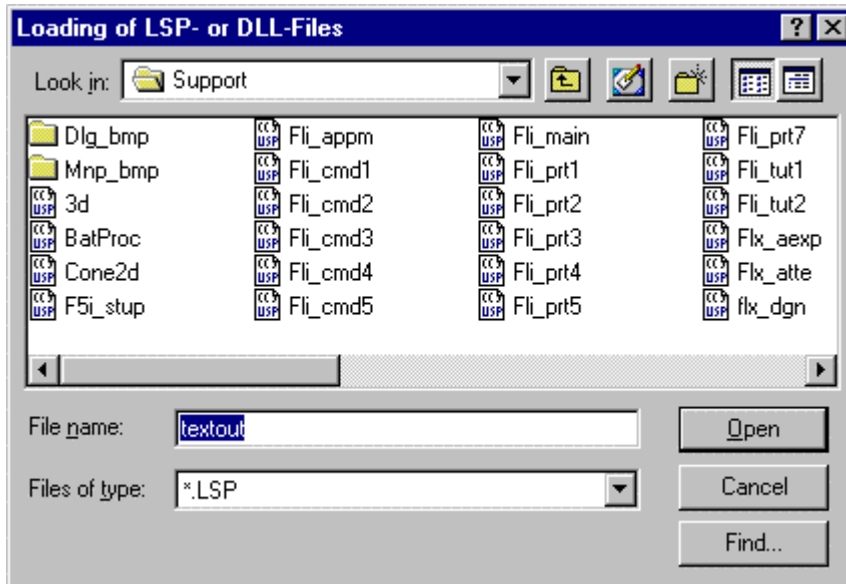
**Alias AP**

**Located:**

1. **Pull-down Menu: Tools – Resource Manager – Load Application**
2. **Or Enter Command or Alias at the Command Line**

The **APPLOAD** command is used to locate and load either LISP programs or .DLL (Dynamic Link Library) files for use in the program. The command is issued from either the "Resource Manager - Load Application" selection on the Tools pull-down menu, or else it can be entered at the command line.

The following dialog lets you choose between LISP and DLL file selection and offers an opportunity to browse and select your file from the dialog display.



This command performs very much the same function as the LOADLISP command, except that it allows you to load LISP and DLL Files, as you may require.

**Note:** Be sure to check the command line as the LISP file is loaded. There you will see the command name to enter whenever you want to run the routine during the current edit session. The example below indicates the command name to use when the "Textout.lsp" file is loaded.

```

Loading D:\FCAD2000\SUPPORT\TEXTOUT.LSP...Command TEXTOUT loaded.
Done.nil
> |

```

A DLL when loaded, however, only gives a confirming message that the load was successful:

```

> APPLOAD
Loading D:\fcad2000\SUPPORT\Rejoin.dll .....OK.
>

```

## Command LOADLISP

**Command:** LOADLISP

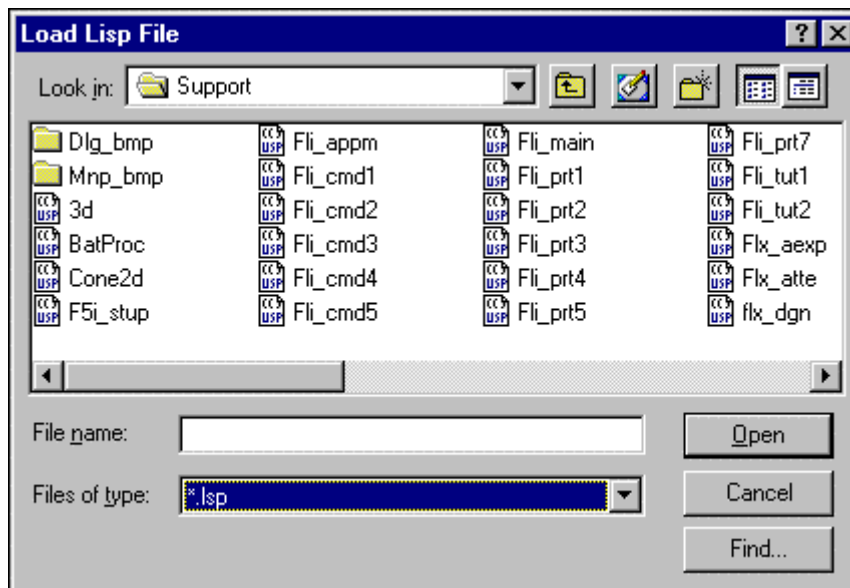
**Alias** LL

**Located:**

1. **Pull-down Menu: Tools – Resource Manager – Load Lisp File**
2. **Or Enter Command or Alias at the Command Line**

The **LOADLISP** command is used to locate and load LISP programs for use in the program. The command is issued from either the "Resource Manager - Load Lisp file" selection on the Tools pull-down menu, or else it can be entered at the command line.

The following dialog offers an opportunity to browse and select your file from the dialog display.



**Note:** Be sure to check the command line as the LISP file is loaded. There you will see the command name to enter whenever you want to run the routine during the current edit session. The example below indicates the command name to use when the "Textout.lsp" file is loaded.

```

Loading D:\FCAD2000\SUPPORT\TEXTOUT.LSP...Command TEXTOUT loaded.
Done.nil
> |

```

## Command RUNMACRO

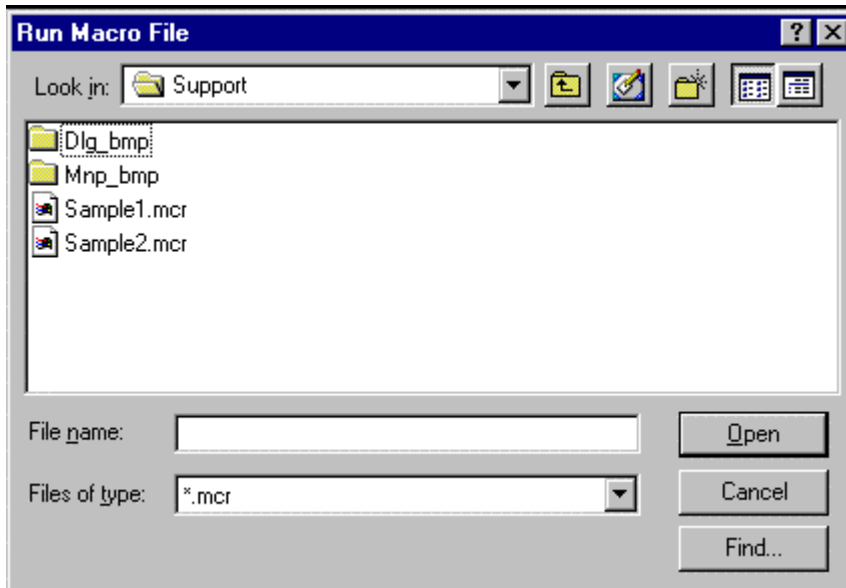
### Command: RUNMACRO

#### Located:

1. Pull-down Menu: Tools – Resource Manager – Run Macro
2. Or Enter Command or Alias at the Command Line

Unlike a LISP program that remains in memory for repeated use, Macros (sometimes referred to as Script files) are executed as they are loaded. The RUNMACRO command can be found on the Tools pull-down menu as "Resource Manager – Run Macro" selection, or else it can be simply typed at the command prompt for execution.

The following dialog appears once the command has been issued, allowing you to browse to the macro file you want.



The macro is executed as soon as selection has been confirmed.

## Command LISTFILE

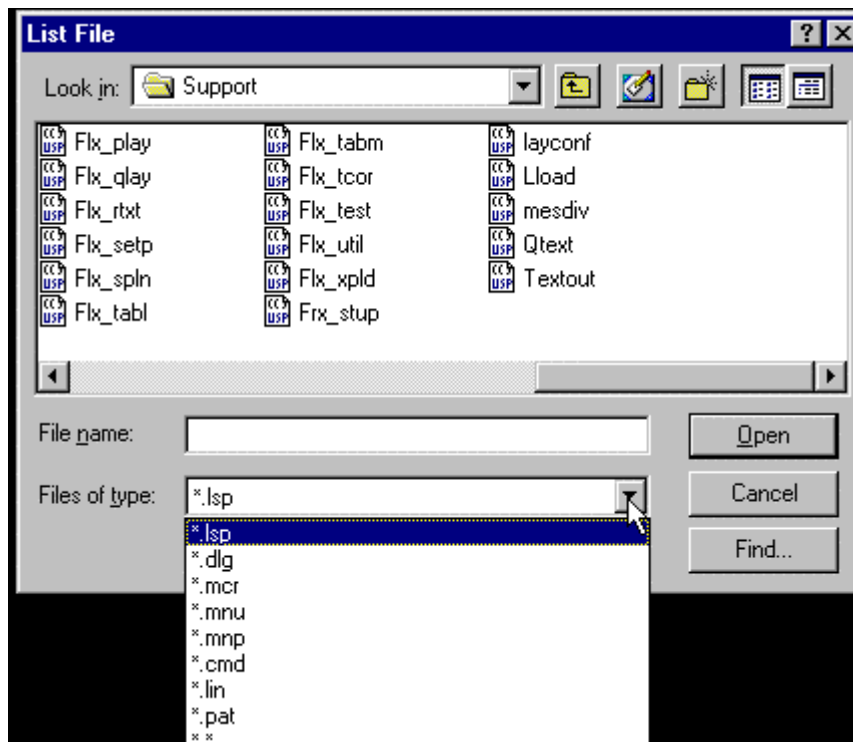
**Command: LISTFILE**

**Alias LF**

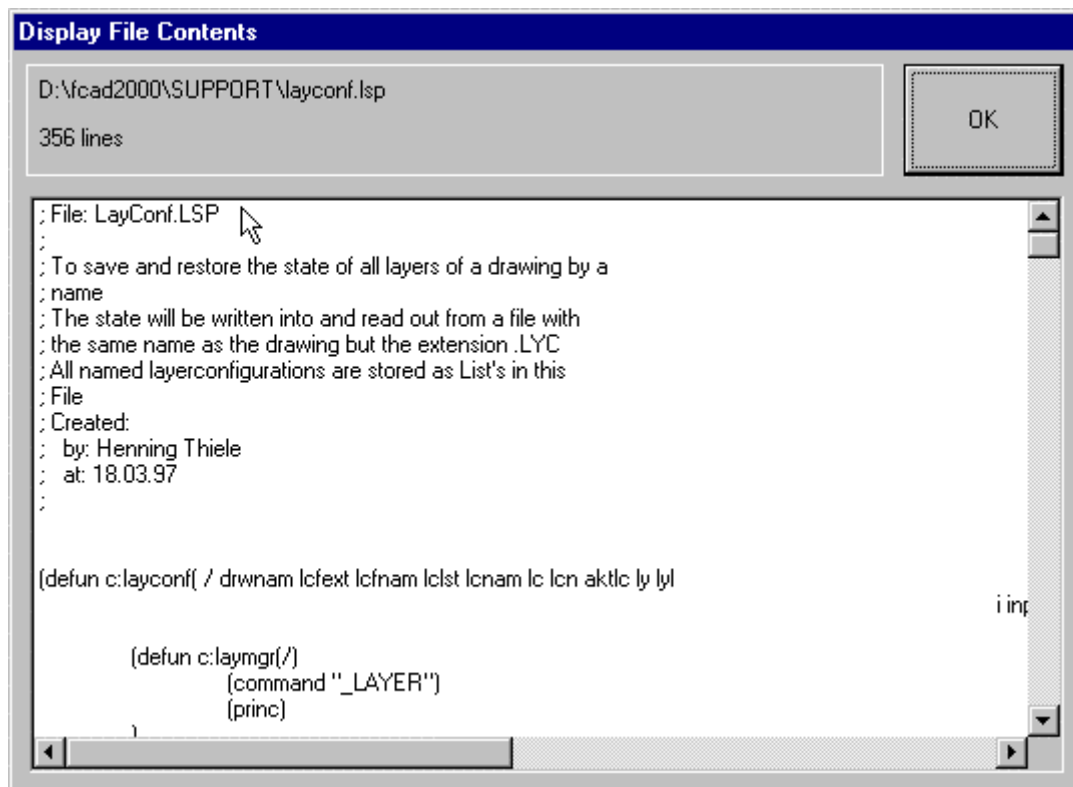
### Located:

1. **Pull-down Menu: Tools – Resource Manager – List File**
2. **Or Enter Command or Alias at the Command Line**

The LISTFILE command, located on the Tools pull-down menu as "Resource Manager – List File..." is used to open and inspect file contents. The command can also be entered at the command line, if preferred. This can be useful to confirm contents before the file is used in the program, for instance. As the dialog below indicates, the most common file types that are used by AllenCad are available for selection display, along with the "wildcard" \*.\* option that will display all files.



Below, you see the window that opens showing the file contents after selection has been made. Click on OK to close the window when you are finished and wish to return to the drawing editor.





## Command EDITOR

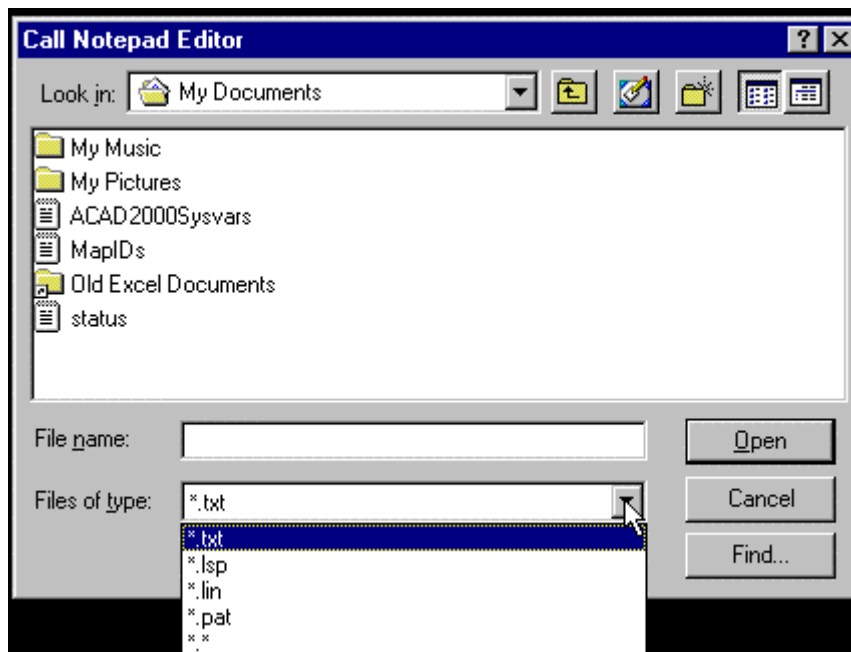
### Command: EDITOR

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Editor**
2. **Or Enter Command or Alias at the Command Line**

The EDITOR command is used to call a file into Window's Notepad program, to allow you to make file edits while working in AllenCad. This can be handy for such tasks as modifying LISP routines, writing Macros or editing a text file prior to importation into AllenCad. The command is located on the Tools pull-down menu under "Resource Manager – Editor" and can be also run at the command line.

Upon execution, you are presented with the following Dialog:



Notice that it allows you to select certain file types that are commonly used by the program to assist in file selection. You can also select \*.\* as an option, to display all files types. After selecting the file type, you can simply browse to locate the file you wish to edit. After selecting the file and pressing the Open button, the file is opened in Notepad for editing in a new window.

## Toggle Orthogonal Mode



**Command: TORTH0**  
**Alias TO**

#### Located:

1. **Pull-down Menu: Tools – Set Drawing Modes – Orthogonal Mode**
2. **Properties Bar**
3. **F-8 Function Key (by default)**

#### **4. Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Orthogonal Mode ON / OFF. When Ortho mode is on, lines that you draw will be forced parallel to the axes (IE straight up and down or sideways).

#### **Toggle Snap Grid Mode**



**Command: TSNAP**

**Alias TS**

#### **Located:**

- 1. Pull-down Menu: Tools – Set Drawing Modes – Snap Grid Mode**
- 2. Properties Bar**
- 3. F-9 Function Key (default)**
- 4. Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Snap Grid ON / OFF. With snap grid on, you can quickly and easily snap to points that are established at grid intersections. This mode is operational, with or without the visual representation of the grid, that is controlled by the TGRID command.

See command PRECPAR for grid setup.

See Also Snapmode

#### **Toggle Grid Display Mode**



**Command: TGRID**

**Alias TG**

#### **Located:**

- 1. Pull-down Menu: Tools – Set Drawing Modes – Grid Reference Mode**
- 2. Properties Bar**
- 3. F-7 Function Key (default)**
- 4. Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Grid Display ON / OFF. The grid can be used as a visual reference to points that are established parallel to the axes and a set distance apart. IF Snap is also switched on, these grid points can be quickly and precisely selected onscreen, while working on your drawing.

See command PRECPAR for grid setup

#### **Precision Aids Parameters**

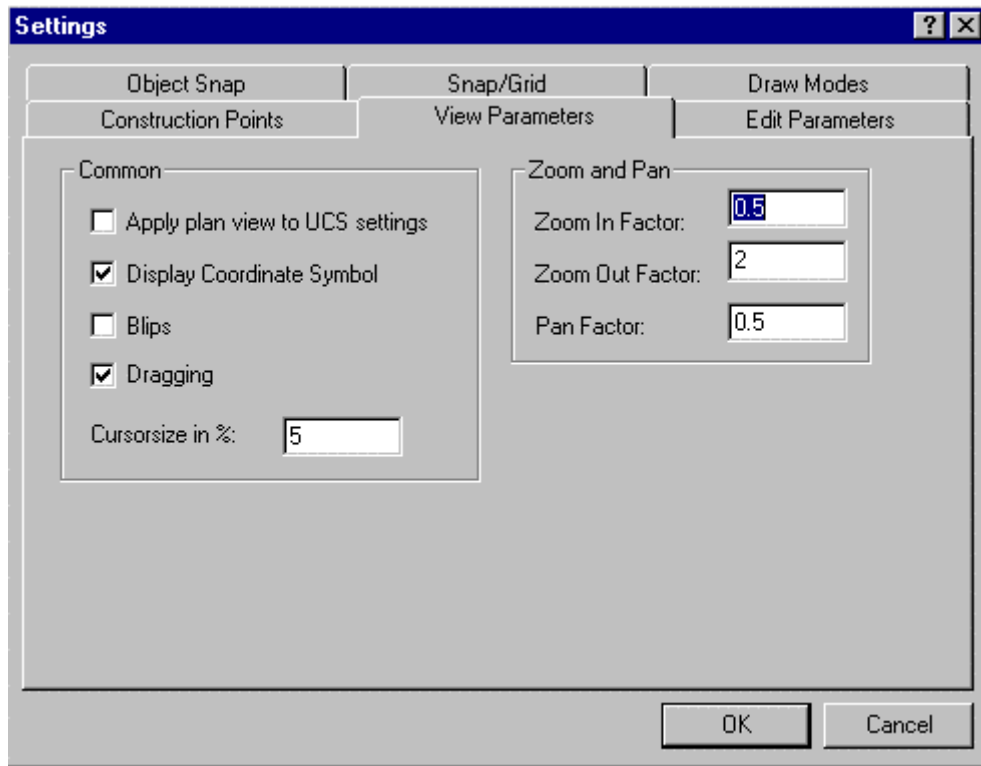
**Command: PRECPAR**

## Alias PCP

### Located:

1. **Pull-down Menu: Tools – Set Drawing Modes – Drawing With Precision**
2. **Or Enter Command or Alias Directly at the Command Line**

The PRECPAR command leads directly to various sections of the SETTINGS dialog, that control selection precision. Each tab has settings that can be adjusted to control drawing precision.



### Grid Display

Activate or de-activate the grid by clicking the check window on the upper left in the box **Grid Spacing**. A cross in this check box indicates the grid display is active.

In the input area **Factor** enter the value that defines the relation of the grid points to the snap grid. Alternatively, the user can specify a predefined value by clicking available buttons.

### Snap Grid

Activate or de-activate *Snap Grid* by clicking the check box on the upper left side in the area Snap Grid. A cross in the check box signals that the snap grid is switched on. From the dialog box, select the input fields Width and Height, and enter values for the horizontal and vertical distances between the points of the snap grid. Units of measurement for these values are, again, drawing units. The value 1.00 represents the distance of one drawing unit.

Using the palette button ( = ) for the value Width enables the user to choose a vertical value equal to the horizontal value.

The input fields marked "Origin" in this box enable the user to define the origin of the snap grid on the X and the Y-axis. The desired coordinates for the origin should always be entered.

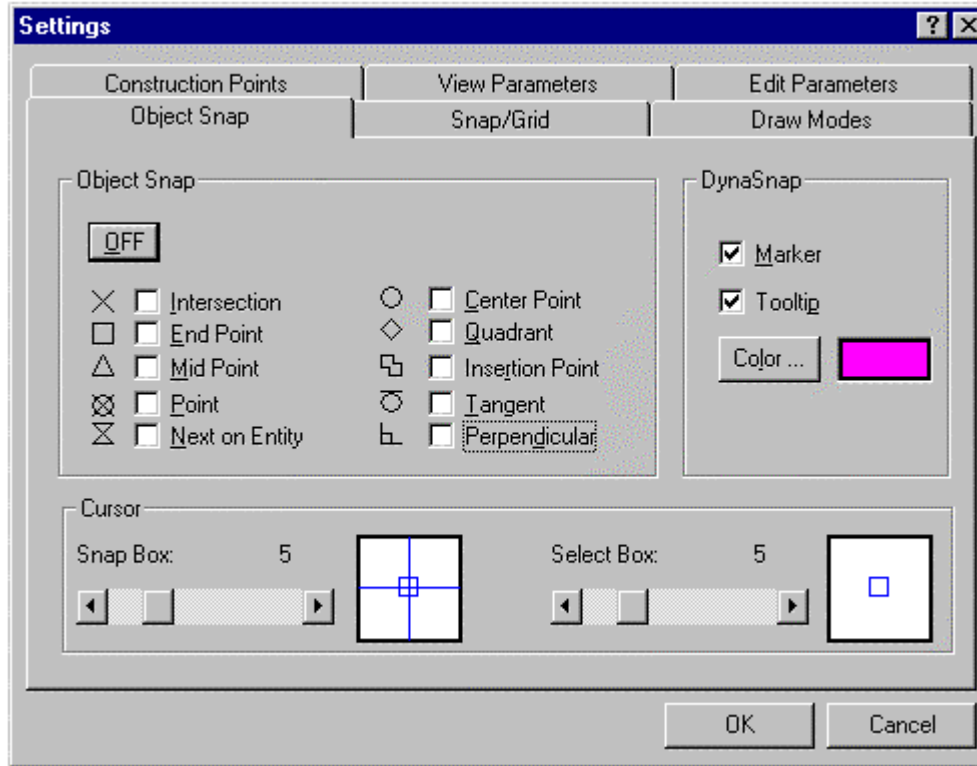
Sample

### Orthogonal Mode

Activate or de-activate **Orthogonal Mode** by clicking the check box.

ORTHOgonal MODE: Drawing and Modifying Objects

### Object Snap



The Object Snap tab sets the active object snaps and controls whether dynasnap (the symbols for each are shown beside each object snap selection box) and tooltips are displayed by the program, or not. The color of both the dynasnap markers as well as tooltip color is also set on this dialog.

In the lower part of the dialog, you may specify the size (in pixels) of both the cursor snap and selection box.

### Cursor

Object Snap tab allows specifying the Select Box and Snap Box size.

View Parameters tab lets you control the size of the cursor axes.

## Command SETUP

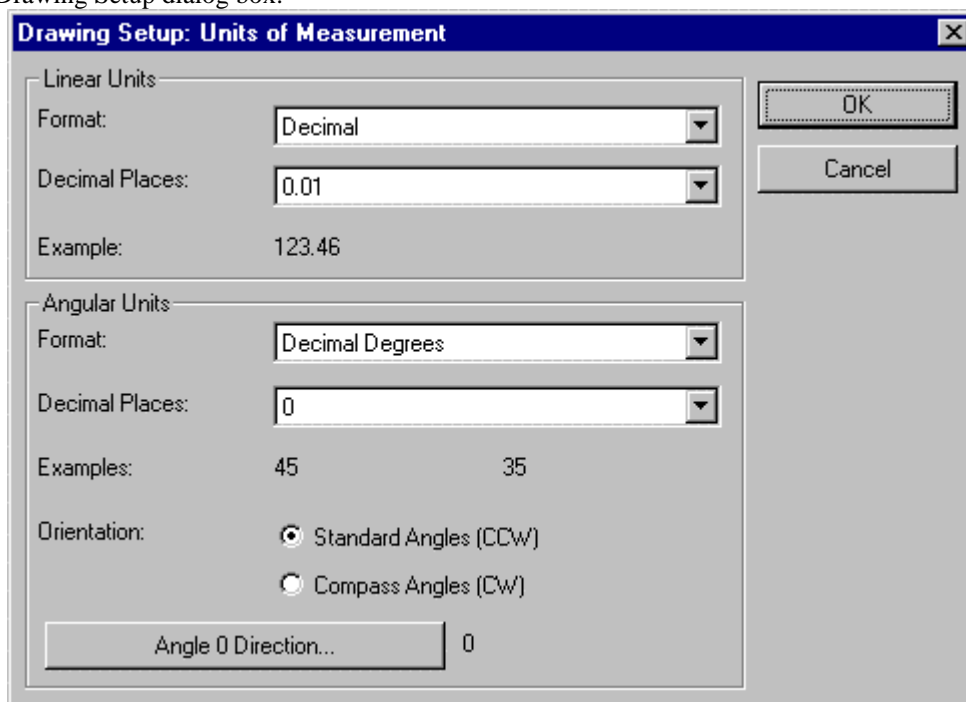
### Command: SETUP

Alias US

#### Located:

1. Pull-down Menu: Tools – Drawing Setup Parameters
2. Or Enter Command or Alias at the Command Line

AllenCad allows you to set the format and display accuracy of numbers and angles. The settings are saved with each drawing, so they can be set differently in each drawing. The Setup command displays the Drawing Setup dialog box:



The command UNITSETUP is identical to this command and has been provided for the convenience of users familiar with AutoCAD commands.

### Linear Units

Linear units can be given a display accuracy ranging from zero to eight decimal places. In the case of fractional units, the display accuracy ranges from 0 (no fractions) to 1/256. You can select five kinds of units:

- Decimal units (metric) - 12.3456.
- Fractional units (unitless fractions) - 12 3/4
- Architectural Units (feet and fractional inches) - 12'-3/4" (See note below)
- Engineering units (feet and decimal inches) - 12-3.456'
- Scientific units (exponential notation) - 1.234E+02

### Angular Units

The Angular Units can be given a display accuracy ranging from zero to eight decimal places. AllenCad recognized these angular measurement systems:

- Decimal degrees - 12.3456
- Degrees - minutes - seconds - 12d34'56.
- Grads (400 grads in a circle) - 12.34g
- Radians - 12.34r
- Surveyors units (Quadrant Bearings)- N12d34'56 E

**Note:** Please refer to "Using Architectural Units" for details on the setup and use of architectural units.

### Orientation

In most drawings, angles are measured in the counterclockwise direction. This is the default in AllenCad and is called Standard Angles.

You can, however, set the orientation so that positive angles are measured in the clockwise direction, called Compass or Azimuth angles.

### Angle Zero Direction

In most forms of drafting, zero degrees is located at the positive X-axis (East, or 3 o'clock). AllenCad allows you to change the location of zero degrees. For example, surveyors normally set zero degrees as up on the page (North, or 12 o'clock). The display of the status bar is updated depending upon the settings of the angle zero direction and the orientation.



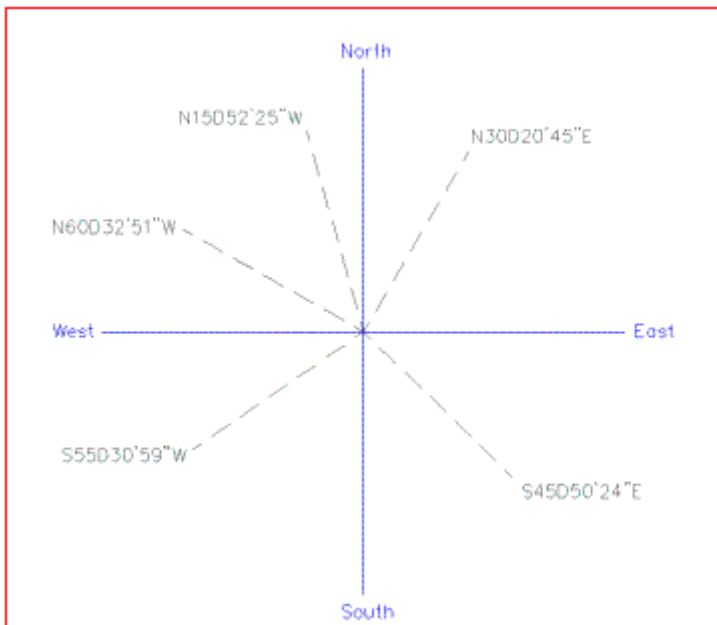
### Surveyor Units Entry

Because the surveyor units are so different from the other forms of angle entry, we have included a few examples:

Type of coordinate	Input	Example (Surveyor Units)
<b>Absolute Coordinates</b>		
Polar	distance<angle	7.5< N34D30'45"W
<b>Relative Coordinates</b>		
Polar	relative distance<angle	@7.5< N34D30'45"W

In surveyor's notation, the N stands for North. The 34D is 34 degrees from North. The 30' is an additional 30 minutes, and the 45" is an additional 45 seconds from North. The W indicates that the angle is towards the West.

There are 60 seconds in a minute, 60 minutes in a degree, and 360 degrees in a circle. Surveyors divide the circle into four parts, each with 90 degrees measured from the North or South poles in the direction of East or West.



In the figure above, we see the four points of a compass: North, South, East, and West. As examples of

how the Surveyor's Units work, several lines have been drawn in various directions. We labeled these lines with the correct bearings. Note that all the letters used in the bearings are in capital letters, as required by AllenCad.

Note: When entering the N, S, E, W and D, be sure to enter them as capital letters -- lower-case letters will not work.

## Configuration



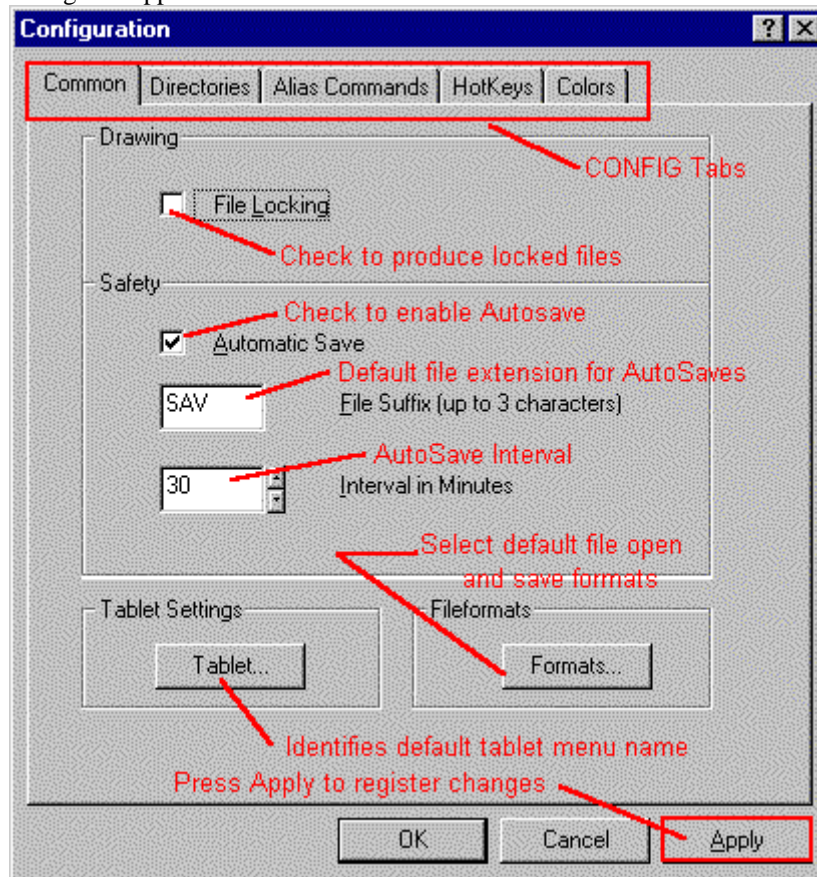
**Command: CONFIG**

**Alias CFG**

**Located:**

1. **Pull-down Menu: Tools – Configure**
2. **Or Enter Command or Alias Directly at the Command Line**

By calling the CONFIG command or selecting Configure on the File pull-down menu, the Configuration dialog will appear.



As can be seen, this dialog contains five tabs, each presenting separate dialogs under the headings: Common, Directories, Alias Commands, Hotkeys and Colors.

Note: Changes made in these dialogs will only become effective after the APPLY button has been selected. No changes will become effective if you close the dialog window by pressing OK, the Cancel button or ESC key.

## Settings

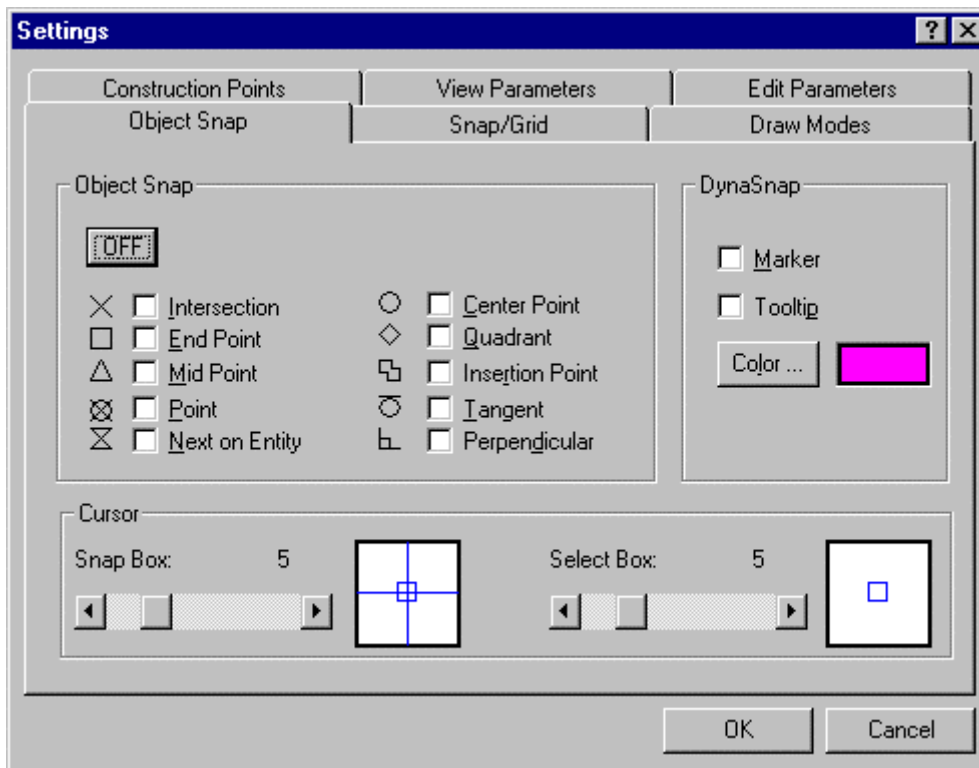


**Command: SETTINGS**  
**Alias ST**

**Located:**

1. **Pull-down Menu: Tools – Settings**
2. **Palette: Snap**
3. **Or Enter Command or Alias Directly at the Command Line**

The command SETTINGS is the main command to set the parameters and modes of the drawing environment. Some of the tabs and settings shown below can be accessed directly, by other commands, in addition. These include: Object Snap – Command PRECPAR; Draw Modes – Command DRAWMODE; Edit Parameters – Command EDITPAR and View Parameters – Command VIEWPAR. The command calls a dialog containing a number of tabs that each have specific windows for entries that control the drawing environment.



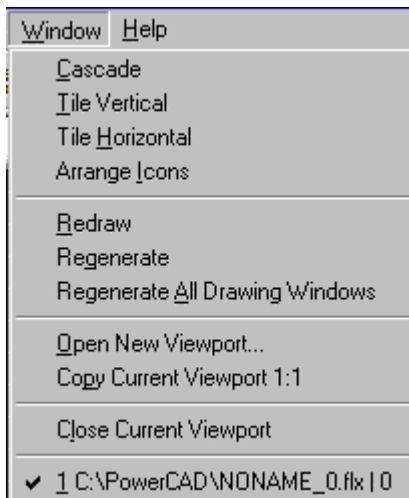
For a more detailed explanation of each of these tabs and their entries, click on the links below:

[OBJECT SNAP](#)  
[SNAP/GRID](#)  
[CONSTRUCTION POINTS](#)  
[DRAW MODES](#)  
[EDIT PARAMETERS](#)  
[VIEW PARAMETERS](#)



## Window

### "Window" Pull-down Menu



The Window pull-down menu controls the display of drawing Windows . For a description of each command, please refer to the specific topics that follow.

## Cascade Drawing Windows



**Command WCASCADE**

**Alias WD**

### **Located:**

- 1. Pull-down Menu: Window - Cascade**
- 2. Or Enter Command or Alias at the Command Line**

Arrange drawing windows in cascade format.

## Tile Vertically



**Command: 'WTILEVER**

**Alias WV**

### **Located:**

- 1. Pull-down Menu: Window – Tile Vertical**
- 2. Or Enter Command or Alias at the Command Line**

Arrange drawing windows in a vertically tiled format.

In the case that more than one viewport has been opened, you can arrange them in a vertical order by choosing the WTILEVER command.

## Tile Horizontally



**Command: 'WTILEHOR**  
**Alias WH**

**Located:**

1. **Pull-down Menu: Window – Tile Horizontal**
2. **Or Enter Command or Alias at the Command Line**

Arrange drawing windows in a horizontally tiled format.

In the case that more than one viewport has been opened, you can arrange them in a horizontal order by choosing the WTILEHOR command.

**Arrange Icons**

**Command: 'WICONARR**

**Alias WI**

**Located:**

1. **Pull-down Menu: Window – Arrange Icons**
2. **Or Enter Command or Alias at the Command Line**

Arrange all drawing window icons.

## Redraw



**Command: 'REDRAW**  
**Alias R**

**Located:**

1. **Pull-down Menu: Window – Redraw**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Or Enter Command or Alias at the Command Line**

Refresh current drawing window.

**Regenerate**

**Command: REGEN**  
**Alias RG**

**Located:**

1. **Pull-down Menu: Window – Regenerate**
2. **Or Enter Command or Alias at the Command Line**

Regenerate current drawing.

## Regenerate All

**Command: REGENALL**  
**Alias RGA**

**Located:**

1. **Pull-down Menu: Window – Regenerate All**
2. **Or Enter Command or Alias at the Command Line**

Regenerate all drawings.

## Open New Drawing Window

**Command: WOPEN**

**Alias WO**

**Located:**

1. **Pull-down Menu: Window – Open New Viewport**
2. **Or Enter Command or Alias at the Command Line**

In order to open a viewing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing.

From the menu *Window* choose the option **Open Another Viewing Window....**

Before the program opens another window you will be asked to determine the type of view in this new window. Choose an individual viewpoint or a standard view from the dialog box, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

## Copy a Drawing Viewport

**Command: QWOPEN**

**Alias QW**

**Located:**

1. **Pull-down Menu: Window – Copy Current Viewport 1:1**
2. **Or Enter Command or Alias at the Command Line**

The option **Copy Viewport** on the *Window* menu allows you

1. to make a quick copy of the current viewport or
2. a plan view, by choosing the desired option button.

## Close Drawing Window

**Command: WCLOSE**

**Alias WC**

**Located:**

1. **Pull-down Menu: Window – Close Current Viewport**

## 2. Or Enter Command or Alias at the Command Line

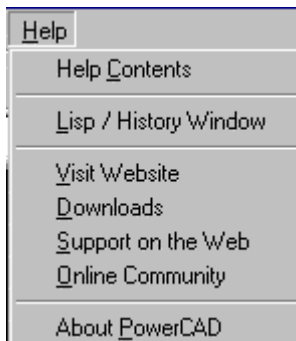
It is possible to close the current viewport, as in all Windows programs, by double-clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

### Drawing History

At the bottom of the File Menu is the Drawing History box. Here, the recently worked upon drawing files are displayed. Clicking on the filename will open that file in the drawing editor.

### Help

#### "?"(Help) Pull-down Menu



The "?" (Help) pull-down menu contains command instructions that activate the programs Help program with Tutorials. There is also a log of all entered commands and program response messages that can be viewed for reference, and details about the AllenCad program, itself. For a description of each command, please refer to the specific topics below.

### LISP / History Window

Often, it can be very helpful for the operator to be able to view a listing of commands that have been entered and the program's response messages to those commands. For this reason, AllenCad offers the LISP / History window, where all commands and responses can be viewed, whether entered from the pull-down menus, palette icons, typed at the command line, or entered by LISP programs or Macros (Script Files).

### Visit Website

HYPERLINK "<http://www.ADSI.com>"

To learn more about AllenCad and ADSI Inc. please visit our website. This link is also provided in the Help pull-down menu to provide direct Internet access from the program directly to the AllenCad Home Page.

To use this feature, you will naturally have to have your computer connected to the Internet, beforehand.

## Downloads

As with other Internet access features provided on the Help pull-down menu, the Downloads selection will work only if your computer has a connection to the Internet. This selection will take you to the AllenCad download center where Service Packs, Lisp Routines and supplemental instructions are available for download as they become available.

## Support On The Web

Located on the Help pull-down menu, this selection will connect you directly to AllenCad's Technical Support section. Here you will find links to online Help resources as well as a Help Request / Bug Report submission form. Direct technical support is available to all registered program users and assistance is usually available within twenty-four hours.

As many problems can be very quickly resolved though the solutions found in the program Help and User Document files, as well as online Help resources, all users are encouraged to review these resources before filing a report.

We at ADSI are most interested in receiving feedback on suspected bugs or suggestions for improvements and encourage all to contact us through the Feedback Input Form available with information of this nature.

## Online Community

All users are invited to join in on the Users Discussion Group, and a link to this Internet site has been provided on the Help pull-down menu to facilitate access. As with all other Internet resources, this link is functional only if you have Internet access available on your computer.

The Users Discussion Group is your opportunity to connect with other users to discuss problems and solutions you have found, or simply to share technical notes. This group is moderated by our technical support staff, who will be pleased to lend assistance or advice, as needed, in this User Forum.

## About

**Command: About**  
**Alias AB**

### **Located:**

- 1. Pull-down Menu: Help - About**
- 2. Or Enter Command or Alias at the Command Line**

The ABOUT box reports

1. Copyright
2. Program Version
3. User Name
4. Serial Number
5. Physical Memory
6. Disk Space

## Palettes











### Using Palettes

Palettes are callable toolbars that contain groupings of command icons that generally provide related functions. These palettes can be called onto the drawing window from the pull-down menus, called by the PALMAN command, or simply selected from the group of palette icons (shown below) that are located on the top toolbar:



As with all desktop icons, holding a mouse pointer over them momentarily, can identify these icons, after which a tooltip will appear with the identifier for that icon.

Individually, the icons and corresponding palettes are as follows:

-  - Draw Palette
-  - Edit Palette
-  - Modify Palette
-  - Detail Palette
-  - Snap Palette
-  - Dimension Palette
-  - Parts Palette
-  - View Palette
-  - Raster Palette
-  - Zoom Palette

Each palette contains a group of related command icons, which can be called onto the edit window for ready access to the commands when you are performing related drawing duties. The Draw palette (pictured below) for instance, contains all program commands that relate to drawing geometrical objects.



This palette can be called whenever you require repeated access to these various commands and it can be dragged and dropped in any convenient location, as you move about and work on your drawing. When finished with it, you simply click in its close button on the top left corner of the palette header, and it closes.

## Palette Manager

Command PALMAN

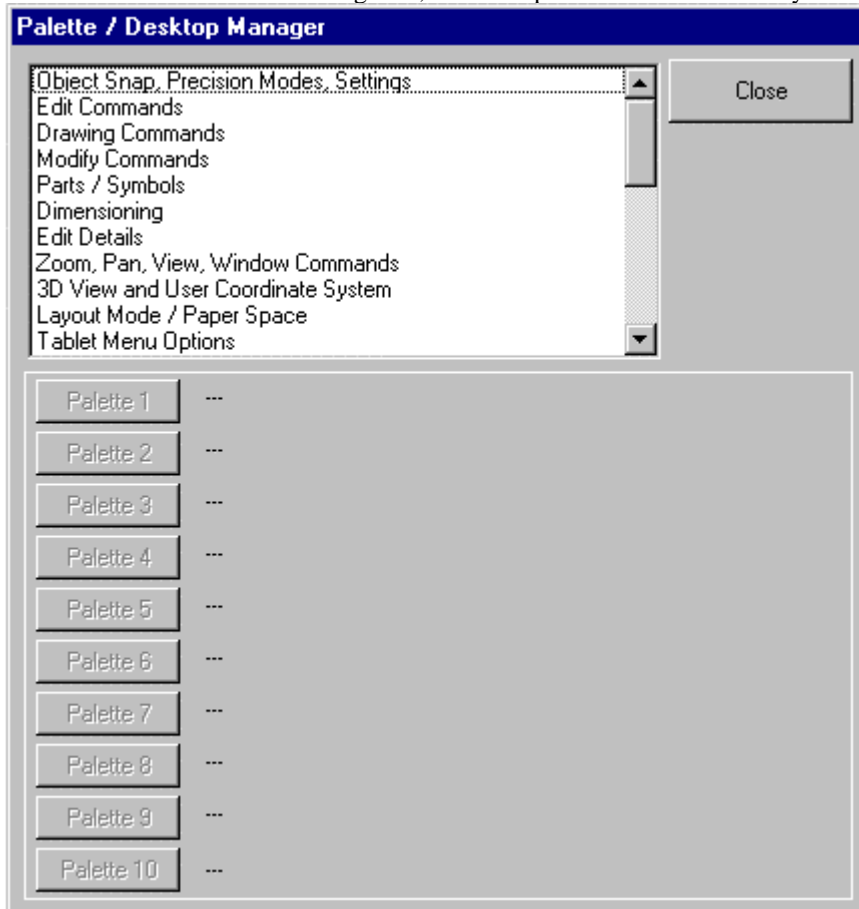
Alias PM

Located:

1. Pull-down Menu: Tools – Palette Manager
2. Or Entered Directly at the Command Line

Most users will elect to use the palette icons on the Toolbar to activate the various palettes, however others may prefer to select *Palette Manager* from the menu *File* and choose the action that they want in a more interactive fashion.

In the selection area of the Dialog Box, all of the palettes that are currently available are shown:



Select the desired palette. Underneath the selection area are three switches with the designations "Palette 1", "Palette 2", and so forth, to "Palette 10". By selecting one of the ten switches, the user can assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID-number already opened, this palette will be closed and replaced by the new palette having that ID number. When the palette is opened, the selecting window will close. If no palette is to be opened or replaced the user may close the window by selecting *Cancel*.

If the user wants to include customized palettes in the Palette Manager she/ he should proceed as follows: Edit the file *<Menu name>.pal* that is situated in the \SUPPORT directory. For instance, if the user menu is called *Pcad.mnu*, the user must edit the file *Pcad.pal*. The structure of the file is:

Palette description 1

palette1.mnp

Palette description 2

palette2.mnp  
and so forth.

The palette description will be found in the list shown in the Palette Manager Dialog Box, and the line following will always name the palette file. The user can complete or delete the entries as he or she wishes.

Load Palettes...



Command: 'PALETTE

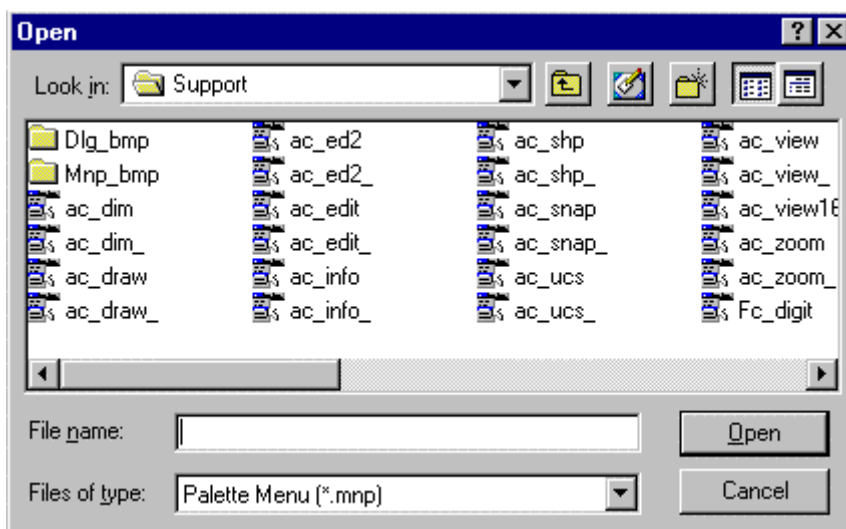
Alias PLT

Located:

1. Pull-down Menu: Tools – Resource Manager – Palette...
2. Or Enter Command or Alias at the Command Line

*Load Palette...* serves to open and place a Palette onto the program desktop. From the Open File dialog box that is opened the user can select the desired palette file to be loaded.

When you issue the command, a dialog box is presented, allowing you to select the palette file (\*.mnp) that corresponds to the palette you want to load. Simply select the file, and the palette will be placed on the desktop.



Every palette opens in its own window. In the title bar of the palette (toolbox) the user will find the name of the palette as well as an ID-number.

Palettes may also be opened through individual selections located on the pull-down menus, or using the palette icons located on the toolbar, as pictured in the image below:



Further information on creating and editing palettes can be found in Chapter 7 of the **Customization and Programmer's Guide**



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**See also:**

Exchange Palettes

Close Palettes

## Exchange Palettes

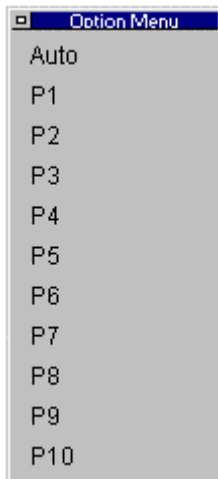
**Command:** 'PALOPEN

**Alias** PLO

**Located:**

1. **Pull-down Menus:** (Different pull-downs for different palettes)
2. **Toolbar icons for individual palettes**
3. **Or Enter Command or Alias at the Command Line**

The PALOPEN command will open a new Palette. If necessary, it replaces another palette.



Command Sequence:

> PALOPEN

Caption or ID of palette to replace: P1 (select from above dialog)

Upper left corner: Right click on the corner

File name: ac\_ucs (enter the filename of the palette to load).

The new palette is then placed on the desktop. Any existing palette loaded under the P1 heading, would have been replaced in this operation.

---

**See also:**

Load Palettes

Close Palettes

## Close Palettes

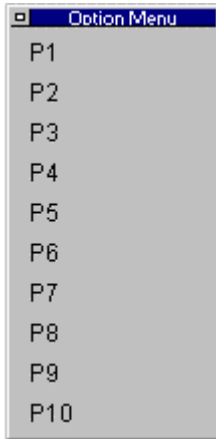
**Command:** 'PALCLOSE

**Alias** PLC

### Located:

#### 1. Entered Directly at the Command Line

To remove a palette from the desktop, use the PALCLOSE command or click on the close button in the palette header. You will be presented with the dialog:



Each open palette has its palette number designation in the header next to the palette name. Simply click on the corresponding number in the dialog to close the palette.

**Note:** Palettes can also be closed by clicking on the close button, located on the left side of the palette header.

---

#### See also:

Exchange Palettes

Close Palettes

Symbol Fields



A series of frequently used commands can be called using symbol fields.

These are groups of permanently displayed symbols in the Function Bar or the vertical symbol bar. These groups are comprised of associated command icons, such as the snap selection icons pictured above.

Commands and functions are called by clicking on the individual command icons.

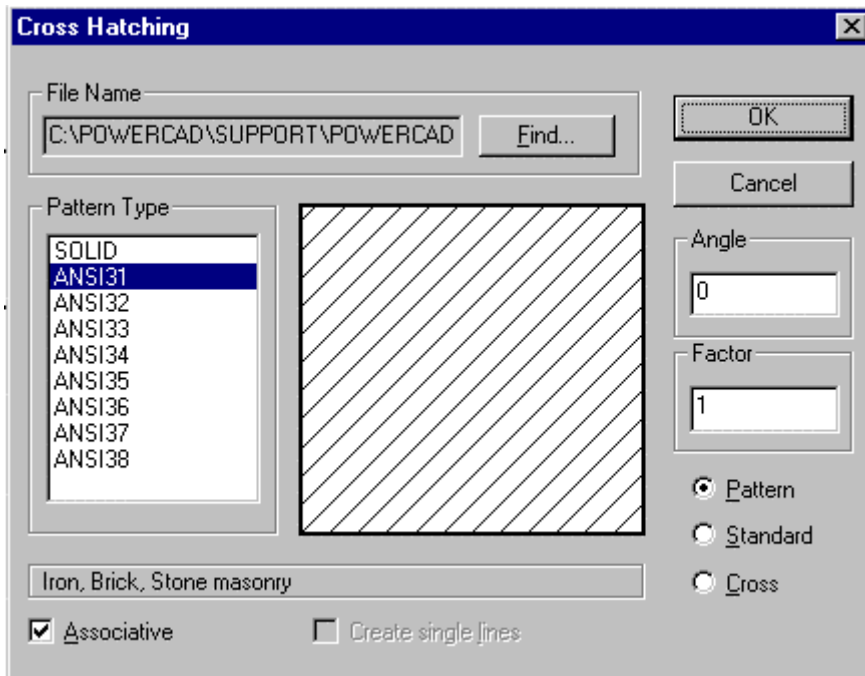
#### Icon Menus

#### Command ICONMENU

### Located:

#### 1. Pull-down Menu: Tools – Resource Manager - Icon Menu...

The Icon menu command is a programming tool that presents the user with selections that are supported by images. Icon menus use groups of images as graphic labels. Icon menus also have a list box containing text labels associated to each image. This combination allows the user to select options



Icon menus are a valuable visual tool, as in the example above, when selections of hatch patterns or text styles should be customized. Also, icon menus are can be useful in creating symbol library selection menus.

With this command, a dialog presenting an assortment of graphical options can be created. A slider bar is added automatically to the dialog if the number of images exceeds display limits.

You can customize the number of columns and rows to be displayed for each icon menu.

### Command Button Chart (Reference)

The following chart should be helpful when creating or modifying palettes or toolbars:

PREFIX	Command	Icon	Key Word	Status Bar
^^	(setvar "TILEMODE" 0)(command "_VIEWPORT" "_V")	fc_vpvw .bmp	Layout Mode Viewport Control	Layout Mode Viewport Control
^^	(setvar "TILEMODE" 0)(command "_VIEWPORT")	fc_vport. bmp	Create or Modify Layout Viewport	Create or Modify Layout Viewport
^^	(setvar "TILEMODE" 1)(command "_ARENDER")	fc_arend .bmp	AutoRender	AutoRender
^^	2DFACE	fc_2dfac .bmp	2DFACE	Draw filled plan faces
^^	3DFACE	fc_3dfac .bmp	3DFACE	Draw 3D faces
^^	3DVIEW	fc_3vie w.bmp	3DVIEW	Set 3D Viewpoint for Current Viewport
^^	ARC	fc_arc.b mp	ARC	Draw an arc

^^	AREA	FC_AR EA.BM P	AREA	Calculate area / circumference
^^	ARRAY C	fc_arrcr. bmp	ARRAY C	Circular Array: Copies rotated
^^	ARRAY I	fc_arr_i. bmp	ARRAY I	Individual Array
^^	ARRAY N	fc_arrcn. bmp	ARRAY N	Circular Array: Copies not rotated
^^	ARRAY R	fc_arr_r. bmp	ARRAY R	Rectangular Array
^^	ATTDEF	FC_ATT DE.BM P	ATTDEF	Define Attribute of part
^^	ATTEDIT	FC_ATT ED.BM P	ATTEDIT	Modify attribute location / value
^^	ATTEXP	FC_ATT XP.BMP	ATTEXP	Write attribute information to file
^^	ATTXEDIT	FC_ATT XE.BM P	ATTXEDIT	Edit Attribute infos of a part
^^	BHATCH	PC_BH AT.BM P	BHATCH	Apply Boundary Cross-Hatching
^^	BMPOUT	FC_BM POU.B MP	BMPOUT	Create bitmap file from drawing
^^	BPOLY	PC_BPO LY.BM P	BPOLY	Apply Bondary Polyline
^^	CALLOUT	Pc_callo ut.bmp	CALLOUT	Insert text box
#_	Cen	fc_s_cen .bmp	Cen	Snap center of circle or arc
^^	CHAIN	fc_chain. bmp	CHAIN	Draw a 2D polyline chain
^^	CHAMFER	fc_cham f.bmp	CHAMFER	Chamfering two entities
^^	CHANGE	PC_CH ANG.B MP	CHANGE	Change Object Attributes Or Location
^^	CIRCLE	fc_circ.b mp	CIRCLE	Draw circle(s)
^^	CLOSE	fc_f_cls. bmp	CLOSE	Close current drawing
^^	CONFIG	fc_f_cfg. bmp	CONFIG	General settings for the program
^^	COPY	fc_copy. bmp	COPY	Copy selected objects
^^	COPYPROP	FC_PRO P.BMP	COPYPROP	Copy properties to other objects
^^	COPYSELECT	FC_CLC P.BMP	COPYSELECT	Copy selected objects to clipboard
^^	CUTSELECT	fc_cut.b mp	CUTSELECT	Cut selection set to clipboard

^^	DELETE	FC_DEL .BMP	DELETE	Erase drawing objects
^^	DELPARTIAL	fc_break .bmp	DELPARTIAL	Delete partial segments of entities
^^	DIMA3P	FC_D_A N3.BMP	DIMA3P	Angular dimensioning (defined by 3 points)
^^	DIMA4P	FC_D_A N4.BMP	DIMA4P	Angular dimensioning (defined by 4 points)
^^	DIMALI	FC_D_A LI.BMP	DIMALI	Aligned linear dimensioning
^^	DIMBAS	FC_D_B AS.BMP	DIMBAS	Baseline dimensioning
^^	DIMCENT	FC_D_C EN.BM P	DIMCENT	Center marking
^^	DIMCON	FC_D_C ON.BM P	DIMCON	Continuing dimensioning
^^	DIMDIA	FC_D_D IA.BMP	DIMDIA	Diameter dimensioning
^^	DIMDRAW	FC_DIM .BMP	DIMDRAW	Create dimensions
^^	DIMEDIT	FC_EDI M.BMP	DIMEDIT	Edit dimensions
^^	DIMHOR	FC_D_H OR.BM P	DIMHOR	Horizontal dimensioning
^^	DIMLIN	FC_D_L IN.BMP	DIMLIN	Linear dimensioning
^^	DIMOBL	FC_D_O BL.BMP	DIMOBL	Modify Dimension Orientation
^^	DIMORD	FC_D_O RD.BM P	DIMORD	Ordinate dimensioning
^^	DIMRAD	FC_D_R AD.BM P	DIMRAD	Radius dimensioning
^^	DIMREST	PC_DRS T.BMP	DIMREST	Restore Dim Style
^^	DIMROT	FC_D_R OT.BM P	DIMROT	Rotated linear dimensioning
^^	DIMSAVE	PC_DS AVE.B MP	DIMSAVE	Save Dim Style
^^	DIMSER	FC_D_S ER.BMP	DIMSER	Serial dimensioning
^^	DIMTHOME	FC_D_T HO.BM P	DIMTHOME	Realign Dim Text
^^	DIMTMOVE	FC_D_T MO.BM P	DIMTMOVE	Move Dim Text
^^	DIMTNEW	PC_DT NW.BM P	DIMTNEW	Modify Dim Text Value

^^	DIMTROT	FC_D_T RO.BM P	DIMTROT	Rotate Dim Text
^^	DIMTYPE	FC_PRE F.BMP	DIMTYPE	Set or modify dimension style
^^	DIMUPD	FC_D_U PD.BMP	DIMUPD	Update Dimensions
^^	DIMVER	FC_D_V ER.BMP	DIMVER	Vertical dimensioning
'	DIST	FC_DIS T.BMP	DIST	Measure distance / angle
^^	DPAN	PC_DP AN.BM P	DPAN	Dynamic Pan
^^	DTEXT	PC_DTE XT.BM P	DTEXT	Direct Text Entry
^^	DZOOM	FC_DZ OOM.B MP	DZOOM	Dynamic Zoom
'	EDITPAR	FC_PRE F.BMP	EDITPAR	Parameters for modify commands
^^	EINFO	FC_OIN FO.BMP	EINFO	Info on selected objects
^^	ELLIPSE	FC_ELL IP.BMP	ELLIPSE	Draw an ellipse
#_	end	fc_s_end .bmp	end	Snap end point
^^	EXPAND	fc_expan .bmp	EXPAND	Extend entities to boundary edge
^^	FILLET	fc_fille.b mp	FILLET	Filleting two entities
^^	FLIP	fc_mirnc .bmp	FLIP	Mirror objects (original deleted)
^^	FONT	FC_FO NT.BM P	FONT	Manage Text Styles
^^	FREEHAND	Pc_freeh and.bmp	FREEHAND	Freehand line and text sketching
^^	HATCH	fc_hatch. bmp	HATCH	Create cross-hatching
^^	HPEDIT	FC_HE DIT.BM P	HPEDIT	Revise Existing Hatches
'	ID	FC_ID. BMP	ID	Identify point coordinate
#_	ins	fc_s_ins. bmp	ins	Snap to base point of part
^^	INSERT	FC_INS ER.BMP	INSERT	Part insertion dialog
#_	int	fc_s_int. bmp	int	Snap intersection
^^	INTERSECT	fc_inter. bmp	INTERSECT	Trim/extend two entities to intersection point
^^	LENGTHEN	fc_leng. bmp	LENGTHEN	Lengthen/Shorten line or arc

^^	LIBRARY	FC_PLI B.BMP	LIBRARY	Insert part from library
^^	LIGHT	FC_LIG HT.BM P	LIGHT	Set light for rendering
^^	LINE	fc_line.b mp	LINE	Draw line(s)
^^	MENU	fc_f_mn u.bmp	MENU	Load another menu
_	mid	fc_s_mi d.bmp	mid	Snap mid point
^^	MIRROR	fc_mirr. bmp	MIRROR	Mirror objects (original remains)
^^	MOVE	fc_move .bmp	MOVE	Move selected objects
^^	MTEXT	PC_MT EXT.B MP	MTEXT	Multi-Line Text Entry
^^	MTEXTEDIT	PC_MT XED.B MP	MTEXTEDIT	Edit Multi-Line Text
^^	NEW	fc_f_ne w.bmp	NEW	Create a new drawing
^^	NGON	fc_ngon. bmp	NGON	Draw a regular polygon
#_	nxt	fc_s_nxt. bmp	nxt	Snap closest point
^^	OFFSET	fc_offse. bmp	OFFSET	Offset-Copy: Parallel or Complement
^^	OPEN	fc_f_opn .bmp	OPEN	Open an existing drawing
^^	PALOPEN Parts Auto fc_part_.mnp	FC_PS WIT.BM P	PALOPEN Parts Auto fc_part_.mnp	Toggle to horizontal (or vertical) palette
	PALSWITCH pc_det.mnp	pc_detail _pal.bm p	PALSWITCH pc_det.mnp	Open Detail Palette
	PALSWITCH pc_dim.mnp	pc_dim_ pal.bmp	PALSWITCH pc_dim.mnp	Open Dimension Palette
	PALSWITCH pc_draw.mnp	pc_draw _pal.bm p	PALSWITCH pc_draw.mnp	Open Draw Palette
	PALSWITCH pc_edit.mnp	pc_edit_ pal.bmp	PALSWITCH pc_edit.mnp	Open Edit Palette
	PALSWITCH pc_modi.mnp	pc_mod _pal.bm p	PALSWITCH pc_modi.mnp	Open Modify Palette
	PALSWITCH pc_part.mnp	pc_parts _pal.bm p	PALSWITCH pc_part.mnp	Open Parts Palette
	PALSWITCH pc_rast.mnp	pc_raster _pal.bm p	PALSWITCH pc_rast.mnp	Open Raster Palette
	PALSWITCH pc_snap.mnp	pc_snap _pal.bm p	PALSWITCH pc_snap.mnp	Open Snap Palette

	PALSWITCH pc_view.mnp	pc_view _pal.bm p	PALSWITCH pc_view.mnp	Open View Palette
	PALSWITCH pc_zoom.mnp	pc_zoom _pal.bm p	PALSWITCH pc_zoom.mnp	Open Zoom Palette
#'	PAN	fc_pan.b mp	PAN	Repositioning of drawing window
#'	PANDOWN	fc_down .bmp	PANDOWN	Pan Down
#'	PANLEFT	fc_right. bmp	PANLEFT	Pan Left
#'	PANRIGHT	fc_left.b mp	PANRIGHT	Pan Right
#'	PANUP	fc_up.b mp	PANUP	Pan Up
^^	PARTDEF	FC_PAR TD.BM P	PARTDEF	Define Part by selection
^^	PARTEXP	FC_PAR TX.BM P	PARTEXP	Write objects to external part file
^^	PASTE	FC_PAS TE.BMP	PASTE	Paste selection set from clipboard
#_	per	fc_s_per. bmp	per	Snap perpendicular to ...
^^	PLANVIEW	fc_planv .bmp	PLANVIEW	Plan view on current UCS
^^	POINT	fc_point. bmp	POINT	Set construction points
^^	POINT _M	FC_PM EAS.B MP	POINT _M	Measures entity with points
^^	POINT _S	FC_PSE GS.BMP	POINT _S	Segments entity with points
^^	POLYEDIT	FC_PED IT.BMP	POLYEDIT	Edit Polylines
^^	POLYLINE	fc_pline. bmp	POLYLINE	Draw a polyline
^^	PRINT	fc_f_prt. bmp	PRINT	Print the current drawing
^^	PROEDIT	FC_CHP RO.BM P	PROEDIT	Change Layer / Linetype
^^	PROPLAYER	FC_CH LAY.B MP	PROPLAYER	Modify the layer attribute of entities
#_	qua	fc_s_qua .bmp	qua	Snap quadrant of circle or arc
^^	RASTER BMP2RLC	fr_b2r.b mp	RASTER BMP2RLC	Convert monochrome Bitmap file to RLC file
^^	RASTER RBORDER	fr_bord. bmp	RASTER RBORDER	Toggle Raster Frame On/Off
^^	RASTER RCUT	fr_cut.b mp	RASTER RCUT	Cut Region from Raster
^^	RASTER	fr_r2b.b	RASTER	Convert RLC file to monochrome



^^	RLC2BMP RASTER RLOAD	mp fr_open. bmp	RLC2BMP RASTER RLOAD	Bitmap Load Raster File
^^	RASTER RMIRROR	fr_mirr.b mp	RASTER RMIRROR	Mirror Raster
^^	RASTER RMONT	fr_mont. bmp	RASTER RMONT	Assemble / Fit Raster Files
^^	RASTER RMOVE	fr_move. bmp	RASTER RMOVE	Move Raster
^^	RASTER RPROP	fr_prop. bmp	RASTER RPROP	Property Settings of Raster File
^^	RASTER RROTATE	fr_rot.b mp	RASTER RROTATE	Rotate Raster
^^	RASTER RSAVE	fr_save.b mp	RASTER RSAVE	Save Raster
^^	RASTER RUNLOAD	fr_clos.b mp	RASTER RUNLOAD	Remove Raster
^^	RCONFIG	FC_RE NDC.B MP	RCONFIG	Preferences for Fast Rendering
^^	RECTANGLE	fc_rect.b mp	RECTANGLE	Draw a rectangle
^^	REDO	fc_redo1 .bmp	REDO	Redo recent Undo
#'	REDRAW	fc_redr2. bmp	REDRAW	Refresh current drawing window
^^	REJOIN	fc_join.b mp	REJOIN	Rejoin broken lines or arcs
^^	RENDER	FC_RE ND.BM P	RENDER	Create rendered bitmap
^^	RING	fc_ring.b mp	RING	Draw filled rings / circles
^^	ROTATE	fc_rotat. bmp	ROTATE	Rotate selected objects
^^	RTEXT	PC_RTE XT.BM P	RTEXT	Text Entry By Reference
^^	RTOP	FR_TOP .BMP	RTOP	Change display order
^^	SAVE	fc_f_sav. bmp	SAVE	Save the current drawing
^^	SCALE	fc_scale. bmp	SCALE	Resize selected objects
^^	SELECT	FC_SEL EC.BMP	SELECT	Select Objects
#'	SETTINGS	fc_pref.b mp	SETTINGS	Preferences and Modes for Drawing and Editing
^^	STRETCH	fc_strec. bmp	STRETCH	Stretch-Moving of Objects
^^	TABLES	FC_TA BLE.BM P	TABLES	Info on drawing database tables
^^	TABSERVICE	fc_tserv. bmp	TABSERVICE	Drawing Tables Maintenance
#_	tan	fc_s_tan.	tan	Snap tangent to ...

^^	TCORRECT	bmp PC_TC ORREC T.BMP	TCORRECT	Multiple Text Revisions By String
^^	TEXT	fc_text.b	TEXT	Generate text in drawing
^^	TEXTEDIT	mp FC_TXT ED.BM	TEXTEDIT	Edit Single-Line Text
^^	TLMODE	P fc_layou	TLMODE	Toggle between Layout Mode and Model Mode
^^	TRAPEZOID	.bmp fc_2cone	TRAPEZOID	Draw a cone (2D)
^^	TRIM	.bmp fc_trim.b	TRIM	Trim entities to cutting edge
#'	TSCREEN	mp FC_F_C MD.BM	TSCREEN	Toggle to/from Text Screen
#	U	P fc_undo	U	Undo previous command
^^	UCS	l.bmp fc_ucs.b	UCS	User Coordinate System Functions
^^	UCS C	mp fc_ucsm.	UCS C	UCS Dialog
^^	UNDELETE	bmp FC_UN DEL.B	UNDELETE	Undo last delete operation
^^	VIEW	MP fc_view.	VIEW	Save or retrieve named view
#'	VIEWPAR	bmp FC_PRE F.BMP	VIEWPAR	Parameters for view commands
^^	VNOTE	Fc_comt	VNOTE	Insert audio or text comment
^^	VTXEDIT	.bmp FC_VT XED.B	VTXEDIT	Edit Polyline Vertices
#'	WCASCADE	MP FC_WC ASC.B	WCASCADE	Cascade drawing windows
^^	WMFOUT	MP FC_WM FOU.B	WMFOUT	Create Windows Metafile from drawing
^^	WTILEHOR	MP FC_WTI LH.BM	WTILEHOR	Tile Windows Horizontally
#'	WTILEVER	P FC_WTI LV.BM	WTILEVER	Tile Windows Vertically
^^	XPLODE	P FC_EXP LO.BM	XPLODE	Break complex objects down to single entities
^^	XPLORER	P fc_xplor.	XPLORER	Xplorer: Drawing Tables Navigator
#	X,Y,Z	bmp fc_xy.b	X,Y,Z	Coordinate Filter X,Y,Z
#'	ZOOMBACK	mp FC_ZB	ZOOMBACK	Zoom to previous view

		ACK.B MP		
#'	ZOOMFIT	FC_ZFI T.BMP	ZOOMFIT	Zoom to drawing extents
#'	ZOOMIN	FC_IN. BMP	ZOOMIN	Zoom In: Nearer to objects
#'	ZOOMOUT	FC_OU T.BMP	ZOOMOUT	Zoom Out: Away from objects
#'	ZOOMWIN	FC_WI N.BMP	ZOOMWIN	Zoom Window: Specified by box

## Create Entities Using Commands

### "Q" Commands

The AllenCad program makes extensive use of command dialogs, primarily to facilitate command entry for the new user. To facilitate command use with macros and LISP routines and to assist advanced users who may be more familiar with AutoCAD direct entry commands, AllenCad also has command-line variants to existing dialog box-based commands. To continue the tradition of the command naming convention, these commands are identified by the Q prefix in the command line version. For example, the Layer command displays a dialog box and the QLayer command displays prompts at the command line. More examples are shown in the table. We call these "Q commands."

### Subtle Differences Between Commonly Used AutoCAD and AllenCad Commands

AutoCAD Command	AllenCad Command	Comments
QSave	Qsave	In AutoCAD, if the drawing is named, the drawing is saved without requesting a filename. If the drawing is unnamed, the Save Drawing As dialog box comes up and allows to save the drawing under the file name the user specifies.
Array	Qarray	In AllenCad, the Array command is processed differently from AutoCAD's Array command. However, the QArray command corresponds to the AutoCAD command in operation and options.
Mirror	Qmirror	In AllenCad, the Mirror command is processed differently from AutoCAD's Mirror command. However, the QMirror command corresponds to the AutoCAD command in operation and options.
Offset	Offset Qoffset	In AllenCad, the Offset command is processed different from AutoCAD's Offset command. The QOffset command corresponds to the AutoCAD command in operation and options.
Text	Qtext	In AllenCad, the Text command uses a dialog box. AllenCad's QText command corresponds to AutoCAD's Text command in operation and options.
AttDef	QattDef	The native AllenCad AttDef command uses a dialog box. QAttDef corresponds to the AutoCAD AttDef command in operation and options.
AttEdit	QattEdit	The native AllenCad command AttEdit uses a dialog box. QAttEdit corresponds to the AutoCAD AttEdit command in operation and

		options.
-Layer	Qlayer	The AllenCad QLayer command corresponds to AutoCAD R14's -Layer command in operation and options.
View	Qview	The AllenCad View command uses a dialog box. QView corresponds to AutoCAD's View command in operation and options.
-Style	Qfont	The AllenCad QFont command corresponds to AutoCAD R14's command - Style in operation and options.

A consistent command naming convention between AllenCad and AutoCAD cannot be achieved for several commands, especially with introduction of the hyphen-commands in Release 14. The following table shows some of the differences:

#### Examples of Command Naming Differences Between AutoCAD and AllenCad

AutoCAD Command Line	AllenCad Command Line	AutoCAD Dialog Box	AllenCad Dialog Box
AttDef	QattDef	DDAttDef	AttDef
Insert	Qinsert	DDInsert	Insert
-Style	Qfont	Style	Font
-Layer	Qlayer	Layer	Layer
-Mtext	Qtext	Mtext	Text

### Line



**Command: LINE**

**Alias L**

#### Located:

1. **Pull-down Menu: Draw - Line**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The **Line** command is used to draw lines or line segments or to connect lines to existing lines or arcs. Inputs to the command are two points; the starting point and the ending point of the line.

#### Options:

##### Continuous

**Continuous** is the standard option when drawing lines, and will automatically be used when no other option is selected. It allows the user to draw line segments in sequence until the command is terminated.

The command will request a start-point for the line, and will continue to request points until it is terminated. Each subsequent point will be connected to the previous point by a line.

From Point:  
To Point:  
To Point:  
...

### Undo

The option **Undo** can be used repeatedly to erase the last drawn line segment of a single **Continuous** line command in succession.

### Close

The option **Close** connects the last drawn point P<sub>n</sub> with the start point P<sub>1</sub> in the **Continuous** line command.

To terminate the command, press RETURN or ESC.

### Line Segments

The option **Segments** allows the creation of single line segments, as opposed to continuous lines.

**Line Segments** are useful when the line that is being drawn for future drawing construction purposes (IE a temporary line that defines a parameter to be used in drawing subsequent objects).

As with continuous lines, you determine the start- and end-point of the first segment. The program will continue to ask for start- and end-points.

In this mode the end of one line segment is not automatically used as the start point of the next segment.

To end the drawing of line segments, press RETURN.

### Append

The option **Append** allows the connection of a line to either the start or end-point of another line or arc.

Choose the line or arc that you wish to connect to. Note that the end of that object closest to the pick point will be used as the start point for the line segment.

#### Line to line

When appending a line to a line, select the line to which the appended line will connect. The point (start or end) of that line closest to the selected point on the line will be the start point for the new line.

#### Line to arc

To append a line to an arc, select the arc to which the line is to be appended. The prompt will ask for the length of the line. Either keyboard the length, or "pick" the length with the cursor. For precision, it is usually better to keyboard the length.

The appended line will be connected to the end point of the arc as a tangent.

To end the command press RETURN or ESC.

## Arc



**Command: ARC**

**Alias A**

#### Located:

1. **Pull-down Menu: Draw - Arc**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

There are several ways, geometrically, to describe an arc, just as with a circle. The program allows the user to choose from among these methods, and draw the arc in the most convenient manner.

The following options for drawing an arc are available:

1. Three Points (starting point, second point, end point);
2. Two points and radius;
3. Two points and angle of opening;
4. Two points and center point;
5. Two points and tangents through starting point;
6. Center point, starting point and angle; or,
7. Center point and two angles.

Variations of the options of this command may combine some of the basic geometry inherent to each method.

**All arcs will be measured to the left, starting from the positive x-axis.** In other words, you pick points on the arc in an anti-clockwise direction. It is important to recognize this directional convention, as failing to do so will produce an arc that is complementary to the one intended (the one that if added to the intended arc, would form a circle). To "pick" points for the construction of arcs, the user should always use OSNAP options.

### Options:

The options are displayed in the *Options Panel* when the command is active.

#### 3Point

Since the method most used for drawing arcs is to select three points on the arc, that is the option that has been chosen as the "standard", or default, option. It will be used if no other option is chosen.

The arc to be drawn is defined by three points. At the prompt, enter the starting point, a second point on the arc's circumference and the end point of the arc. Remember the directional convention; the arc is described anti-clockwise to the left starting with the positive X-axis.

Starting point: P1  
Second point: P2  
End point: P3

This option is also available in other options of the drawing command ARC.

#### SEnd (Starting point - End point - Radius/Angle)

Using the option **SEnd**, the user may create arcs with only the starting point and end point defined. The "arc" of the element is defined by entering or "picking" a third point on the circumference, or by specifying an angle.

The basic variation of this option simply follows the prompts, entering the starting point and end point and the radius of the arc by "picking" or by entering through the keyboard.

Starting point: P1  
End point: P2  
Radius: Value

When the user is "picking" the radius of the arc, it must be remembered that the distance of the center point to the **x-axis** of the end point of the arc is important, not its distance to the y-axis.

It is also possible to enter a radius, an angle or a direction from the starting point instead of the starting point and the end point. To do that, choose the option **Angle** or **Starting point**.

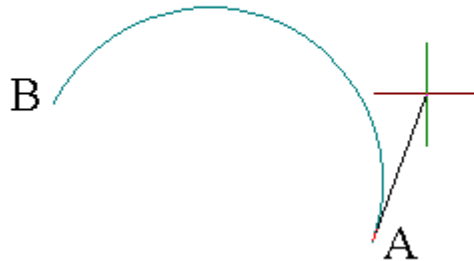
#### Option Angle

With this option the arc is defined by the included angle of the arc (angle between the starting point, the center point of the arc and its end point), the starting point, and the end point. The prompts will request a value with the words **Included angle**:

Enter that value either through the keyboard, or by "picking" a value. When picking a value, remember that the x-axis is the point of reference, not the last point entered.

### Direction

The arc is defined by a tangent that touches the arc at its starting point, next to its starting point and at its end point.



In this example, we have defined the start and end points of the arc (A and B) and have selected the direction option. This results in a line being represented from the arc's start point to the current cursor position. The arc radius varies with the cursor location such that the arc remains tangential to that line representation at the start point.

The program will ask the user to pick or enter the coordinates of a point that fixes the cursor location and thereby defines the arc radius.

### SCenter (Starting point - End point - Center point)

This is a variation of the option **SEnd**, in which the arc is drawn by determining its starting point, end point and center point.

Choose the option **Center point** from the options panel after having entered the starting point. Enter the coordinates of the center point manually or "pick" them with the cursor. The arc is defined when the end point is entered.

It is also possible to end the arc by entering an angle manually or "picking" it instead of the end point. The program expects a closed angle and defines it as the angle between the starting point, the center point and the end point of the arc.

### CStart

The option **CStart** draws the arc starting from the center point.

After choosing the option **CStart** from the options panel, define the center point. Then enter the starting point and the end point of the arc (it is possible to enter a closed angle instead of the end point).

Center point: P1  
Starting point: P2  
End point: Angle

### CRadius

**CRadius** also starts an arc by defining the center point. After that, determine the radius as well as the starting angle and the ending angle of the arc to be drawn. A rubberband line that starts in the center of the arc helps the user to determine the points to enter for the angle.

Center point: P1  
Radius: r  
Starting angle: P2

Ending angle: *P3*

## Append

It is possible to append, or connect, arcs to lines or other arcs with the option **Append**. The arc is determined by two points, the starting point being the connection point, and the end point of the new arc.

First choose a line or an arc to which the new arc should be appended. Then determine the end point of the arc to be drawn.

Choose the line or arc: *P1*

End point: *P2*

**Note:** The "smoothness" with which the arc is represented onscreen is controlled by the variable **CIRCLERES**. By issuing the **SETTINGS** command, users can go to the Draw Modes tab and enter a new value (between 8 and 1024) in the Arc Segments box that controls smoothness. A low value will result on a rough arc or circle, while a higher value will produce a smooth arc or circle.

## Chain



**Command: CHAIN**

**Alias CN**

### Located:

1. **Pull-down Menu: Draw - Chain**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The Chain command is a combination of the commands Line and Polyline.

The Chain command makes it possible to draw continuous lines in a manner similar to the command **LINE** using the option *Continuous*. The procedure is the same as drawing continuous lines.

But there are some important differences in drawing lines, polylines, and chains:

- Firstly, when you draw a continuous line, each different segment is a separate element. That way, each element can be chosen separately, using the object selection function.
- With a chain, in contrast, the resulting entity is a single entity, either for editing or for other purposes.
- Second, for a chain, it is possible to choose the line width. This is similar to the command Polyline. The last width value entered for a polyline will be offered as the default width for the next chain, and
- Finally, (differently from a Polyline) it is only possible to choose a single width of line for the entire line when drawing a Chain.

A chain is drawn by describing the points that will be the starting and end point of each chain segment. The end point for each segment will serve as the starting point for the next segment. A single width of the line must also be specified.

> CHAIN

First point: **P1**

Next point: **P2**

Next point: **P3**

...



You can terminate the Chain command by pressing RETURN or ESC.

### Options:

The following options will be available in the options panel:

#### Width

The option *Width* allows the user to define the width of the entire line. **The width must be specified before entering the first point.** Picking or entering the first point is understood as a verification of the required value.

The previous entered value from either a Chain or Polyline command will be offered as the default width value for the new chain. Verify this predetermined value by pressing RETURN or enter a new value for the width.

#### Undo

The option *Undo* lets the user reverse the last step taken. In other words, the last-drawn chain segment will be erased. The end-point of the previous segment is then used as a starting point of the next segment. Undo's can be repeated as long as segments exist.

#### Close

The option *Close* allows the user to close the chain to the start point. The two end points of the chain will be connected to each other and the command will be terminated.

#### Circle



**Command: CIRCLE**

#### Alias CI

##### Located:

1. **Pull-down Menu: Draw - Circle**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

There are several ways to describe a circle, geometrically. The program allows the user to choose from among these ways, and draw the circle in the most convenient manner.

A circle can be drawn by choosing one of the following options:

1. Center of the circle and radius or diameter;
2. Two Points on the circumference (that define the diameter and center point)
3. Three Points on the circumference
4. Two Tangents

These options will appear on the *Options Panel* when the command circle is active.

### Options

#### Center

Since the *Center Point* option is the most often chosen, it has been selected as the default option and is activated automatically if no other option is chosen. The first prompt for this option will be the location of the center point; the second will ask for the radius. If the circle being drawn is preceded by another circle, the last radius will appear at the prompt as the default radius value. To verify the default, just press RETURN, or to change the option, enter or "pick" a new radius.

If it is more convenient to enter the diameter than the radius, the user may choose that option from the

options panel.

## 2P

If the circle is to be tangent to two other lines, this option is handy.

The prompt will ask for two points in succession. Again, the points may be picked using the object snap options, or may be entered from the keyboard. The mid-point between the picked points will be the circle center and the distance between the points, the circle diameter.

## 3P

This option allows the user to draw a circle by choosing three points on the circumference of the circle. It is convenient to use when the circle must intersect three tangent lines, or when drawing the circle within a polygon.

The prompt will ask for three points successively. The user may point to these points on the screen, perhaps using the option **object snap**, or enter the coordinates through the keyboard.

## Radius

This geometry is particularly useful when the radius of the circle is a known, and the center point is undefined, or when the radius follows from geometry already established.

You are first asked to pick two points that define the circle radius length. These options are for entering the radius:

- Verify the default radius value (the radius of the last circle drawn) by pressing RETURN;
- Enter a new value for the radius manually; or
- Pick the radius on the screen. This option is especially useful when the radius follows from the geometry and only has to be copied.

Next, you are asked to locate the circle center point.

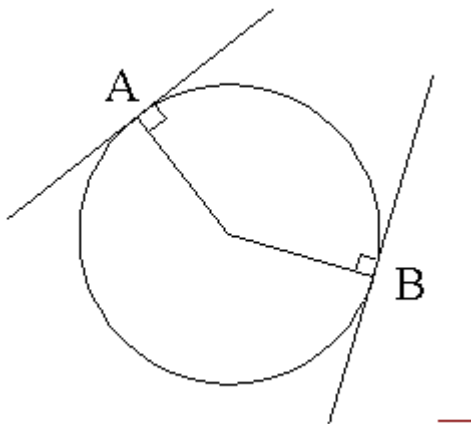
The circle is drawn once the center has been located, however the command assumes that you may wish to draw another circle with the same radius and it asks for another center point. This continues until the command is terminated by pressing Esc, Return, or selecting another command.

## Diameter

The use and the implementation of this option follows exactly the use and implementation of the option **Radius** except that instead of choosing the radius of the circle, the user must specify the diameter.

## Tangent

This option is frequently useful, as it will draw a circle of given radius or diameter **tangent to two lines**.



The first prompt will ask for the objects that the circle is to be tangent to. These objects must exist in the drawing before this command is issued. Pick both objects on the screen. In the example shown, we pick

the two lines.

After choosing the second object, enter the radius of the circle to be drawn. The radius of the last circle drawn is offered as default by the program. Verify this choice by pressing RETURN or enter a new value manually, or by "picking" it on the screen.

The circle will be placed so that it touches, tangentially, the two chosen elements with its circumference. In other words, the line from the circle center to the intersections is at right angles to the objects. This is illustrated in the example above, where points A and B are the points of calculated tangency.

If the chosen radius is smaller than the distance of the two objects to each other, the circle will be moved in the direction of the imaginary point of intersection, so that the defined radius will touch the extension of the elements. Note: the original objects are not extended. The extensions are only calculated to determine circle location.

If a tangential intersection is impossible (parallel lines, two concentric circles or arcs as chosen objects, etc.), the program gives the following message: "Circle does not exist!"

**Note:** The "smoothness" with which the circle is represented onscreen is controlled by the variable CIRCLESRES. By issuing the SETTINGS command, users can go to the Draw Modes tab and enter a new value (between 8 and 1024) in the Arc Segments box that controls smoothness. A low value will result on a rough arc or circle, while a higher value will produce a smooth arc or circle.

## Ellipse



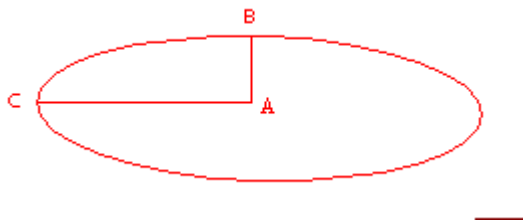
**Command: ELLIPSE**

**Alias ELL**

### Located:

1. **Pull-down Menu: Draw - Ellipse**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The command ELLIPSE allows you to draw a full ellipse.



The default is to identify *Center* point "A" of the ellipse first and then specify Point "B" (the first axis of the ellipse) Note that in this step, however, the orientation of the ellipse is established by either determining the location for "B" by picking it onscreen, or through distance/angle input as indicated below. Lastly, the secondary ellipse axis (A - C - at a 90° orientation from A - B) is input, either by picking point C, or entering the A-C dimension.

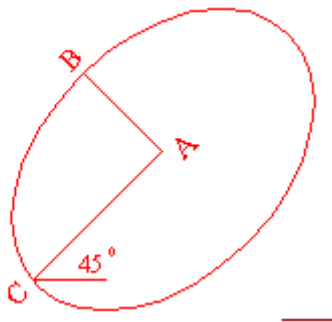
> ELLIPSE

Center point: Point A

Primary radius: **Dimension A-B**

Secondary radius: **Dimension A-C**

When you know the orientation of the **A-B** and **A-C** axes of the ellipse, you can specify this orientation when you input the primary A-B axis, by specifying the distance and angle of the A-B axis in relative terms, using the format (@distance<angle).



> ELLIPSE  
Center point:  
Primary radius: @3<45  
Secondary radius: 2

In this example, pictured above, the A-B alignment (3 units) has been rotated 45 degrees to the left, as has the C-D alignment.

Note: The ellipse will be created as a closed polyline with straight-line segments.

## Construction Points



**Command: POINT**

**Alias POI**

### Located:

1. **Pull-down Menu: Draw – Construction Point**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

It is possible to manually insert points of construction into a drawing using this command. Points can also be created in the drawing as the result of other commands and inputs, such as MEASURE and DIVIDE and the importation of survey data. The primary purpose of points is to serve as a reference in the construction of other drawing elements.

The form and size of the points is determined by specifying these parameters using the menu option *Drawing Parameters...* (See: Command DRAWMODE).

### Options:



### Point (Individual Points of Construction)

This option is activated automatically when calling upon this command and thus does not have to be called up separately.

Determine the position of the points to be inserted either via keyboard or via pointing on the screen.

The command remains active after the insertion of the first point, and the user can repeat this procedure as often as is required. End this function by pressing RETURN.



### Measure (Using Construction Points to Measure)

**Measure** allows the user to insert points of construction on 2-D lines, polylines, circles and arcs in a defined spacing.

This divides the entity into sections of a certain size. The drawing object is not cut into different pieces but remains one complete object.

Remember that all objects that are not an exact multiple of the length of the segment will have a remainder.

First choose a drawing element and determine the spacing of the segments (in drawing units), either by entering a numerical value or by cursor picks.

The insertion of the points into lines, polylines and arcs will be done beginning at the starting point of the segment and progressing to the end point. When inserting points of construction into a circle, the starting point lies at the intersection of the radius that has its starting point at the center point of the circle and is in the direction of the positive x-axis.

**Segments** makes it possible to separate lines, polylines, circles and arcs into segments of equal length by inserting points of construction. The size of the segment is determined by the length of the distance to be dissected and the number of construction points to be inserted. The DIVIDE command is the equivalent to Measure - Segments option.

In contrast to the option **Measure**, there will not be a remainder when using the **Segments** option.

The object being segmented is not cut into different pieces but remains one complete object.

First choose a linear object, and determine the number of segments into which the object is to be segmented. The number of segments should be a whole integer between 2 and 4096.

The insertion of points of construction into lines, polylines and arcs is done starting from the beginning of the segment and progressing to its end point, without inserting a construction point at the beginning or at the end. When inserting points of construction into a circle, the starting point lies at the intersection of a radius that begins at the center of the circle and continues along the positive x-axis. The beginning point is determined to be the end closest to the point at which the object was picked.

### Point Representation

Points can be shown onscreen in a number of different ways, apart from the common dot representation. The SETTINGS command's Construction Point tab has a number of alternate symbols that you may set as default. Note, however, that **all** construction points in the drawing will be shown with the same symbol, regardless of which you choose.

---

#### See also:

DRAWMODE

Polyline



Command: POLYLINE

Alias PO

**Located:**

1. **Pull-down Menu: Draw – Polyline**
2. **Left Side Bar**
3. **Palette: Draw**
4. **Or Enter Command or Alias at the Command Line**

**Note: For boundary polylines see Command PCONTOUR.**



Polylines are drawing elements that consist of connected lines or connected arc segments. Different segments can be of different widths. The width of a polyline segment can also be defined differently for the starting and end-point.

A polyline is defined as a single element and treated as such for further operations (like Edit, or Undo), in contrast to continuous lines drawn with the command Line.

For Polyline Type control, please refer to the variable PLINETYPE.

Polylines will always be drawn with solid fill.

When you give the POLYLINE command, several options appear on the Options Bar or Menu::

- Arc
- Width
- Halfwidth
- Back
- Closing
- Append

Selecting Arc will make the next segment an arc,

Width will specify the width if the next segment, Tapered, if the initial (default) width is adjusted in the end width entry.

Halfwidth will specify a linewidth based in a half linewidth measurement. Like Width, the begin and end widths of each segment can vary, giving a tapered segment width.

Regardless of whether they are arcs or lines, polyline segments are always drawn by defining their starting and end as well as width.

The program expects that the user will want to draw more than one segment when drawing a polyline. After drawing one segment the user will be asked to name another end point for a new polyline segment. The starting point will be the end point of the segment last drawn.

The user can end the drawing of the **Polyline** by pressing RETURN or ESC.

- First point: **P1**
- Next point: **P2**
- Next point: **P3**
- ...

## **Options**

## Width / Halfwidth

The options **Width** and **Halfwidth** are available from the Options Bar to assist in defining the width of a polyline segment. **Width** defines the entire width of a polyline segment, whereas **Halfwidth** defines the width as the distance of an imaginary centerline to one side of the polyline.

Verify the last entered value by pressing RETURN or enter a new width manually or by "picking" it.

The following rules are in effect when using **both** options:

If the user chooses the option **Width** or **Halfwidth** before determining the starting point for the first polyline segment, then the chosen width is valid for the entire length of the segment.

If the user chooses the option **Width** or **Halfwidth** after entering the starting point for the first polyline segment, it is possible to enter different values for the width at the starting point and end point.

The last chosen width will be used for all following polyline segments on its entire length. To determine another width, choose the option **Width** or **Halfwidth** again.

## Arc

**Polylines** has an option for drawing arcs, and a width may be correlated to the arced polyline. The method of drawing arcs is identical to drawing elements using the command ARCS.

To draw polyline arcs with a special line width, follow these steps:

- Determine the first point of the arc. This can also be the end point of the previous polyline segment when drawing a polyline;
- Call up the chosen option, **Width** or **Halfwidth**, and enter the parameters for the starting width and ending width.
- Determine the other parameters for drawing arcs using the same procedure as drawing "regular" arcs.

## Append

**Append** allows the user to connect a polyline with another line or an arc.

Choose the element (line or arc) to which the polyline still to be drawn is to be appended. Call the options **Width** or **Halfwidth**, if required, and determine the parameters for the starting width and the ending width.

## Polyline to Line

To choose an object, select an already existing line. Remember that the polyline will be connected to the end point of the line that is closest to the point selected when choosing the element.

To append a polyline, the user must now determine the end point of the line to be connected. This point will be appended to the end point of the chosen existing line.

## Polyline to Arc

To append a polyline to an existing arc, the user must enter a polyline length after choosing the existing arc. A polyline with the specified length will be connected tangentially to the end of the arc.

To end the polyline (appending), press RETURN.

## Undo

The option **Undo** allows the user to "undo" the last step taken. The end point of the last segment may then be used as a starting point of another segment if required. Undo's can be repeated as long as a segment exists. The last segment will be erased with each successive UNDO.

## Close

The option **Close** allows the user to draw a closed polygon. The two end points of the chain will be connected to each other and the command will be terminated.

---

**See also:**

[POLY3D](#)

Chain

## 3D Polyline



**Command: POLY3D**

**Alias 3P**

### **Located:**

- 1. Pull-down Menu: Draw – 3D – 3D Polyline**
- 2. Or Enter Command or Alias Directly at the Command Line**

3D Polylines are drawing elements created from straight-line segments. It is possible to define 3D coordinates for each individual 3D Polyline control point (start and end points of the individual segments).

The user may include arc segments in 2D polylines, but this is not possible with 3D polylines. It should also be noted that each individual element is assigned a standard width of zero.

As opposed to a line characteristic created using the line drawing command, the program treats 3D polygon characteristics as a single item, this also applies when the object is being processed with the edit or undo functions.

3D Polylines may be created by selecting the option from the menu item **3D Entities**, in the **Draw** menu, or by typing in the command POLY3D.

Conclude the 3D Polyline function by pressing the RETURN key.

```
> POLY3D
First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: RETURN
```

### **Options:**

#### **Undo**

The **Undo** option allows the user to undo the last line created. The end point of the last remaining segment will act as the starting point for the next segment.

The **Undo** function can be repeated as many times as necessary until the Undo List is empty. Each **Undo** step will erase the last drawn 3D polygon element.

#### **Close**

This option enables the user to create closed 3D Polygon objects.

Either selecting **Close** from the options bar or typing the **close** command at the keyboard will cause the last two remaining end points to be joined together.

---

**See also:**

2D Polylines

Rectangle





**Command: RECTANGLE**

**Alias RT**

**Located:**

1. **Pull-down Menu: Draw - Rectangle**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The command RECTANGLE allows you to construct rectangles of any size and side ratios.

To draw a rectangle, enter or pick two opposite corners.

When the command is active, the option *Width* is available in the Options Panel. The option **Width** allows you to specify a value for the line width applied on the rectangle creation. To use the last entered value (that will be offered by the prompt) just press RETURN.

### Examples

```
> RECTANGLE
First point of the rectangle: P1
Opposite corner point: P2

> RECTANGLE
First point of the rectangle: Width
Width <0.05>: 1.00
First point of the rectangle: P1
Opposite corner point: P2
```

## Rings



**Command: RING**

**Alias RI**

**Located:**

1. **Pull-down Menu: Draw - Ring**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**



A ring is a special case of a closed polyline. It is used to draw solid rings or circles. Two concentric circles are drawn, the smaller one is the inner diameter, and the larger one the outer diameter, of the ring. The space between the circles will be filled.

Rings can now be edited using any of the editing commands for polylines and will behave just like a polyline object.

The last entered values will be used as an option for the next entry when drawing rings. If the user wants to keep using the same values, just enter a new center point. This step can be repeated as often as needed, always creating identical rings.

> RING  
Center point: **P1**  
Center point: **P2**  
...

### Options:

#### Diameter

To determine new values for the inside or outside diameter, just choose the option **Diameter** from the options panel.

With this option it is possible to set the inside and outside diameter of the rings, where new values can be input either via keyboard or by picking points on the screen.

> RING  
Center point of the ring: **Diameter**  
Inside diameter <0.500>: **0.75**  
Outside diameter <1.00>: **1.25**  
Center point: **P1**  
Center point: **P2**  
...

With command repeat terminated by Esc, it is possible to draw as many identical rings as required by simply locating their successive center points.

#### Filled Circle

A filled circle is a special case of rings. It is a ring with the interior diameter equal to zero.

#### Trapezoid

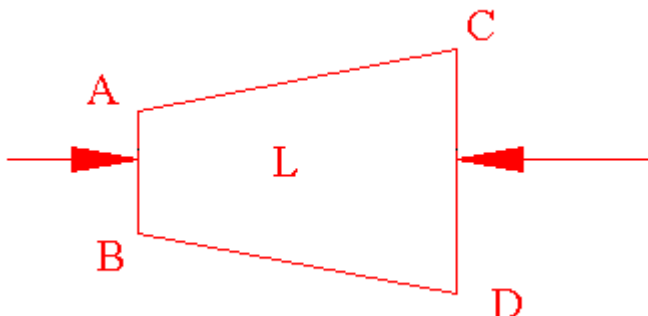


**Command: TRAPEZOID**

**Alias TZ**

#### Located:

1. **Pull-down Menu: Draw - Trapezoid**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**



With the command TRAPEZOID it is possible to draw a single conical element (trapezoid) on the active layer.

Start this function by choosing the like-named option in the menu *Draw*, its palette symbol, or through entering the command TRAPEZOID.

After entering the point and insertion angle, the cone is defined by its length and the diameter of its two ends. To do that, determine the needed points successively through pointing or entering the values or coordinates manually.

The cone can also be drawn using a **Ratio** of tapering. To do that, press RETURN after the question of the diameter of the second side and enter a tapering ratio at the asking.

Command input sequence:

> TRAPEZOID

Insertion point: **Point A**

Insertion angle <0>: **Angle L will be oriented to**

Diameter of first side: **Dimension A-B**

Length: **Dimension L**

Diameter of second side: **Dimension C-D**

## Filled Faces



**Command: 2DFACE**

**Alias 2F**

**Located:**

1. **Pull-down Menu: Draw – Filled Faces**
2. **Or Enter Command or Command Alias Directly at the Command Line**

**2D Faces** make it possible to draw solid filled planes that have straight-line borders. This is done by creating and connecting any number of three or four-cornered planes. It may be done continuously or using segments. It is also possible to connect 2D planes to already existing 2D faces.

2D Faces are always drawn by determining the corner points (three or four). The option **Close** is available to draw triangular planes so that the third entered point is connected with the starting point of the plane.

### Indication

Remember, when defining the end points of the edges that the order in which these points are entered influences the resulting shape.

The user should always specify the end points of the edges in the same direction. If the end points of the edges of a plane are determined clockwise or counterclockwise, a "butterfly" effect results.

Options

### Continuous Filled Faces

"Continuous" is the standard option for drawing filled planes; it will be used automatically if no other option is chosen.

When drawing rectangular planes, determine the end points of the edges. Remember that the first and third and the second and fourth point are connected. The edges will make up the plane.

After determining the end points of the first two edges, the user may enter additional points, and they will be used as third and fourth edge (and so on) points for another plane immediately connected to the first one. To terminate the command, press RETURN.

**Close**

To draw a triangular plane, enter three end points for two of the edges successively, and choose the option **Close** from the options panel or enter the command **Close** manually.

### Filled Face Segments

The option **Segments** is used to draw a single rectangular or triangular filled plane. In contrast to the option **Continuous**, this option will end after the entry of the points required for the plane.

### Appending Filled Faces

This option makes it possible to append newly created planes to already existing ones. It is only possible to append a plane to the edge last drawn.

After entering the option **Append**, choose a side from an existing plane to which the new plane is to be appended. Determine the corner points of appending planes.

```
> 2DFACE
Choose a side: Choice
Third point: P5
Fourth point: P6
...
```

Terminate the command by pressing RETURN.

## 3D Faces



**Command: 3DFACE**

**Alias 3F**

### Located:

1. **Pull-down Menu: Draw – 3D – 3D Faces**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

The **3D Faces** drawing function creates three or four sided surfaces "in space", or, in other words, in an X, Y, Z coordinate system. The difference between 2D and 3D surfaces is that the 3D surfaces are not shown as solid areas, but are drawn as wire frame models.

The 3D Faces definition points are fixed in either a clockwise or counter-clockwise direction so that one edge of a 3D surface is used as the base line for the next 3D surface.

The user may also draw "invisible" lines, a function that may increase the clarity of complicated 3D structures.

### Options

#### Continuous

The option **Continuous** is the default option when drawing 3D surfaces.

To draw a continuous 3D surface, define the corner points in a clockwise or counter-clockwise direction.

For a triangular surface, define all three corner coordinates and then end the drawing with the **Close** option located. The last defined coordinate will then be joined to the first coordinate location, forming a triangle, and the drawing will be completed.

```
> 3DFACE
First point: P1
Next point: P2
Next point: P3
Next point: Close
```

For a rectangular 3D surface, define the four corner coordinates. After having completing the fourth entry, the last coordinate will be joined to the origin, thus creating a four-sided object.

This process is then continued. The user will be asked to define further points or coordinates. The next 3D surface to be defined will use the last drawn line as the starting point or base line for the next 3D surface.

End the 3D surface drawing process by pressing the RETURN key.

```
> 3DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
Third point: P5
Fourth point: P6
Third point: P7
Fourth point: P8
... and so on.
```

### 3D-Face Segments

To draw a single 3D Face, use the **Segment** option. The method is almost identical to that just described.

```
> 3DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
```

In contrast to the **Continuous** option, the drawing process will end with the definition of the fourth point and the consequent closing of the contour. No further surfaces will be "connected".

This option also allows the use of the **Close** function, thus enabling triangular shaped 3D surfaces to be created.

Should the user wish to add further 3D surfaces to an existing one, this can be achieved at any time by utilizing the **Append** option.

### Append

This option allows the user to connect new 3D surfaces to existing objects. First select the edge that is to act as the base line for the connection, then proceed by defining the required corner points for the new 3D surface.

The **Append** option will automatically change the mode to **Continuous**. This means that after having connected one 3D surface the user can define additional pairs of coordinates to create more surfaces.

### Invisible Edges

The **Invisible** option is used to draw hidden lines in a 3D-Face. This feature is particularly useful when dealing with complex drawings containing a large number of 3D surfaces. The use of hidden lines will make the drawing more understandable.

This function is available for all modes used in 3D surface drawings. The function must be activated prior to the definition of the starting coordinate of the invisible line.

## 3D Mesh

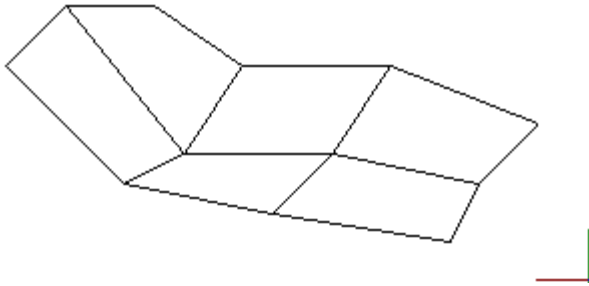
**Command: 3DMESH**

**Alias 3M**

**Located:**

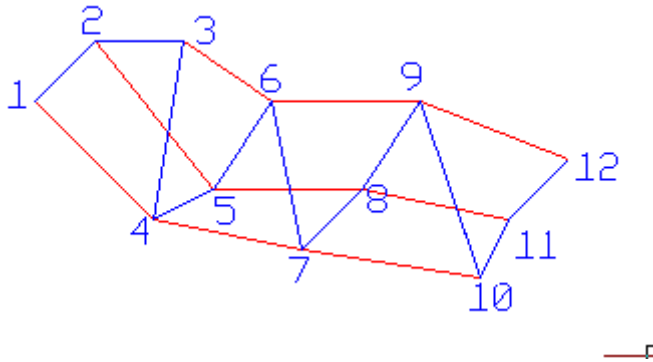
1. **Pull-down Menu: Draw – 3D – 3D Mesh**

## 2. Or Enter Command or Alias Directly at the Command Line



The 3DMESH command creates polygon meshes. Usually it is used to generate irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

A 3D mesh is a wireframe rectilinear blanket composed of  $M$  column lines by  $N$  row lines passing through a matrix of  $M \times N$  three-dimensional points in space.  $M$  and  $N$  are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh.



### To Create a Polygon Mesh

First you are prompted to specify the mesh  $M$  size and the Mesh  $N$  size. This is simply the number of wire intersections wide and long.

Then the program continues to ask for corners or vertices, until you have defined all of the vertices in the one ( $M$ ) direction and the other ( $N$ ) direction. Remember that to complete the mesh, you will need to enter  $M \times N$  vertices. You are prompted for each vertex in order, starting with vertex (0,0), continued with the second vertex in the first column (0,1), and so on. Once all vertices of a column are specified, the first vertex of the next column is requested. The blue line traces the input order in the example further below.

1. During vertex input, the mesh does not appear onscreen but is instead being constructed in memory. Terminating the command places the mesh onscreen
2. Meshes can be open or closed depending on whether the mesh joins in either the  $M$  or  $N$  direction, or both. A donut mesh would be an example of a mesh closed in both directions.

### Example

```
> 3DMESH
M-value of the mesh (between 2 and 256): 4
N-value of the mesh (between 2 and 256): 3
Vertex (0,0): 110,170,4
Vertex (0,1): 130,190,4
Vertex (0,2): 160,190,6
Vertex (1,0): 150,130,0
```

Vertex (1,1): **170,140,0**  
Vertex (1,2): **190,170,2**  
Vertex (2,0): **200,120,0**  
Vertex (2,1): **220,140,-2**  
Vertex (2,2): **240,170,0**  
Vertex (3,0): **260,110,0**  
Vertex (3,1): **270,130,0**  
Vertex (3,2): **290,150,0**

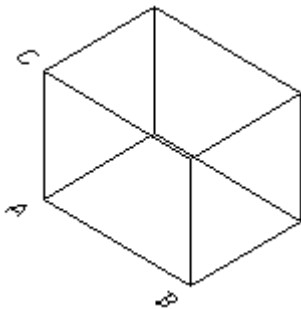
## Command 3DBOX

**Command:** 3DBOX

**Alias** 3B

### **Located:**

1. **Pull-down Menu: Draw – 3D – Box**
2. **Or Enter Command or Alias at the Command Line**



The command 3DBOX allows simple creation of a 3 dimensional box. The command can also be accessed from the Draw - 3D pull-down menu.

Upon executing the command, you are asked to input the first corner ("A" in the diagram). As with the input of all location definition, you may enter coordinate values in the form x,y,z or you may pick an existing reference point in your drawing. Unless the point you want has a z or height/depth value of zero (default value for 2D objects), be sure that the point you pick has the correct z value.

Once the first corner has been established you will be presented with an option dialog offering you the opportunity to input a length to establish the second corner. The z value (height) of the second point is the same as the first and the first side is parallel to the x-axis (Y values of A and B are equal).

You will be prompted for width if you chose the length option and presented with a dialog offering Cube and Square options.

Selecting **cube** indicates that the x, y and z dimensions are equal to A-B and the cube is completed. Choosing **square** indicates that the x and y values (only) are equal to A-B and the square is drawn with a following prompt for height input, that completes the box.

The second point (picked point "C") would be regarded as the diagonally opposite corner of a rectangle and be drawn, if you were to ignore the Length option and input a point with a different y value, after setting the first corner. A prompt for height would follow.

For height, an absolute value can be input. Picking a point "D", however, will set height as being the 3 dimensional distance between points "A" and "D".

## Command 3DCONE

**Command:** 3DCONE

**Alias** 3DC

### **Located:**

1. **Pull-down Menu: Draw – 3D – Cylinder / Cone**
2. **Or Enter Command or Alias at the Command Line**



The command 3DCONE allows creation of a 3D cone, cylinder, prism, or pyramid. This command can also be executed from the Draw - 3D pull-down menu. Object creation is determined by user input. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

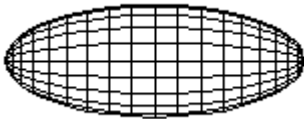
## Command 3DELLIPS

**Command:** 3DELLIPS

**Alias** 3L

### **Located:**

1. **Pull-down Menu: Draw – 3D – Ellipsoid**
2. **Or Enter Command or Alias at the Command Line**



The command 3DELLIPS (Draw>3D>Ellipsoid) allows creation of a 3D ellipsoid object. This command may also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

## Command 3DSPHER

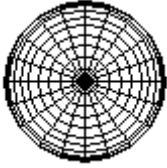
**Command:** 3DSPHER



## Alias 3DS

### Located:

1. **Pull-down Menu: Draw – 3D – Sphere**
2. **Or Enter Command or Alias at the Command Line**



The command 3DSPHER creates a 3D sphere (ball). This command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

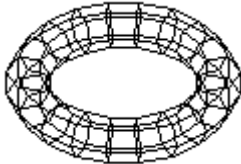
## Command 3DTORUS

**Command: 3DTORUS**

**Alias 3T**

### Located:

1. **Pull-down Menu: Draw – 3D – Torus**
2. **Or Enter Command or Alias at the Command Line**



The command 3DTORUS allows creation of a 3D torus. The command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

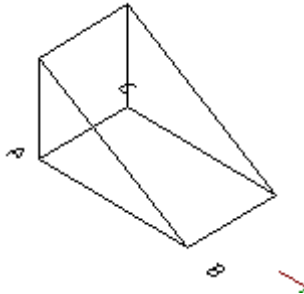
## Command 3DWEDGE

**Command: 3DWEDGE**

**Alias 3W**

### Located:

1. **Pull-down Menu: Draw – 3D – Wedge**
2. **Or Enter Command or Alias at the Command Line**



The command 3DWEDGE allows creation of a 3 dimensional wedge. This command can also be entered from the Draw - 3D pull-down menu.

## Polygon (N-gon)



**Command: NGON**

**Alias NG**

### **Located:**

1. **Pull-down Menu: Draw - Ngon**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**



The command **N-gon** is used to draw regular polygons. It is possible to draw polygons with up to 1024 sides, a value that should lie above any practical need.

When using this command, three options, based on different principles of construction, are available:

1. Definition of the length of sides of the polygon;
2. Definition of a center point and a radius of a circle outside ; or
3. Definition of a center point and a radius of a circle inside.

Select the command for drawing a polygon and choose one of the options from the options panel:

**Sides**  
**Outside**  
**Inside**

### **Options:**

#### **Length of the sides**

**Sides** is the standard option for drawing regular polygons. It will be used if no other option is chosen.

The regular polygon is defined by the length and placement of one side.

To draw a regular polygon, enter the number of its sides first. To do that, enter a value between 3 and 1024 and verify this entry by pressing RETURN.

If there have been polygons drawn previously, then a number will appear in the prompt. That number is the number of sides of the polygon drawn last. The user may elect to use that number by simply pressing RETURN.

After specifying the number of sides, determine the starting point and the end point of the first side either by entering the coordinates or by picking it with the cursor. Remember that the polygon will be drawn counterclockwise, starting from the first side.

```
> NGON
Number of sides <4>: 6
First corner point of the side: P1
Distance or second point of the side: P2
```

## Circumscribing the Polygon

### Outside

The option **Outside** defines a polygon by determining a center point and the distance to its corner points. This distance is the radius of a circle that contains the polygon, a point on whose circumference all corner points of the polygon lie.

After entering the number of sides, choose the option **Outside** from the Options Panel.

Determine the center point of the polygon by entering its coordinates or picking it. Next, determine the distance to its corner points. The polygon will be drawn counterclockwise, starting from the first corner point.

```
> NGON
Number of sides <4>: 6
Center point of the polygon: P1
Distance to its corners: P2
```

### Inside

**Inside** allows the polygon to be defined by the circle whose circumference touches each side of the polygon at its mid-point.

Choose the option **Inside**, then enter the number of sides. Next determine the center point of the polygon by entering its coordinates or "picking" it; then determine the radius of the circle. The polygon will be drawn counterclockwise, starting from the thus defined side.

```
> NGON
Number of sides <4>: 6
Center point of the polygon: P1
Distance to its corner points: P2
```

## Boundary Hatching



**Command: BHATCH**

**Alias BH**

### Located:

1. **Pull-down Menu: Detail – Boundary Hatch**
2. **Palette: Detail**
3. **Or Enter Command or Alias Directly at the Command Line**

The command BHATCH fills a region with a hatch pattern or solid, bounded by linear entities, just by pointing to any point within the boundary.

The BHATCH command defines a boundary by creating a polyline that follows the entities that make up

the boundary. Select each entity to define a boundary around the area you want to hatch.

> BHATCH

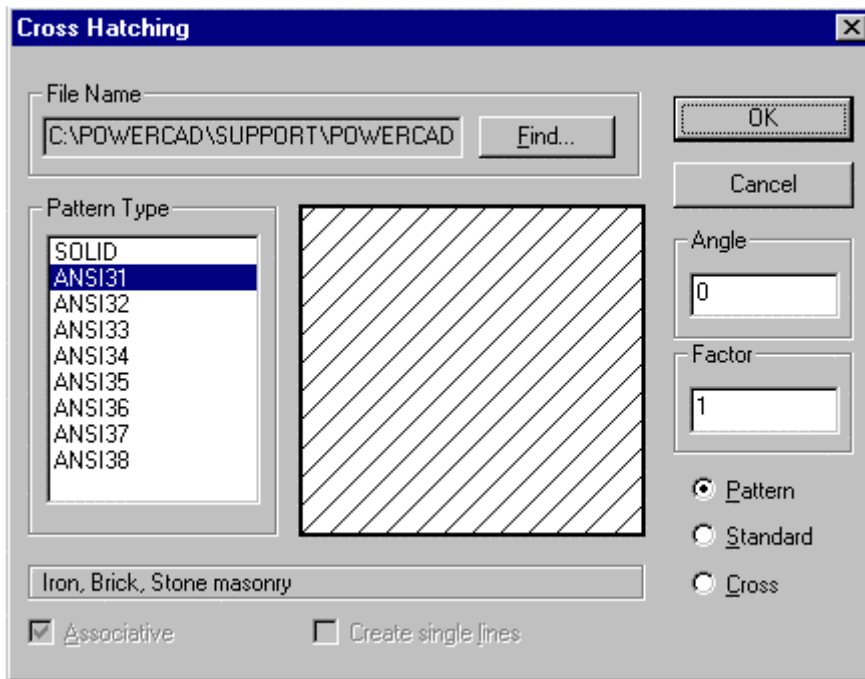
Select objects:

Select point inside boundary contour <+X>:

Select additional elements? <N>:

<Dialog box appears>

Note: to exclude internal areas from hatch application, select the internal boundaries to be ignored when prompted for "Select additional elements."



Hatch patterns are stored together in **.pat** files. The file name input at the top of this dialog allows you to select a pattern from any of a number of .pat files that you may have available. Having selected the file, all patterns included in that file are listed in the Pattern Type window on the left. Clicking on any of these causes a representation of that pattern to appear in the central window, making selection easier.

The selections to the right on this dialog modify the pattern, in which:

- Angle – Will cause a counter-clockwise rotation of the pattern by the angle entered;
- Factor – Will cause the pattern to become more condensed or expanded, depending upon the entered factor value;
- Pattern will display a non-uniform pattern (if that is available in the particular selection);
- Standard – Will modify the pattern to a uniform spacing and
- Cross – Will create a "criss-cross" pattern effect.

The hatch will be applied immediately after the OK button is pressed.

### Hatch Color

Hatch color is, as with other drawn entities, determined by current layer color setting, or else by a set color that over-rides bylayer color settings. Color fills are achieved by making the selection of "Solid" in the pattern name window.

Further information on hatch pattern definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

---

**See also:**

Hatching  
Modify Hatching  
Polyline Boundary  
Polyline Contour

## Polyline Contour



**Command: PCONTOUR**

**Alias PC**

### Located:

1. **Left Side Bar**
2. **Entered Directly at the Command Line**

This command allows creating a closed polyline on top of (overlying) objects that make up a closed boundary. Such a polyline contour can serve for hatching regions of the drawing with the HATCH command or to determine the area or circumference of a region with the AREA command (applying the option *Object*). The BOUNDARY command is identical in operation to this command and is provided for users who may be more familiar with the term.

From the menu *Draw*, choose **Polyline Contour**. At the command prompt, enter PCONTOUR.

It is recommended that you set a specific layer before invoking the command (like a hatch or contour layer), because the polyline contour will overlap entirely or partially existing drawing entities, making selection otherwise difficult.

After giving the command, you are prompted to select objects to be examined to find the contour.

**Important Note:** At this point, it is not required to select only those entities that should make up the contour. You might enter the option *All* as object selection mode, however, the supplying a smaller selection set can improve the command efficiency, as fewer objects are evaluated in boundary establishment.

After the selection set to be examined has been created, the following **Ray Casting** methods for contour finding are available to find the next boundary object: Normally, the default works well, however there may be instances where you may wish to select a specific option:

1. **Nearest** sends radial rays from the pick point you specify (see below) to the nearest object.
2. **+X** sends a linear ray in the positive X direction from the pick point you specify to find an entity.
3. **-X** sends a linear ray in the negative X direction from the pick point you specify to find an entity.
4. **+Y** sends a linear ray in the positive Y direction from the pick point you specify to find an entity.
5. **-Y** sends a linear ray in the negative Y direction from the pick point you specify to find an entity.

Next you are prompted to pick the start point from which the program begins assessing the intended boundary..

Once a boundary object has been found in the direction of the specified pick point, the command traces the boundary in a counterclockwise direction.

After analyzing the specified selection set successfully, a closed polyline will be produced on the current layer, covering the boundaries of overlying drawing entities.

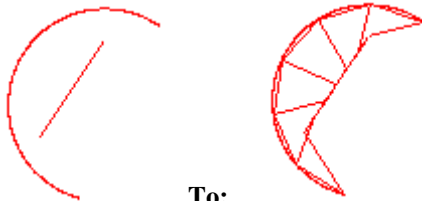
## Ruled Surface

**Command: RULESURF**

**Alias RSF**

### Located:

1. **Pull-down Menu: Draw – 3D – Ruled Surface**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**

The RULESURF command allows you to create a ruled surface between two boundary curves (two profiles).

Valid entities that define the edges of the surface can be points, lines, arcs, circles, 2D polylines or 3D polylines.

The system variable SURFTAB1 determines the number of equal intervals between the resulting vertices on the defining profile entities. The number of intervals is the same for each curve.

The resulting surface is a polygon mesh (stored in the drawing database as a complex polyline entity).

### To create a tabulated surface

First pick one entity defining the first profile, then select another entity for the other boundary outline:

> **RULESURF**

Select first defining curve of ruled surface:

Select second defining curve of ruled surface:

### Note

If the first entity is an open entity (such as an arc or line), the second entity must also be an open entity. If the first entity is a closed entity (such as a circle or a closed polyline), the second entity must also be a closed entity. If one entity defining an edge is a point, the other entity cannot be a point entity.

### Tip

To create a commonly used polygon mesh, select each entity at the same end. If you select each outline entity at opposite ends, you generate a self-intersecting polygon mesh.

Please check in the Learning Center for a fully documented example of this command's use.

## Copy



**Command: COPY**

**Alias C**

### Located:

1. **Pull-down Menu: Modify – Copy**
2. **Top Function Bar**
3. **Palette: Modify**
4. **Or Enter Command or Alias Directly at the Command Line**

## Duplicating drawing objects

This function allows the user to copy drawing elements to AllenCad's clipboard and later put them at a specified place.

The copies created by this command are exact copies of the selected entities. All entity attributes such as layers, line type, color, and all other attributes are copied as well as the outline of the object(s).

### Options:

#### Basepoint

If **Basepoint** is the selection, enter the base point, then, as prompted, the **Target Point** (the position to which the Copy is to move). The copy will be made in the position specified.

#### Rel.Point

**Rel.Point** may be chosen if the copy is to be placed in a position relative to the current image of the object.

The prompt will ask for the relative points by specifying the move in the X, Y, and Z directions. If the move is to be made in the X and Y directions only, simply do not specify a Z delta. The copy will be placed in the specified position.

#### Offset



**Command: OFFSET**

Alias O

#### Located:

1. **Pull-down Menu: Modify – Offset Through Point**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Offset** creates congruent pictures of lines, circles, arcs and 2D-polylines. This command enables a copy of a selected entity to be created and placed a distance from the original entity. The original entity(s) stays in position.

The operating mode of the command depends on the type of drawing entity:

- For lines and polylines identical copies are created and moved in the direction of the X or Y-axis.
- For circles and arcs copies are also created, but, in addition, they will be scaled by a selected factor. As a base point for scaling the center of the original circle or arc is used. In this way, concentric circles or arcs are created.

#### Options

##### Offset for Lines/ 2D-Polylines

After entering the command, select the line(s) or polyline(s) that are to be offset by "picking" with the cursor. Then you either select the distance for the offset by picking, or select **Distance** from the Option Bar, and enter the offset distance. Then answer the prompt "Side to Offset?" with a point on the desired side.

##### Offset for Concentric Circles/ Arcs

Concentric circles or arcs can be created with a radius or diameter smaller or larger than the original entity.

After having selected the objects to be offset, the position of the offset is determined as follows:

- By indicating, or picking, a point;
- By input of coordinates of a point on the circle or arc; or
- By using the option **Distance** from the Option Bar (see below).

#### **Distance (Command DOFFSET)**

##### **Located:**

1. **Pull-down Menu: Modify – Offset Distance**
2. **Or Enter Command or Alias Directly at the Command Line**

The option **Distance**, selected from the Option Bar, allows the offset to be created a specific distance from the original entity.

Next, determine the distance between the original entity and the offset. Do by either a numeric value input (in drawing units) or by cursor selection.

Then, point (and click) with the cursor to the side of the original entity on that the offset is to occur.

---

##### **See also:**

DOFFSET

Mirror



**Command: MIRROR**

**Alias Mi**

##### **Located:**

1. **Pull-down Menu: Modify - Mirror**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Mirroring drawing objects produces identical copies of selected entities that are placed in a "mirrored" position about an axis specified by the user.

The selected objects have, after "mirroring", the same relative position to the symmetry axis as they had prior to " mirroring".

The mirror axis can be defined freely, or parallel to the X-axis or the Y-axis

In order to use the command the user must select an object to be mirrored. Select the object or objects, remembering that the selection of multiple objects will result in the treatment of the objects as one.

The next prompt will ask for the definition of the mirroring axis.

#### **Options:**

##### **Mirroring With a Free Axis**

The Option **Axis** allows the user to freely define the mirror axis. It is the standard option for mirroring objects and is used when no other option has been selected.

Select this option, then define the position of the mirror axis. This can be made by indicating two points or by inputting their coordinates. The mirror axis can have any alignment with regard to the objects to be mirrored as well as with regard to the axes of the coordinate system.

If the mirror axis should run through points of an existing geometry (for instance, along a solid edge), the user should employ the object snap options for precise definition.



### Mirroring on a Horizontal Axis

To choose a horizontal-mirroring axis, select the option **Horizontal**. This option creates a mirror axis that runs parallel to the Y-axis of the coordinate system. Its exact position is indicated by a single point.

### Mirroring on a Vertical Axis

To choose a vertical-mirroring axis, select the option **Vertical**. This option creates a mirror axis that runs parallel to the X-axis of the coordinate system. Its exact position is indicated by a single point.

### Note: Mirroring Text

If there is text associated with the selected object or objects, it is likely that the user does not want to mirror that text. In order to mirror the text it is necessary to activate an option called **Text Mirroring**. Activating this option will result in the text also being mirrored.

## Polygon Face

### Command: PFACE

Alias PF

#### Located:

1. Pull-down Menu: Draw – 3D – Polygon Mesh
2. Or Enter Command or Alias Directly at the Command Line

The PFACE command creates three-dimensional polygon meshes of arbitrary topology and surface properties.

### To create a polyface mesh

Specify the control points (vertices) first.

Then specify that vertex is on that face of the polyface mesh.

### Note:

The PFACE command is rarely used as an explicit command in drawing practice. However, the command provides an easy way to generate meshes by Lisp routines based on 3D point lists (like surveyor's measurement points). The command is primarily designed for use within applications (either based on Lisp or on other programming languages) to construct general polyface meshes.

Do not confuse polyface meshes created by PFACE with polygon meshes created by the 3DMESH command.

### Example

```
> PFACE
Control point 1: Specify a point
Control point 2: Specify a point
Control point 3: Specify a point
Control point 4: Specify a point
Control point 5: Specify a point
Control point 6: Specify a point
Control point 7: Specify a point
Control point 8: RETURN
Face 1, Control point 1: 1
Face 1, Control point 2: 2
Face 1, Control point 3: 3
Face 1, Control point 4: 4
Face 1, Control point 5: RETURN
Face 2, Control point 1: 1
Face 2, Control point 2: 4
```

Face 2, Control point 3: **5**  
 Face 2, Control point 4: RETURN  
 Face 3, Control point 1: **5**  
 Face 3, Control point 2: **6**  
 Face 3, Control point 3: **7**  
 Face 3, Control point 4: RETURN  
 Face 4, Control point 1: **4**  
 Face 4, Control point 2: **5**  
 Face 4, Control point 3: **6**  
 Face 4, Control point 4: RETURN  
 Face 5, Control point 1: **3**  
 Face 5, Control point 2: **4**  
 Face 5, Control point 3: RETURN  
 Face 6, Control point 1: RETURN

Another null response terminates the command.

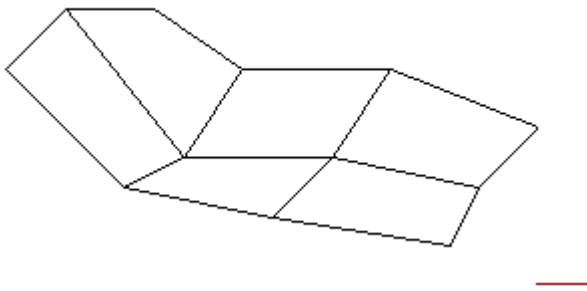
## 3D Mesh

**Command: 3DMESH**

**Alias 3M**

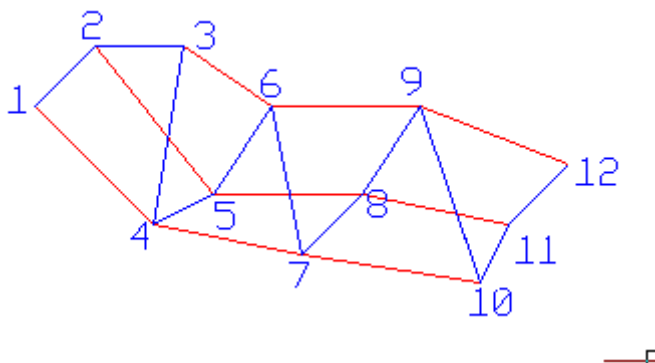
**Located:**

- 1. Pull-down Menu: Draw – 3D – 3D Mesh**
- 2. Or Enter Command or Alias Directly at the Command Line**



The 3DMESH command creates polygon meshes. Usually it is used to generate irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

A 3D mesh is a wireframe rectilinear blanket composed of  $M$  column lines by  $N$  row lines passing through a matrix of  $M \times N$  three-dimensional points in space.  $M$  and  $N$  are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh.



## To Create a Polygon Mesh

First you are prompted to specify the mesh *M* size and the Mesh *N* size. This is simply the number of wire intersections wide and long.

Then the program continues to ask for corners or vertices, until you have defined all of the vertices in the one (*M*) direction and the other (*N*) direction. Remember that to complete the mesh, you will need to enter *M* x *N* vertices. You are prompted for each vertex in order, starting with vertex (0,0), continued with the second vertex in the first column (0,1), and so on. Once all vertices of a column are specified, the first vertex of the next column is requested. The blue line traces the input order in the example further below.

1. During vertex input, the mesh does not appear onscreen but is instead being constructed in memory. Terminating the command places the mesh onscreen
2. Meshes can be open or closed depending on whether the mesh joins in either the *M* or *N* direction, or both. A donut mesh would be an example of a mesh closed in both directions.

### Example

```
> 3DMESH
M-value of the mesh (between 2 and 256): 4
N-value of the mesh (between 2 and 256): 3
Vertex (0,0): 110,170,4
Vertex (0,1): 130,190,4
Vertex (0,2): 160,190,6
Vertex (1,0): 150,130,0
Vertex (1,1): 170,140,0
Vertex (1,2): 190,170,2
Vertex (2,0): 200,120,0
Vertex (2,1): 220,140,-2
Vertex (2,2): 240,170,0
Vertex (3,0): 260,110,0
Vertex (3,1): 270,130,0
Vertex (3,2): 290,150,0
```

## Array

**Command: ARRAY**

**Alias RA**

### Located:

1. **Pull-down Menu: Modify - Array**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Array** represents a variant of Copy, except that Array will create multiple copies, and they can be arranged at random, or in rows, or in vertical order, or on a circle, or on an arc.

The entities created by the command are exact copies of the selected entities in which all entity attributes such as layers, line types and color are copied.

After entering the command, select an option from the Option Bar:

Individual  
Rectangular  
Circ-NRot  
Circ-Rot

### Options



### Individual Array

This mode allows the user to copy the Array as many times as is required during one command sequence and place them at random on the drawing area.

The use of this option is very similar to the command Copy.

After choosing the option, select the entities to be copied using the selection modes described in the Option Bar. Always confirm object selection by pressing RETURN.

Determine the base point for the copying. A particularly discreet point in the geometry or in one of the selected objects (edges, corners, intersecting points etc.) will be useful. Next, either input the coordinates of the target point(s) or "pick" the points for placement of the Array. The program will create a copy of the Array after each definition of a target point.

As long as the command remains active the user can produce and place as many copies as are required. Terminate the function by pressing ESC or RETURN.



### Rectangular Array

This option will arrange the Array copies in rows or columns. The use of this function is the same as was the use of the **Individual** mode.

Determine the number of the columns and rows in which the Array copies should be arranged. Define the distance between the columns and the rows.

Note that, when planning the arrangement, the rows and columns must be in the positive X or Y-axis. This infers that the original Array is situated in the lower left-hand corner of the quadrant plus-XY. If it is not located there, it may be useful to move the original Array before attempting to begin defining the copying positions. Use the command Move to do this.

### Circular Arrays

The Array can also be reproduced in a circular path. The circle (arc) itself is not displayed on the screen. The reproduction of the Array is described as follows:

1. The center of the circle (arc)
2. The number of times the Array is to be reproduced;
3. The included angle of the arc (360 if the path is to be a full circle) to be filled;

There are two ways to reproduce the Array on a circular path:



### Placing the Arrays Without Rotation

This Option, will arrange the reproductions on the circle (arc) in the same alignment to the principal axes as the original Array.

Define the center of the circular path. Then determine **either**;

The number of the entities to be arrayed (including the original); and  
the included angle of the circle (arc),

or

The included angle of the circle (arc); and  
the included angle between the Arrays to be reproduced (measured **between** the reference points of the objects).

Note that, during the planning of the reproduction of the Array on a circle that the circle (arc) runs **counter-clockwise** from the starting point. For a **clockwise rotation** enter negative values for the angle inputs.



### Placing the Arrays With Rotation

This option will rotate the Arrays when placing them on the circular path. The Arrays will be rotated in the same direction as the circle arc that describes the path of reproduction. The rotation angle will correspond to the angle between the Arrays (measured between the reference points of the Arrays).

After selecting the command, choosing the objects in the Array, and choosing the option **Ci.Rot** from the Option Bar, define the center of the circular path and then determine **either**;

the number of the entities to be arrayed (including the original); and  
the included angle of the circular arc,

or

the included angle of the circular arc; and  
the included angle between the entities to be arrayed (measured between the reference points of the objects).

As parameter value for the Arc angle there is always used the standard value 360 degree, i.e. the copies are arrayed on a full circle, if this value has been confirmed.

Please note during the planning of the arraying on a circle of multiple copies that the circle (arc) runs from the starting point that is the original object anti-clockwise. For a clockwise rotation please enter negative values for the angle inputs.

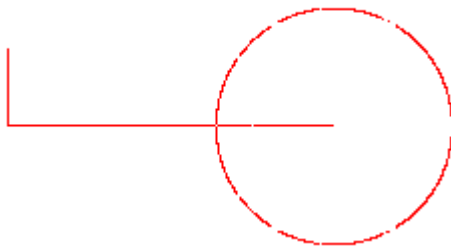
### Surface of Revolution

**Command: REVSURF**

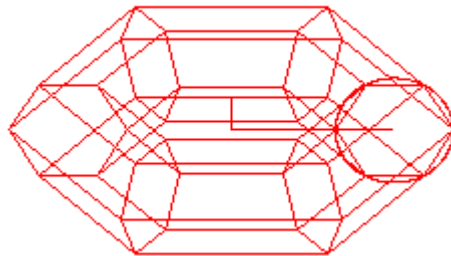
**Alias RS**

#### Located:

1. **Pull-down Menu: Draw – 3D – Surface of Revolution**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**



**To:**

The REVSURF command creates a surface by revolving a path curve (the profile) around an axis of revolution.

The curve sweeps around the selected axis to define the surface in the *N* direction of the mesh. The *M* direction of the mesh is determined by the axis of revolution. The result is determined by the SURFTAB1 and SURFTAB2 settings.

#### To generate a surface of revolution

First, pick the entity (a line, arc, circle, or polyline) to revolve. This determines the profile entity.

Next, pick the axis (a line or open polyline) around that the profile entity revolves.

Finally, you are prompted to determine the start angle and the included angle to revolve the profile. The default is a full circle.

> REVSURF

Select path curve to revolve:

Select axis of revolution:

Start angle <0>:

Included angle to revolve entity (+ccw, -cw) <360>:

**Tip**

To close the mesh in the *N* direction, select a circle or closed polyline as the curve.

Please check in the Learning Center for a fully documented example of this command's use.

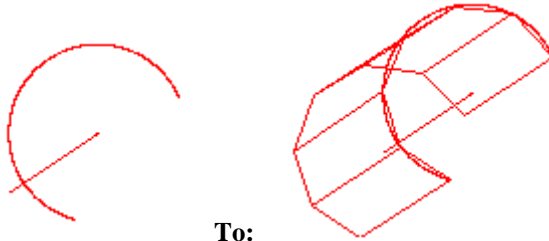
**Tabulated Surface (Extruded Surface)**

**Command:** TABSURF

**Alias** TF

**Located:**

1. **Pull-down Menu: Draw – 3D – Tabulated Surface**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**

The TABSURF command creates a tabular surface using a path curve and a direction vector.

You can extrude lines, arcs, circles, 2D or 3D polylines. The entity provided as path curve defines the surface of the mesh. The direction vector (extrusion path) must be determined by picking a line or an open polyline.

The system variable SURFTAB1 defines the number of *M*-direction surfaces on the extruded entity. When extruding a line, arc, or circle, the number of *M*-direction surfaces is evenly divided determined by the value of the SURFTAB1 system variable. When extruding a polyline, extrusion lines are drawn at the ends of the line segments, and the arc segments are divided evenly into the number of surfaces determined by the value of the SURFTAB1 system variable.

If the polyline has been curved (fitted) or splined, the number of *M*-direction surfaces is evenly divided using the SURFTAB1 variable. Like with the command RULESURF, you only need to set SURFTAB1 system variable.

The resulting surface is a polygon mesh (stored in the drawing database as a complex polyline entity).

**To create a tabulated surface**

After you select the entity for the path curve, you specify the direction vector:

> **TABSURF**

Select path curve:

Select direction vector (line or open polyline):

The direction and length of the line determine the direction and length of the extrusion. Only the first and last points of a polyline are used when determining the extrusion path. The point at that you specify the entity determines the extrusion direction.

Please check in the Learning Center for a fully documented example of this command's use.

**Attach an External Drawing File to the Current Drawing**

## **Command: XINSERT**

**Alias XI**

### **Located:**

#### **1. Entered Directly at the Command Line**

The command XINSERT is processed similar to the XLINK command applied with the *Attach* option. The command is provided for convenience, allowing you to directly attach an external drawing to the current drawing.

---

### **See also:**

XLINK

## **Externally Referenced Drawings**



**Command: XLINK**

**Alias XL**

### **Located:**

- 1. Pull-down Menu: Parts – External References**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Externally Referenced Drawings (XREF's) serve especially to

4. Create assembly drawings from different files
5. Insert detail drawings into a finished one
6. Temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is realized within a running project where a workgroup uses XREF's to create several drawings and details in a network.

Another advantage is that within the current drawing, only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file. This keeps the file size small and as such it is easier to work with. In this way, very complex drawings can be created from a group of relatively small files. In these instances, XREF's have an advantage over the use of blocks.

### **XLINK command: Control externally referenced drawings**

The XLINK command allows you to create a link to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

#### **Differences to Part Insertions**

Although the treatment of linked drawings is basically similar to those of part insertions (block insertions), some fundamental differences include:

An Xref establishes a link to another drawing file, but it does not become a permanent part of the current drawing.

#### **Dependent Symbols**

When attaching an external file, those symbol tables are loaded into current drawing that are required to display the referenced coincident to its original: layers, linetypes, text styles, dimension styles, and block definitions.

The program utilizes the following scheme to name dependent symbols in the current drawing:

Logical\_Xref\_Name| Dependent\_Symbol\_Name

For example, the layer names of an externally referenced drawing attached with the logical name SHAFT are listed as:

SHAFT|CONSTRUCTION

SHAFT|DETAILS

SHAFT|DIMENSION

Other dependent symbol table entries like linetypes or blocks will carry the same naming convention.

Note that you cannot redefine or rename dependent symbols. Also, dependent blocks cannot be inserted and a dependent layer cannot be made the current layer.

But the dependent symbol conventions allow you to control the visibility, the colors, and the linetypes of externally referenced drawings. The layer dialog box displays the names of the dependent layers and you can apply the options On/Off, Thaw/Freeze, Linetype, and Color on them.

The logical name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

To keep the visibility and display settings over subsequent sessions, make sure to set the system variable **VISRETAIN** to 1. By default VISRETAIN is set to a value of 0 that means that visibility and display modifications made to the dependent layers are not maintained when you leave the current drawing.

### **The Dialog Box *Externally Referenced Drawings***

The dialog box displays a list box and a column of option buttons.

The list box displays the following information on externally referenced drawing files that have been inserted into the current drawing

1. The logical name assigned to it when the file was attached, and
2. The path and filename of the referenced drawing

The options available to manage externally referenced drawings are provided in a column of buttons:

<b>Attach</b>	Insert an external drawing file into the drawing as block
<b>Reload</b>	Update externally referenced drawing to display its current state
<b>Path</b>	Relocate/redirect or alter source of the referenced file
<b>Bind</b>	Transform an externally referenced drawing to an ordinary block in the drawing
<b>Detach</b>	Delete an externally referenced drawing from the current drawing and erase the dependent symbols from the drawing database tables

These options are described in detail in the following sections.

#### **Attach**

The *Attach* option allows you to insert an external drawing as a block to the current drawing.

#### **To attach (set up a link and insert the contents) to an external drawing file:**

1. The standard file dialog box allows you to specify the drawing you want to attach to the current drawing.
2. Within the dialog box *Attach External Drawing*, a unique Logical Name (block name) needs to be determined for the associated referenced file. The program suggests using the name of the file. However, in many cases it is recommended to use a short logical name, because the logical name will be used as prefix for the names of dependant symbol names for layers, linetypes, etc.
3. The other options of the dialog box let you specify preferences for the insertion and are identical



to those found in the dialog box of the INSERT command (insertion point, scale factor, and rotation angle), as described previously.

The drawing attached is based on the most recently saved version of the drawing file.

### **Reload**

The option *Reload* allows you to update an external reference any time during a current drawing session. Any Reload reflects any subsequent modification made to the external file.

Note that when you open a drawing that contains externally referenced drawings, these are displayed automatically in their current state. When working on a project in a workgroup over a network, it might be necessary to update one or more external references while you are working on a current drawing. The program will reload the specified drawing(s) in the shape as they have been saved most recently.

#### **To reload externally referenced drawings in a current drawing session,**

1. select the external reference to be updated from the list box
2. click the *Reload* button

### **Path (Relocate or alternate source of the referenced file)**

Occasionally the storage location of an externally referenced drawing might have been changed or the referenced file has been renamed. Also, it might happen that, when sending the project files including referenced drawings to someone else, the path to the referenced files needs to be updated.

In such a situation the option *Path* of the dialog box allows you to re-establish the path to the referenced drawing, to reload a renamed referenced file, or to replace a referenced drawing by another file.

#### **To edit or relocate the path to a previously linked reference drawing,**

1. select the external reference to be relocated from the list box
2. click the *Path* button

From now on, the link to the referenced file will be maintained as specified and the data of the referenced file will be updated from that source.

### **Bind**

In certain situations it is useful to make an externally referenced drawing a permanent part of a drawing, for example when you need to send the drawing to someone else or if it has been finalized and is going to be archived. Binding the referenced drawing to the current drawing has the advantage that there is no need to gather an assembly of drawings within an archive or mail.

The option *Bind* of the XLINK command allows you to transform an externally referenced drawing to an ordinary block in the drawing. Once you bind an externally referenced drawing, the link to it is broken and you will no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definition, text styles, and dimension styles into the current drawing. Within each corresponding dependent symbol name, the vertical bar symbol (|) is replaced by a new three character symbol \$n\$, where n is a number that is usually 0. For example, the layer name HOUSE|BASEMENT will be altered to HOUSE\$0\$BASEMENT.

#### **To bind one or more external references to the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Bind* button

### **Detach**

Externally referenced drawings that are not needed any longer in the current drawing can be removed from the base drawing with the *Detach* option. This will remove all of its dependent layer, linetype, and block references as well.

Note: Although you can erase an externally referenced drawing with the DELETE command, it is recommended to use the Detach option of the XLINK command, as this operation will also remove the Block definition from the drawing database.

**To remove one or more external references from the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Detach* button

**Open**

As the Multiple Document Interface allows you to display up to four drawings on the desktop, the option *Open* of the dialog box allows you to open an externally referenced drawing simultaneously on the CAD desktop. This allows you to modify a referenced drawing, for example, if you have detected discrepancies or inconsistencies in the linked drawing that needs to be fixed.

**To open a referenced drawing in another drawing window (viewport),**

1. select the external reference you want to open from the list box
2. click to the *Open* button

Once you have modified a referenced drawing, save the drawing (and, if the modification has been finished, close it). To display the revised external reference in the master drawing, call the command XLINK and reload the corresponding file (see above: *Reload*).

**Insert**

The option *Insert* allows you to insert another copy of the Xref chosen from the list box into the current drawing. You may then define the insertion point, the scale factor and the rotation angle for that additional instance of the external drawing in the current drawing.

You can move, scale, rotate, and copy externally referenced drawings. But, you cannot explode an external reference.

**Note:** You should not remove an Xref from the current drawing with the DELETE command, because it does not purge the Xref block definition from the drawing database and the associated symbols will be maintained. To remove an external reference correctly from the current drawing, it is recommended you use the Detach option of the XLINK command.

The Insert option, however, can serve to reinsert an external reference that accidentally has been deleted.

---

**See also:**

XINSERT

**Object Selection (Selection Sets)**

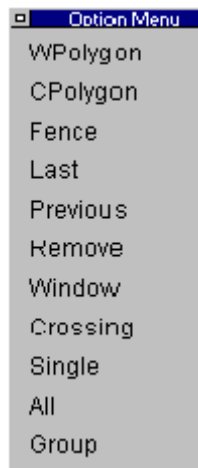
**Object Selection**

Commands often need objects to act upon, so the selection of these objects (called creating a selection set), is a natural part of command input. Usually, you issue the command and then you select the objects that you wish the command to work upon. Later, you indicate which command options you wish to apply. This is the default command input order and it is controlled by the PICKFIRST variable. If it seems more natural for you to select command options before selecting objects, then you can reset the value of this variable and reverse the order, accordingly.

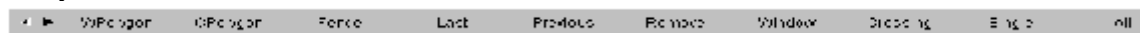


```
> *** Cancel ***
> move
Select objects:
```

When you issue the command, the system will display all available object selection options in the popup Option Menu or horizontal Option Bar (depending on your configuration settings; see Desktop Configuration).



If you select Option Menu in Desktop configuration, you will get an option menu placed on the desktop when you issue commands, such as the one above.



If you select to have an Option Bar instead of menu, a toolbar located between the menu and properties bar will display command options as above.

Selection options remain active until you press the Enter key or click the right mouse button indicating you have selected all the elements desired for editing. As elements are selected, they are highlighted on screen. Magenta is the default color for highlighting in the system; this can be changed under the Colors tab of the Configuration (*config* command) menu.

You select as many elements as you want for modification. This group of entities that you select is referred to as your "selection set".

Once a Selection Set is established, the current command (e.g. copy, move, etc.) will be executed on the complete Selection Set.

A proper understanding of Selection Sets is important to efficient CAD editing. Aside from experimentation, you will greatly enhance your productivity by understanding each option that is available to you, including Grouping of Objects.

#### **See Also**

- Object Selection Option - Wpolygon
- Object Selection Option - Cpolygon
- Object Selection Option - Fence
- Object Selection Option - Last
- Object Selection Option - Previous
- Object Selection Option - Remove/Add
- Object Selection Option - Window
- Object Selection Option - Crossing
- Object Selection Option - Single
- Object Selection Option - All
- Object Selection Option - Group

## **Object Selection Modes**

In order to select objects the user can employ several modes of OBJECT SELECT. Two standard modes are at your disposal immediately. Other modes are available in the option bar.

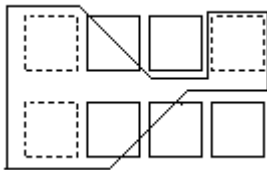
The modes can be called by:

- ▶ Selecting the proper switch;
- ▶ Inputting of the name of the object-selecting mode;
- ▶ Inputting the abbreviation for the name of the object-selecting mode.

The abbreviations to choose an object selection mode are as follows:

Window	w
Crossing	c
WPolygon	wp
CPolygon	cp
Fence	f
Last	l
Previous	p
Add	+
Remove	-
Single	si
All	*
Group	g

### Object Selection Option - WPOLYGON



#### **WPolygon**

The mode Window Polygon functions in principle by the same proceeding as the option "Window". This means, that all drawing entities are marked whose contours are entirely inside of the selection window. The difference between the two modes is in the form of the selecting window. The mode Window Polygon allows the user to design an irregular window with as many corner points as are desired. This allows more flexibility in selecting that does "Window".

To use WPolygon, first define in succession the corner points for the selecting window. The "preview" mode (from the option bar) will offer a preview of the emerging polygon.

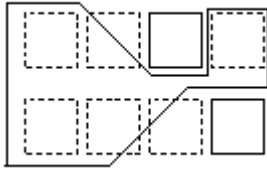
The corner points for the selecting window can be defined in a clockwise or counter-clockwise series. The program allows the polygon lines to cross or intersect themselves and to shape "loops".

ENTER will conclude the definition of the selecting window. All objects that lie totally inside of the window will be marked.

#### **See Also:**

Object Selection Options - Summary  
Object Selection Option - CPolygon  
Object Selection Option - Fence  
Object Selection Option - Last  
Object Selection Option - Previous  
Object Selection Option - Remove/Add  
Object Selection Option - Window  
Object Selection Option - Crossing  
Object Selection Option - Single  
Object Selection Option - All  
Object Selection Option - Group

## Object Selection Options - Cpolygon



### **CPolygon**

By the selecting mode Crossing Polygon there can also be created a irregularly formed window with as many corner points as you like. All objects that lie inside of the selecting window or are crossed by the window frame will be marked.

Define successively (clockwise or counterclockwise) the corner points for the selecting window. The preview mode will offer a preview of the emerging polygon.

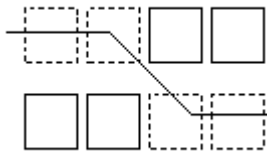
The program also allows the polygon lines to cross or intersect themselves and shape "loops".

ENTER will conclude the definition of the selecting window. All objects that lie totally inside of the window or are crossed by it will be marked.

### **See Also:**

- Object Selection Options - Summary
- Object Selection Option - Wpolygon
- Object Selection Option - Fence
- Object Selection Option - Last
- Object Selection Option - Previous
- Object Selection Option - Remove/Add
- Object Selection Option - Window
- Object Selection Option - Crossing
- Object Selection Option - Single
- Object Selection Option - All
- Object Selection Option - Group

## Object Selection Options - Fence



### **Fence**

The Object Selection option Fence represents an effective variant of a crossing selection mode. The object to be selected will not be closed or crossed by a window, but only "cut through" by a line, or "fence".

The line of the fence can be defined by as many points as the user desires, and the fence may even intersect or cross itself.

ENTER will conclude the definition of the fence line. All objects that are touched or crossed by the line will be marked and added to the selecting set.

### **See Also:**

- Object Selection Options - Summary

- Object Selection Option - Wpolygon
- Object Selection Option - CPolygon
- Object Selection Option - Last
- Object Selection Option - Previous
- Object Selection Option - Remove/Add
- Object Selection Option - Window
- Object Selection Option - Crossing
- Object Selection Option - Single
- Object Selection Option - All
- Object Selection Option - Group

## Object Selection Options - Last

### **Last**

The Object Selection option Last marks the last drawn entity and adds it to the selecting set.

It does not have a memory function, that is, it is not possible to add to the selecting set the entity created during subsequent "last" operations by repeated calling of the option.

*Note:* If the last drawn entity lies on a "frozen" layer you will receive the message in the command line area that no entity was selected.

### **See Also:**

- Object Selection Options - Summary
- Object Selection Option - Wpolygon
- Object Selection Option - CPolygon
- Object Selection Option - Fence
- Object Selection Option - Previous
- Object Selection Option - Remove/Add
- Object Selection Option - Window
- Object Selection Option - Crossing
- Object Selection Option - Single
- Object Selection Option - All
- Object Selection Option - Group

## Object Selection Options - Previous

### **Previous**

The Object Selection option Previous (P) restores the previous selecting set. All entities are marked (and highlighted), that have been selected by the previous selecting operation.

*Note:* This mode has the same restrictions as the modes "Last" and "All", in that entities that are located on a frozen layer will not be marked, entities on a locked layer will be marked but cannot be handled.

### **See Also:**

- Object Selection Options - Summary
- Object Selection Option - Wpolygon
- Object Selection Option - CPolygon
- Object Selection Option - Fence
- Object Selection Option - Last
- Object Selection Option - Remove/Add
- Object Selection Option - Window
- Object Selection Option - Crossing
- Object Selection Option - Single
- Object Selection Option - All
- Object Selection Option - Group

## Object Selection Option - Remove/Add

### Remove Mode / Add Mode

The last field of the option bar contains a toggle button to switch between *adding objects* to the current selecting set and *removing objects* from the selecting set.

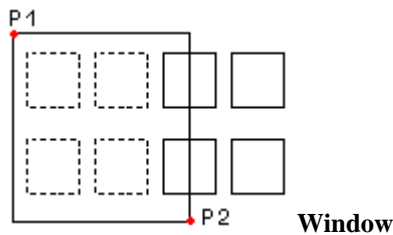
The mode *Add Objects* is the default active mode, so the user can select further objects and add them to the selecting set at any time.

By selecting the button Remove in the option bar, the user changes the selection mode to *Remove Objects*. Objects already selected and marked will be removed from the selection set by this change.

### See Also:

- Object Selection Options - Summary
- Object Selection Option - Wpolygon
- Object Selection Option - CPolygon
- Object Selection Option - Fence
- Object Selection Option - Last
- Object Selection Option - Previous
- Object Selection Option - Window
- Object Selection Option - Crossing
- Object Selection Option - Single
- Object Selection Option - All
- Object Selection Option - Group

## Object Selection Option - Window



A (rectangular) selecting window is defined by two diagonal opposite corner points. Position the cursor on the first corner of the selecting window and select that point.

The user must take care that no object is inside the cursor window (box) during the fixing of the first corner point. Otherwise, the option "Crossing" will be active instead of the option selecting window and the object that is situated in the cursor box will be marked.

Determine the second corner point of the selecting window. This will select and mark all of the entities that are entirely within the window frame described.

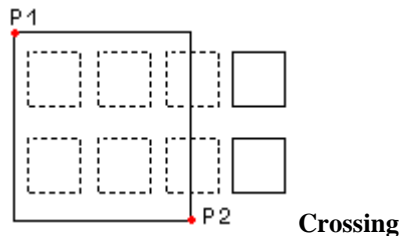
The command line will again display the prompt to select objects. Continue the object selection operation with the same or another selection mode, or terminate it by pressing ENTER.

### See Also:

- Object Selection Options - Summary
- Object Selection Option - Wpolygon
- Object Selection Option - CPolygon
- Object Selection Option - Fence
- Object Selection Option - Last
- Object Selection Option - Previous
- Object Selection Option - Remove/Add

Object Selection Option - Crossing  
Object Selection Option - Single  
Object Selection Option - All  
Object Selection Option - Group

### Object Selection Option - Crossing



When selecting an object by Crossing, only entities whose contours touch, or "cross", the window frame will be chosen.

Position the cursor so that its window crosses the contour of the desired drawing entity and select to mark the object.

An object will not be marked if the cursor is only situated on the area that is *included* by the contours of the entity, without touching the contour itself. In this case, the program will activate the selection mode "Window" and interpret the input as the first corner point of the window.

#### See Also:

Object Selection Options - Summary  
Object Selection Option - Wpolygon  
Object Selection Option - CPolygon  
Object Selection Option - Fence  
Object Selection Option - Last  
Object Selection Option - Previous  
Object Selection Option - Remove/Add  
Object Selection Option - Window  
Object Selection Option - Single  
Object Selection Option - All  
Object Selection Option - Group

### Object Selection Option - Single

#### Single Selection Mode

The option Single (called by the abbreviation SI) de-activates all Object Select modes for the next and subsequent operations, with the exception of the "standard" modes WINDOW and CROSSING.

Note: This option is not offered in the option bar, but may to be called by keyboard input.

#### See Also:

Object Selection Options - Summary  
Object Selection Option - Wpolygon  
Object Selection Option - CPolygon  
Object Selection Option - Fence  
Object Selection Option - Last  
Object Selection Option - Previous



Object Selection Option - Remove/Add  
Object Selection Option - Window  
Object Selection Option - Crossing  
Object Selection Option - All  
Object Selection Option - Group

## Object Selection Option - All

### **All**

The Object Selection option All, that is called by the input of ALL or the abbreviation A marks all entities included in the drawing with the exception of those located on frozen layers and adds them to the selecting set.

### **See Also:**

Object Selection Options - Summary  
Object Selection Option - Wpolygon  
Object Selection Option - CPolygon  
Object Selection Option - Fence  
Object Selection Option - Last  
Object Selection Option - Previous  
Object Selection Option - Remove/Add  
Object Selection Option - Window  
Object Selection Option - Crossing  
Object Selection Option - Single  
Object Selection Option - Group

## Object Selection Option – Group

### **Group**

The Object Selection option Group, that is called by the input of GROUP or the abbreviation G prompts you for the names of Groups of objects that have been previously stored by the command GROUP. All objects belonging to the named groups are then added to the selection set being assembled for the related command.

### **See Also:**

Object Selection Options - Summary  
Object Selection Option - Wpolygon  
Object Selection Option - CPolygon  
Object Selection Option - Fence  
Object Selection Option - Last  
Object Selection Option - Previous  
Object Selection Option - Remove/Add  
Object Selection Option - Window  
Object Selection Option - Crossing  
Object Selection Option - Single  
Object Selection Option - All

## Tips For Object Selection

When selecting multiple entities that lie in the same region of the drawing, by far the most common selection options used are Window and Crossing. Both of these options rely on a rectangular area that is picked onscreen and that define the group of objects being selected.

This rectangle is defined by picking one corner onscreen, and then picking the diagonally opposite corner.

By convention, users select a Window by picking the left corner of the rectangle and secondly the right corner of the window, whereas with a Crossing the pick order is the reverse (right corner then left). AllenCad is set up to recognize two picked points as a selection rectangle for a Window or Crossing, just by the pick order. This means that you do not have to enter a Window or Crossing option at all – you just pick your rectangle in the correct order.

Note that this shortcut extends only to Window and Crossing selection options. Window Polygon and Crossing Polygon must be specifically selected as options in order to work properly. It is recommended, however, that you maintain the left-right Window pick order and the right-left order for Crossing designations for these commands as well, as this will become an intuitive choice and assist you in developing good operating procedures for the other two commands.

## Coordinate Filter...

ALLENCAD offers the possibility to use Coordinate Filters when point values are requested. Coordinate Filters allow the user to relate the point value inputs to the X-, Y and Z-coordinates of existing entities. Some possible inputs that might be requested for point values are:

x, <b>value</b>	x, y
<b>value</b> , y	x, y, z
value, value, z	<b>value</b> , y, z
x, <b>value</b> , z	x, y, <b>value</b>

If the user answers the question with the input: x,10 the user will be requested to enter the x-value separately. It is now possible to catch a second object or the geometrically significant point of an object by using an Object Snap function. The X-value of this point will be interpreted as input value for the original point definition (x,10).

> LINE

From point: P1

To point: x,10

X-coordinate: end

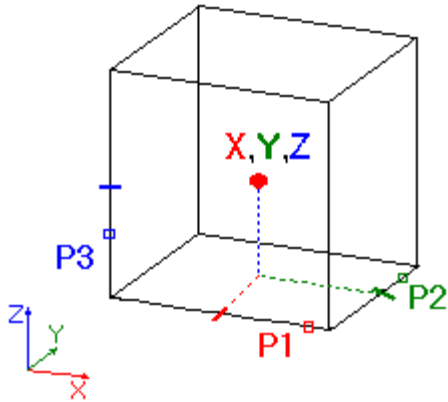
from: P2

To point: 15,**y**

Y-coordinate: cen

from: P3

Example: Identifying the center of a cube



> POINT  
 Point: X,Y,Z  
 X-Coordinate: MID  
 of object: P1  
 Y-Coordinate: MID  
 of object: P2  
 Z-Coordinate: MID  
 of object: P3

### Grouping Entities



**Command: GROUP**

**Alias G**

#### Located:

1. **Pull-down Menu: Parts – Group**
2. **Palette: Parts**
3. **Palette Edit**
4. **Or Enter Command or Command Alias Directly at the Command Line**

Drawing entities can be combined in named Groups. Essentially, groups are a named selection set, and as such, the user is able to manipulate all entities within the Group in a single operation (Moving, Rotating, Mirroring, or Scaling, for example) just by specifying the group name. Single entities of a Group may still be *edited* individually (for example, intersecting or stretching). Entities may be removed or added to a Group at any time. An object may be a member of more than *one* Group. *Groups* may be contained in *other* Groups. Group definitions are stored in a distinct table of the drawing database. Groups are maintained in this fashion from one drawing session to the other.

#### Differences Between ‘Parts’ and ‘Groups’

A Part (a block) has its own insertion point. A Group has none. A Part can occur multiple times within the drawing. If a block is redefined, all of its occurrences are updated. Groups are not treated like that. Groups can, however, be copied. A Group that originates from another Group that has been copied is a separate union of entities. Each Group has a unique name.

**Note:** If an entity that is member of a Group is included in a part definition, the entity is deleted from the drawing and removed from the Group.

#### The Command GROUP

To group entities, or to manage and manipulate Groups, use the command GROUP. The options within the command are:

<b>Create</b>	Brings entities into a newly named Group
<b>Explode</b>	Abolishes a Group definition
<b>Rename</b>	Alters the name of an existing Group, especially of anonymous groups
<b>Add</b>	Adds drawing objects to an existing Group
<b>Remove</b>	Removes drawing objects from a Group
<b>Selectable</b>	Determines whether a Group is selectable or not

### Creating Groups

To create a group, first determine a unique name. The name will be required to identify the group later for a number of command options. In addition, specify whether the group should be selectable or not (see below).

#### To define a new group,

1. Call the command GROUP with option Create,
2. Specify a unique group name (max. 31 characters),
3. If desired, enter a description (max. 64 characters),
4. Determine whether the group should be selectable or not (the default is Yes), and
5. Select the drawing objects that should belong to the named group.

To create a group using Command Line:

```
> GROUP
Select option: Create
Group name: G1
Description: Screw
Should the group be selectable (Yes/No)? Yes
Select objects: Select the entities that should belong to the group
```

### Exploding Groups

Group definitions may be abolished by using option **Explode** of the GROUP command. When a group definition is dissolved the corresponding entry in the group table is deleted in the drawing database.

The only way to abolish a group definition is by abolishing its name.

To explode a group,

1. Choose the command GROUP with its option *Explode*,
2. At the prompt "Group name: " enter the name of the group, to be exploded.

Because a single entity may belong to multiple groups and groups may be part of other groups, the **Explode** option does not allow the user to pick an entity within a group to explode the group.

### Renaming Groups

The name of a group may be changed by using the GROUP command **Rename**. It is useful to employ this method when "anonymous" groups that have been created by copy commands exist. The system will automatically apply anonymous names to copies group (\*\*A1, \*\*A2, etc.) that exist through the use of one of the commands like COPY, MIRROR, or ARRAY.

To rename a group,

1. Call the command GROUP with its option Rename;
2. At the prompt "Current group name: " enter the name of the group (for example "A0");
3. At the prompt "New group name: " enter the name of the specified group (for example "Screw\_Copy1")

### Add / Remove Entities

Two other options of the command GROUP are provided to allow the user to integrate additional entities into an existing group or to remove entities that are currently in the group.

After a group name is specified, the entities that belong to the group are highlighted.

► To add entities to an existing group, simply 'pick' the entities that are to be added (they will *not* be highlighted).

► To remove entities from an existing group, 'pick' the highlighted entities that are to be removed

**Note:** Should *all* entities be deleted from a group, remember that *the group will still exist as long as it has a name!*

### Working with Groups

Editing commands will treat all entities in a group as a single object. Commands like MOVE, ROTATE, or COPY will act on the entire group.

If the user should select an object for one of these commands at the prompt that selection will lead to the automatic selection of all group entities.

A typical prompt of a command like this would be:

Select objects:

The rule is: In commands that allow **multiple selection**, identifying a *single* entity of a group is a selection of *all* entities belonging to the group(s). When the user is *prompted* to select multiple objects, he or she may choose the object selection option *Group* from the options bar, and enter a group name to select all of the entities belonging to the group. Groups, therefore, can also be named selection sets.

Other MODIFY commands (Offset, Lengthen, Intersect, Trim, etc.) request a single entity in one or more steps (usually to modify the geometry of drawing entities). A typical prompt for a command like this would be:

Select entity (Line, Circle, Arc, 2D-Polyline):

In this case **single selection** is being used. Any time the user is prompted to select a single entity, the group does not matter. These commands modify the geometry of single selected entities even though those entities belong to a group. The modified entities will remain members of the group. This feature makes allows great flexibility in working with GROUPS and provides significant advantages for the user compared to using BLOCKS.

The following table summarizes the editing commands and describes how entities that are member of a group will be processed when using each command.

#### DELETE

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. When a single entity of a group is deleted, the entity is removed from the group definition.

#### MOVE, ROTATE, SCALE, FLIP

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used.

#### MIRROR, COPY, ARRAY

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. The system will apply anonymous names to copied groups ("\*A1", "\*A2", etc.) that are created by the use of these commands.

#### STRETCH

Those entities belonging to a group that have been selected using the object selection modes *Crossing* or *Cpolygon* will be stretched.

#### TRIM, EXPAND

The prompt to "Select cutting edges" or "...boundary edges" will include all entities of a group, unless the Object Selection Mode *Single* has been used.

#### OFFSET, FILLET, CHAMFER, LENGTHEN, INTERSECT

These commands always prompt the user to select a *single* entity to be modified. That an entity belongs to a group does not matter in these operations. Single entities altered by these commands will remain in the group. If *new* entities *originate from these commands* they will *not* be automatically included in the group.

#### DELPARTIAL, REJOIN

When an entity (line, circle, arc or 2D-Polyline) is broken into two or more entities by partial deletion, only one of the entities will remain in the group. When an entity that has been broken into two or more pieces is rejoined, the newly created entity is no longer member of any group, even if one or more of the pieces was part of a group.

#### PARTDEF, HATCH, HATCHEDIT

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. This also applies to the command HATCHEDIT when the user is prompted to select additional objects: to be included to the set of entities for associative hatch editing.

#### TEXTEDIT

The text entity retains its association to its group if the command TEXTEDIT is used to modified text.

## External References, Parts and Attributes

### Using Parts

Parts allow the user to create a drawing using existing entities. These entities, defined as Parts, may contain single objects, groups of objects, partial drawings or complete drawings, among other possibilities. The principle feature of these entities is that they are inserted into a drawing as a "whole" entity, and not a group of single entities.

### Properties of Parts

A part is a complex object that consists of several drawing entities or objects. Examples of those objects can be:

Basic entities or objects, such as lines, circles, arcs, etc.;

Drawing entities or objects with hatches, dimensioning or details;

Complete drawings; and

Drawings already defined as Parts.

After an entity or group of entities is selected and defined as a Part, it is saved under a Part name and treated as one single entity for future operations.

Editing, Changing, and Deleting always will always effect the defined part as a whole. A special command allows a part to be dismantled (exploded) into its original components.

A part can be inserted in an existing drawing by selecting the command INSERT. This command will facilitate changes in the scale and the rotation angle so that the part may be "fitted" to the current drawing. When the part is inserted into the drawing, all of its defined properties (layer, color, line type) will be inserted with it.

The part will be inserted on a layer, as it was defined. If the part contains another layer (for instance, if the part is a group of parts, each on its same layer) that does not exist in the current drawing, that (those) layer(s) will be created in the current drawing during the insertion of the part. The exception to this rule is entities of a part that are situated on Layer 0. These entities will automatically be inserted on the current layer.

## Internal and External Parts

Internal and external parts are treated differently with regard to storage location and with regard to relationship with the current drawing.

Internal Parts are parts that have been created and stored in the current drawing. They are *internal* to the current drawing. They may be reused in the current drawing, but are not available for use in other drawings without an additional operation.

External Parts, on the other hand, are created and stored as separate drawing files. They may be used independent of the current drawing. It is good management to "group" these parts with functionally similar parts in a library, and name that library so that it may be used repeatedly.

By building these libraries, the user will facilitate her or his drawing tasks greatly.

## Attributes

A part can be given Attributes. Whereas the drawing of the part may adequately describe its physical outline or dimensions, Attributes are used to describe other physical characteristics. A piece of lumber 2 inches by 4 inches by 8 feet, for instance, may be of redwood, or fir, or a hardwood. This characteristic cannot be shown by the physical outline of the piece of lumber, but it can be described as an Attribute, and, if the piece of lumber is in a library as a part, that attribute can be carried with it into any drawing the user desires.

### See also:

Defining Internal Parts

[External Parts](#)

Inserting Parts

Exploding Parts

Part/Symbol Library

Part / Symbol Library...



**Command: PARTLIB**

**Alias LIB**

### Located:

#### 1. Enter Command or Alias Directly at the Command Line

The command PARTLIB in the menu *Parts* allows the user to manage drawing files such as project drawings, template drawings, part files, or symbol files.

A library is defined by a parameter file (of file type **.plb**) and a set of drawing files, assembled in a directory where the library parameter file is located.

The command PARTLIB called with the name of a parameter file (for example, c:\AllenCad\drawing.plb) will display a dialog that allows you to select a drawing to be inserted. The advantage of the command is that it displays all drawings with a preview bitmap, with verbal information in a list box.

Within the dialog box the user will see the preview of the drawings and information about the drawing.

There are two modes for the list box:

**1.Info Mode** the drawings are displayed together with the verbal information.

2. In **Quick Mode** the drawings are displayed only with their preview bitmap.

The other buttons of the dialog have the following functions:

- **Close:** Closes a dialog without further action
- **File Info:** Opens a dialog to display and to change the file information.
- **Open:** This opens the selected file as separate drawing.
- **Insert:** This button lets the user insert the drawing into your current drawing.

---

#### See also:

Setup and Manage Part Libraries

This section describes how to create PLB files to customize the PARTLIB command to personal needs.

#### Command SETLIB



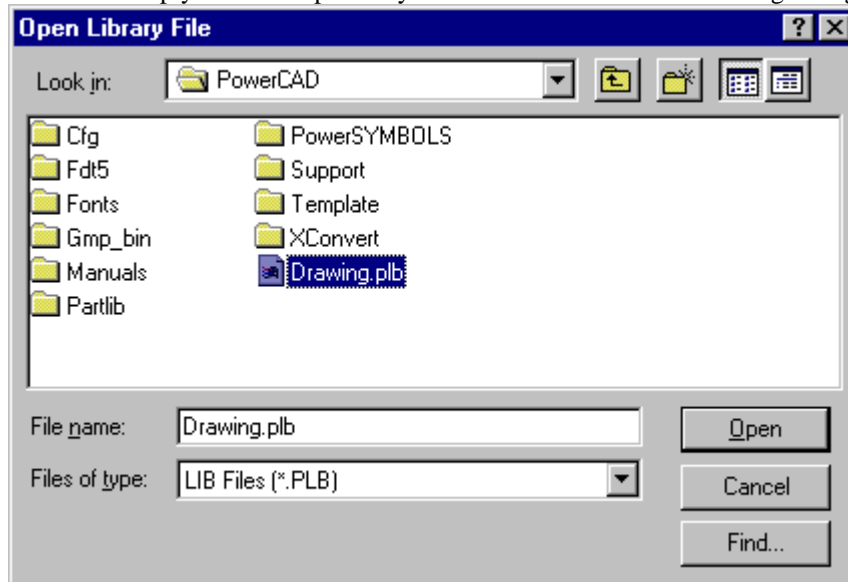
**Command: SETLIB**  
**Alias STB**

#### Located:

1. **Pull-down Menu:** Tools – Resource Manager – Set Default Part Library  
Also - Parts – Set Current Library
2. **Palette:** Parts
3. **Or Enter Command or Alias at the Command Line**

The LIBRARY command displays the Set Current Part Library / Symbol Library file dialog box. Select a part library (.plb) file.

The **SETLIB** command, on the other hand, sets the default library to be loaded whenever LIBRARY is executed. Simply select the .plb file you want as default in the following dialog:



Alternately, enter the PARTLIB command, which prompts you for the name of a part library at the command prompt. This command is meant for use with macros and FLISP functions. Enter the name of a .plb file, such as:

```
> partlib  
File name: c:\AllenCad\partlib\partlibdrawing.plb
```



Once a part library has been chosen, the Library command continues to use that same library as default. Use the **SETLIB** (Part > Set Current Library) to change the part library.

### **Insert Part ...**



**Command: INSERT**

**Alias I**

#### **Located:**

1. **Pull-down Menu: Parts – Insert Part**
2. **Palettes: Draw and Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

**Command: QINSERT**

**Alias QI**

#### **Located:**

1. **Pull-down Menu: Parts – Quick Insert Part**
2. **Or Enter Command or Alias Directly at the Command Line**

The command INSERT offers broad support for the user in its Dialog Box. The command QINSERT is the command line version of the command. The command macro QINSERT ? allows fast insertion of internal parts (parts already defined in the drawing).

The command INSERT provides selection areas for:

- Selection of external parts;
- Definition of the insertion point, scale factors, and rotation angle, with future choices of using coordinate/ numeric input or cursor selection;
- Parameter values may be reset; and
- The Part may be "exploded" into its components.

### **Selecting a Part**

To select the Part to be inserted, type the name of the Part (or select an existing Part) in the field Part Name. Use the buttons External and Internal to select the correct Part listing.

### **Defining the Insertion Point**

Defining of the Insertion Point either by entering the proper coordinate values, or by selecting Interactive, and "picking" the point with the cursor.

### **Scale Factor**

The Scale Factor input areas will offer 1.0 as the default XYZ scale factor. This will insert a part at its original size. Remember that a scale factor greater than 1.0 will increase the size of the part, and a factor less than 1.0 will reduce the size of the part to the percentage of the original size specified by the Factor. A factor of 2 will result in the part being twice its original size, while a factor of 0.5 will be half

original size.

By default, AllenCad prompts you for x and y scale factors only. If you need to specify a z scale factor, simply respond to the X scale factor prompt with an xyz response. You will then be prompted for all three scale factors in turn.

The option *Interactive* allows the user to define the size of the Part during the insertion as well. To insert the Part using the Interactive option, simply define by opposite corners the rectangle into which the Part is to fit.

Since dragmode is active during this operation, the user will have the benefit of seeing the selection results before confirming the selection.

### **Rotation Angle**

The Rotation Angle can be set from the Dialog Box. Enter a value for the Rotation Angle in the proper input window. For counter-clockwise rotation enter a positive value, for clockwise rotation enter a negative value between 0 and 360 degrees

### **Default**

Default allows the user to reset the parameters for the Insertion Point, the Scaling Factor and the Rotation Angle back to the original parameter values.

### **Exploded**

Exploded allows the user to "explode" the part into its component entities. Exploding will cause the loss of the characteristics that were retained as a part.

### **Troubleshooting**

If you find that the blocks you are inserting are not placed in the specified location, check the settings for INSBASE and ELEVATION in both drawing files. These should be set to 0,0,0 and 0 respectively, if you do not want them to affect your block insertion.

## **Write Part File ...**



**Command: PARTEXP**

### **Alias PX**

### **Located:**

1. **Pull-down Menu: Parts – Write Part File**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

**Write Part File...** works identically to the function **Part Definition...** (Command PARTDEF) but is used to export parts as external drawing files.

In practice, it is used to create new drawing files (.flx, .dwg or .dxf) consisting of only selected drawing entities, regardless of their intended use. It is similar to the SAVEAS command, except that Saveas will include all current drawing elements in the new file.

Both PARTDEF and PARTEXP functions create part definitions within the current drawing. The proposed part name, the part insertion point, an internal part definition and/or selected drawing entities are input during the export process.

Define Parts...



**Command: PARTDEF**

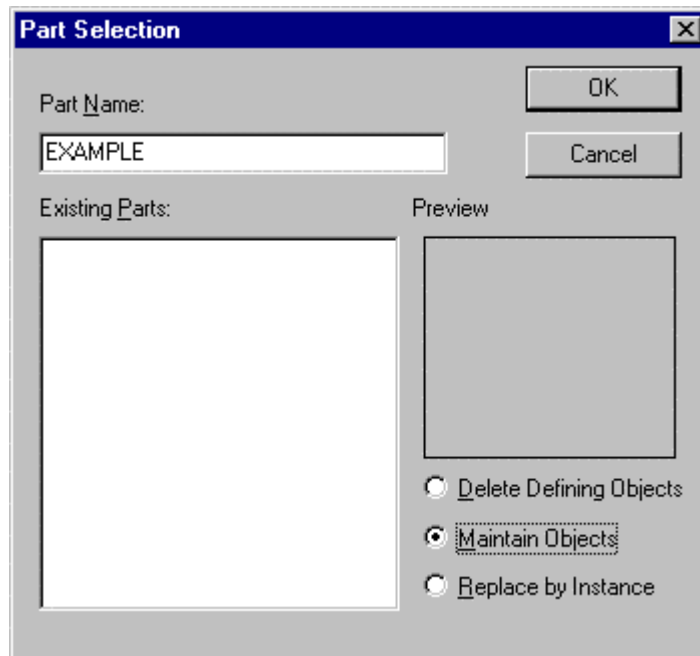
## Alias PDF

### Located:

1. **Pull-down Menu: Parts – Define Part**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

The PARTDEF command defines a new internal part. Select the function **Define Part...** from the menu *Part*, or enter the command PARTDEF via the keyboard. Once defined, the internal part definition will allow for multiple part insertions *within the current drawing*.

When issuing the command, you are presented with the following dialog:



### Part Name

Names of parts can contain up to 31 characters. In addition to letters and numbers the special characters \$, -, and \_ are allowed. The box is not case-sensitive.

Enter the name of the part in the input field of the Dialog Box. Existing Parts are listed in the selection window below. If one of those parts is required, select the part from the Selection Window using the cursor.

If there are parts of the same name already in the selection box, a message will appear so stating. The message will give the user the option of choosing a new name, or of re-defining the existing part.

The user may also, in this Dialog Box, elect to delete the part in the original drawing. The appropriate box should be checked to exercise this function.

After this choice has been made, verify the selections of the Dialog Box by OK. The Box will close, and the part will have been created.

### Existing Parts

If there are any existing part definitions in the drawing, they will appear in the window on the left side of the dialog. They can be selected here for inclusion in the new part definition, if required. The **Preview Window** to the right will show a thumbnail picture of the part selected in the window on the left.

## Other Dialog Options

The remaining three options determine what action the program will take with the objects being included in the part definition.

- **Delete Defining Objects** – deletes the objects after including them in the part definition
- **Maintain Objects** – Leaves them in the drawing as individual objects as well as including them in the definition.
- **Replace by Instance** – Substitutes the new Part in place of the individual defining objects.

This completes dialog input, where pressing OK moves you on to the final command line entries that complete the PARTDEF command.

You are next asked to define: **Insertion point**

Each part must have a defined insertion point.

The **insertion point** is a reference point that enables the user to position the part properly during insertion. The insertion point of a part is also the base point for changing the scale, or for use as a rotation point during the insertion.

Any point on - or even off of - the part may be selected as the insertion point. Typically, users will select the left lower edge, the center of an object, or some other discreet point of the geometry as the insertion point. It is a good idea to use the Object Snap options for this selection, as it insures precision.

## Object selection

Next, the user must select the object or objects that will make up this part. These will join any existing part definitions selected in the dialog, in the new definition. To do this, the Option Bar or Dialog furnishes the standard object selection options. After selection is complete, confirm by pressing Enter. At this point, definition is made and the command terminates. The part is now ready for insertion as required.

## Attributes

It is important to understand the procedures for working with Attributes. The attachment of Attributes to entities or parts can facilitate the drawing process, and help create a more meaningful drawing. It also helps set the stage for further design work using the drawing parameters, since Attributes can lend meaning to strengthen of materials calculations and other engineering requirements.

Attributes are, in general, text objects that are related to or joined to Parts. They are treated together with the Part as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single Attribute with a Part, or several.

Using Attributes, the parts can be further defined by textual information. This information may be used to label Parts in the drawing, or may be used to contribute to a data base program for analysis later in the engineering process.

## Attribute Data

The Name is the designation of the Attribute under that it is integrated in a part. Especially if a part contains more than one Attribute, the name is the most important criteria for the effective usage of the Attribute.

The Request is a component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally, the request contains the invitation to enter a variable Attribute value.

The Value is that portion of the Attribute that is written, together with the Part, into the drawing. It can be constant or variable, visible or transparent.

## Parameters for an Attribute

The following parameters for an Attribute may be defined during the creation of the Attribute, and edited later in the design process.

Insertion point

Font options, specifically Font name, font height and font angle

Reference point for insertion

Visibility, whether visible or hidden

Attribute value, whether constant or variable.

***See also:***

Defining Attributes

Inserting Parts with Attributes

Editing Attributes

Attribute Modification

## Define Attribute...



**Command: ATTDEF**

**Alias AD**

**Located:**

- 1. Pull-down Menu: Parts – Define Attribute**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Attributes are, in general, text objects that are related to, or joined to, Parts (Blocks). They are treated together with the Part as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single Attribute with a Part, or several.

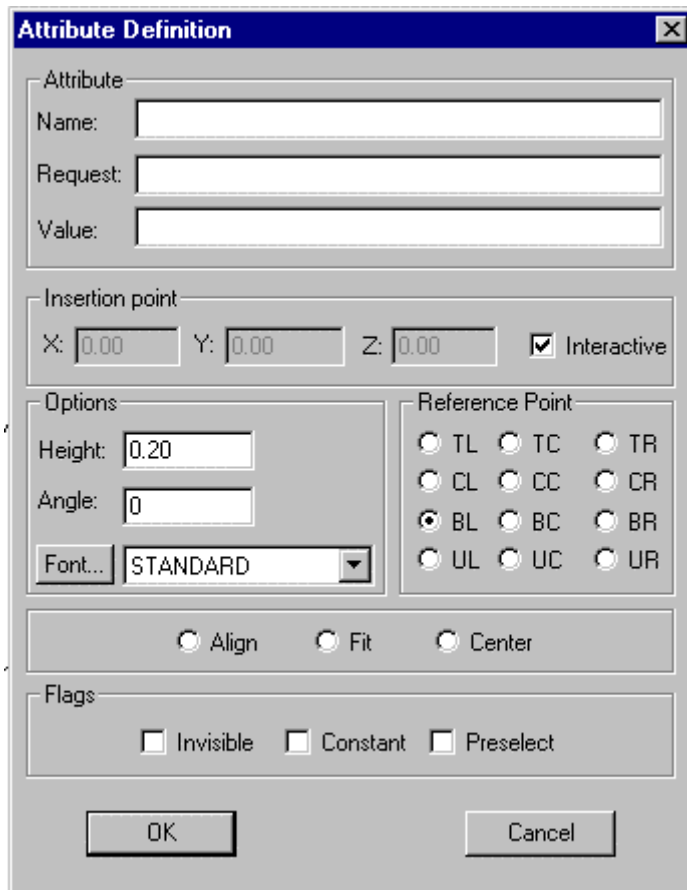
Using Attributes, the parts can be further defined by textual information. This information may be used to label Parts in the drawing, or may be used to contribute to a data base program for analysis later in the engineering process.

An Attribute consists mainly of three components; its name; its request; and the value. There are also a series of options that can be elected during the creation of an Attribute.

- The **Name** is the designation of the Attribute under that it is integrated in a part. Especially if a part contains more than one Attribute, the name is the most important criteria for the effective usage of the Attribute.
- The **Request** is a component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally, the request contains the invitation to enter a variable Attribute value.
- The **Value** is that portion of the Attribute that is written, together with the Part, into the drawing. It can be constant or variable, visible or transparent.

The following parameters for an Attribute may be defined during the creation of the Attribute, and edited later in the design process.

- Insertion point;
- Font options, specifically Font name, font height and font angle;
- Reference point for insertion;
- Visibility, whether visible or hidden; and
- Attribute value, whether constant or variable.



The image shows a software dialog box titled "Attribute Definition". It contains several sections for configuring an attribute:

- Attribute Section:** Includes input fields for "Name:", "Request:", and "Value:".
- Insertion point Section:** Includes input fields for "X:", "Y:", and "Z:" (all set to 0.00), and a checked "Interactive" checkbox.
- Options Section:** Includes "Height:" (0.20), "Angle:" (0), and a "Font..." dropdown menu currently set to "STANDARD".
- Reference Point Section:** A grid of radio buttons for selecting a reference point: TL, TC, TR, CL, CC, CR, BL (selected), BC, BR, UL, UC, UR.
- Alignment Section:** Radio buttons for "Align", "Fit", and "Center".
- Flags Section:** Checkboxes for "Invisible", "Constant", and "Preselect".
- Buttons:** "OK" and "Cancel" buttons at the bottom.

## Dialog Options

### Attribute Name

Enter the name of the Attribute in the input field. This is the name that the Attribute will carry as it is joined to the Part, so it is important to use as descriptive a name as possible. When a part contains several Attributes, the name is an important criteria for effective use of the Attribute. An Attribute name can consist of up to 31 characters, including the special characters \_ (Under line) and \$ (Dollar).

### Request

The Request is a textual component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally the Request contains an invitation to enter a variable Attribute value.

Request can consist of up to 256 possible characters. Blanks and special characters are allowed.

The user may enter in this input field, for instance, "Enter norm term". This would be a variable. A typical answer to Request would be to enter information like "Schedule 40 steel pipe".

The answer to Request is interpreted as an Attribute value and as such inserted in the drawing (see below).

Occasionally the entering of a request can be omitted. This would be the case especially when the part is using a constant Attribute value (see below).

### Value

The Value is that component of the Attribute that is written, with the Part, into the drawing. It serves as a designation of the Part and contains information to be analyzed. Its content is determined either by the input in the input window Value or by the answer to Request (see above). An Attribute Value can

consist of up to 256 possible characters. Blanks and special characters are allowed.

### **Insertion Point**

There are two options in defining this point in the Dialog Box; the entry of absolute coordinate values in the X, Y, and Z axes, or Interactive.

Interactive allows the user to define the Insertion Point by the cursor. By filling the control box marked Interactive, the user can exercise this option. The advantage to this option is that Dragmode is active as well, offering a preview of the positioning.

### **Text Options**

With the help of the text options the user may determine the height (font size), the rotation and the Font style of the Attribute text.

**New text styles** can be directly created from the **ATTDEF** command

The **FONT** command can be issued by depressing the font button on the text dialog

Define the font size by the input field Height. Enter the desired text height in drawing units, not in text points.

Via the input field Angle, specify the insertion angle of the Attribute value. For an counter-clockwise rotation enter a positive value, for a clockwise rotation enter a negative value, between 0 and 360 degrees.

A selection window is available to define the Font style. A list of the available fonts is displayed. Select the desired Font style.

### **Reference Point**

The Reference Point determines the position and alignment of the text object in relation to the Insertion Point of the Attribute. To this end, a Reference Point is defined on the text object. During insertion, this point will be coordinated with the Insertion Point.

If the reference point, for instance, is the lower left corner of the text object, then the text will run from the Insertion Point to the right, and the lower edge of the text will be located at the height of the Insertion Point.

In total there are twelve Reference Points available. Select one of them with the cursor. The Reference Points represent the initials of the position on the text object. BL, for instance, represents Bottom Left, while UR represents Upper Right. Center positions are available as well.

### **Align**

Text objects may be rotated or scaled during the process of insertion by using the function Align. Determination of rotation angle, as well as a scaling factor, is made by indicating two points in the drawing using the cursor.

Activate the option Align by filling the circle marked "Align". Activating this option will remove the options Insertion Point, Height, Angle, and Reference Point. Align requires that the user pick two points on the drawing. The text object will be placed left justified on this baseline.

Confirm the use of the option Align after the input of the string by clicking OK.

Then simply follow the prompts for a first point, then a second point. The first point is interpreted as the Insertion Point, the second point sets the angle of insertion. The second point will also determine a scaling factor that will, in turn, determine the text height.

### **Fit**

Fit enables the user to rotate and expand a text object during the insertion. The text object is not scaled, so the text height remains unchanged, but the text string is expanded or compressed as required by the Fit. The procedure for this option is:

Activate Fit by clicking the panel with this name at the lower left side of the dialog window Text. The options Insertion Point, Angle and Reference Point will not be available. The parameters required for Fit are two points that must be defined during the inserting of the text object. The reference point of the text object is automatically positioned left justified on the basis line;

Answer the prompt for "first point". This point is interpreted by the program as the Insertion Point. It is also the point of rotation for the rotation angle and the first point for the expansion factor; then

Indicate the second point. This will determine the rotation angle and expansion factor, and set the text height.

### **Center**

Center allows fast insertion of a text object with the vertical and horizontal alignment centered. Text height and rotation angle may be specified in the normal fashion.

### **Flags**

#### **Visibility**

Invisible is a switch panel that controls the visibility of the Attribute Value. A cross in the control box displays active status of the option.

If Invisible is activated, the Attribute Value will not be displayed during the insertion of the Attribute in the drawing. this is the option to choose if the Attribute contains information to be analyzed later that does not have to be displayed on the drawing.

It is also convenient to use this option when there are so many Attributes that to show them all would make the drawing to "busy", confusing the drawing.

#### **Constant**

This switch panel is used for switching between constant and variable Attribute values (see above). A cross in the control box displays the active status of the option.

With Constant activated, the text entered in the Value field will be written as a constant Attribute Value in the drawing. There will be no opportunity to insert any variable data.

With Constant de-activated, the request at the command line to insert variable Attribute Values will appear.

#### **Preselect**

The Preselect mode allows the user to create Attributes that accept their default values.

With Preselect active, Attribute Values are not requested. The Preselect mode is the most flexible mode, since the mode allows the user to edit pre-selected Attribute Values after insertion by using the editing commands.

Verify the entries by OK switch or by pressing RETURN. The Attribute will be inserted as specified.

**See also:** QATTDEF

### **Inserting Parts containing Attributes**

The insertion of internal or external parts that include Attributes is done in the same way as inserting parts without Attributes. If variable Attribute values are used the request for input of Attribute text is added. Enter the desired Attribute value in the command line area and confirm the input or confirm a possible parameter value by pressing RETURN. If the part contains several variable Attributes, repeat the input of Attribute value the required number of times. The part will be inserted in the specified position.

**See also:**

Inserting Parts with Attributes

### **Modify Single Part Attributes (one by one) ...**





## **Command: ATTEDIT**

**Alias ATT**

### **Located:**

1. **Pull-down Menu: Parts – Modify Part Attributes (1 by 1)**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

Enter the command ATTEDIT in the command line area or select **Modify Attributes...** from the *Part* menu.

The first prompt will request that the part be selected. Make this selection either by naming the part, or by selecting the part with the cursor.

After the part selection, a dialog box will appear. The editing process from this point is very similar to the procedure followed in the creation of an Attribute, except that the name is now displayed, and cannot be changed.

If a part contains several Attributes, individual Attribute values can be displayed in the input field *Value* using these external. Clicking the right double arrow will display the next Attribute, while clicking the left double arrow will display the previous Attribute is displayed. That Attribute can at this moment be edited results from the name display above the input field.

---

### **See also:**

Edit Part Attribute

## **Attribute Modification**

### **Command: ATTVALUE**

**Alias AV**

### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Modify Attribute Text Value**
2. **Or Enter Command or Alias at the Command Line**

### **Command: ATTMOVE**

**Alias ATM**

### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Move Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

### **Command: ATTROT**

**Alias ATR**

### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Rotate Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

### **Edit Attribute Text Value**

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu *Modify Attribute Text*. ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

### **Move Attribute Text**

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

### **Rotate Attribute Text**

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

## **Preload Parts**

### **Command: DINSERT**

#### **Alias DIN**

#### **Located:**

##### **1. Entered Directly at the Command Line**

The command PINSERT incorporates an external block into the current drawing without creating an instance of the block in the drawing. This variant can be seen as discrete insertion.

The main purpose of this command is

- to preload parts (blocks) for later use
- to load table definitions defined in a template drawing (layers, linetypes, text styles, dimension styles, etc.)

The only request of the command is to specify the file name of the external part to be brought in, but not actually be inserted into the drawing. Only the symbol tables will be enhanced by those (additional) entries found in the discrete inserted drawing.

This command is provided especially for use in the programming interfaces.

---

#### **See also:**

INSERT

## Explode Complex Objects...



**Commands:** EXPLODE, XPLODE

**Alias** EX. XPL

### **Located:**

1. **Pull-down Menu:** Edit – Explode Complex Objects
2. **Palettes:** Edit and Parts
3. **Or Enter Command or Alias at the Command Line**

Parts and other complex objects, like polylines, crosshatches, and dimensioning, inserted in the drawing constitute one single drawing object.

In this mode, it is not possible to edit any of the individual entities that make up the part. If, for any reason, there is a requirement to edit one of the entities comprising a part, then the part must be "exploded" (broken into its individual entities), the element(s) edited, and then the component parts reassembled again.

The explosion of complex objects is executed by the command EXPLODE, or by selecting and activating the option **Explode Complex Objects...** in the *Edit* menu.

When the object selection is complete, the objects (parts, polylines, hatches, or dimensions) will be exploded.

Now you may choose any of the entities for individual editing.

### **Notes:**

1. It is important to realize that, if there are attributes attached a part to be exploded, these attributes will be lost. Instead, the attributes defining names will appear.
2. Exploding a part does not change the part definition in the drawing. Another part or parts with the same name and original definition can be re-inserted into the drawing at any time.
3. To change the part definition in the drawing, you must modify the part components after exploding, then create a new internal or external part with the PARTEXP or PARTDEF commands, using the old part name. The program will ask you to confirm the re-definition when you do so.

## Externally Referenced Drawings



**Command:** XLINK

**Alias** XL

### **Located:**

1. **Pull-down Menu:** Parts – External References
2. **Palette:** Parts
3. **Or Enter Command or Alias Directly at the Command Line**

Externally Referenced Drawings (XREF's) serve especially to

7. Create assembly drawings from different files
8. Insert detail drawings into a finished one
9. Temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is realized within a running project

where a workgroup uses XREF's to create several drawings and details in a network.

Another advantage is that within the current drawing, only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file. This keeps the file size small and as such it is easier to work with. In this way, very complex drawings can be created from a group of relatively small files. In these instances, XREF's have an advantage over the use of blocks.

### **XLINK command: Control externally referenced drawings**

The XLINK command allows you to create a link to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

#### **Differences to Part Insertions**

Although the treatment of linked drawings is basically similar to those of part insertions (block insertions), some fundamental differences include:

An Xref establishes a link to another drawing file, but it does not become a permanent part of the current drawing.

#### **Dependent Symbols**

When attaching an external file, those symbol tables are loaded into current drawing that are required to display the referenced coincident to its original: layers, linetypes, text styles, dimension styles, and block definitions.

The program utilizes the following scheme to name dependent symbols in the current drawing:

Logical\_Xref\_Name| Dependent\_Symbol\_Name

For example, the layer names of an externally referenced drawing attached with the logical name SHAFT are listed as:

SHAFT|CONSTRUCTION

SHAFT|DETAILS

SHAFT|DIMENSION

Other dependent symbol table entries like linetypes or blocks will carry the same naming convention.

Note that you cannot redefine or rename dependent symbols. Also, dependent blocks cannot be inserted and a dependent layer cannot be made the current layer.

But the dependent symbol conventions allow you to control the visibility, the colors, and the linetypes of externally referenced drawings. The layer dialog box displays the names of the dependent layers and you can apply the options On/Off, Thaw/Freeze, Linetype, and Color on them.

The logical name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

To keep the visibility and display settings over subsequent sessions, make sure to set the system variable **VISRETAIN** to 1. By default VISRETAIN is set to a value of 0 that means that visibility and display modifications made to the dependent layers are not maintained when you leave the current drawing.

#### **The Dialog Box *Externally Referenced Drawings***

The dialog box displays a list box and a column of option buttons.

The list box displays the following information on externally referenced drawing files that have been inserted into the current drawing

1. The logical name assigned to it when the file was attached, and
2. The path and filename of the referenced drawing

The options available to manage externally referenced drawings are provided in a column of buttons:

<b>Attach</b>	Insert an external drawing file into the drawing as block
<b>Reload</b>	Update externally referenced drawing to display its current state
<b>Path</b>	Relocate/redirect or alter source of the referenced file
<b>Bind</b>	Transform an externally referenced drawing to an ordinary block in the drawing
<b>Detach</b>	Delete an externally referenced drawing from the current drawing and erase the dependent symbols from the drawing database tables

These options are described in detail in the following sections.

### **Attach**

The *Attach* option allows you to insert an external drawing as a block to the current drawing.

#### **To attach (set up a link and insert the contents) to an external drawing file:**

1. The standard file dialog box allows you to specify the drawing you want to attach to the current drawing.
2. Within the dialog box *Attach External Drawing*, a unique Logical Name (block name) needs to be determined for the associated referenced file. The program suggests using the name of the file. However, in many cases it is recommended to use a short logical name, because the logical name will be used as prefix for the names of dependant symbol names for layers, linetypes, etc.
3. The other options of the dialog box let you specify preferences for the insertion and are identical to those found in the dialog box of the INSERT command (insertion point, scale factor, and rotation angle), as described previously.

The drawing attached is based on the most recently saved version of the drawing file.

### **Reload**

The option *Reload* allows you to update and external reference any time during a current drawing session. Any Reload reflects any subsequent modification made to the external file.

Note that when you open a drawing that contains externally referenced drawings, these are displayed automatically in their current state. When working on a project in a workgroup over a network, it might be necessary to update one or more external references while you are working on a current drawing. The program will reload the specified drawing(s) in the shape as they have been saved most recently.

#### **To reload externally referenced drawings in a current drawing session,**

1. select the external reference to be updated from the list box
2. click the *Reload* button

### **Path (Relocate or alternate source of the referenced file)**

Occasionally the storage location of an externally referenced drawing might have been changed or the referenced file has been renamed. Also, it might happen that, when sending the project files including referenced drawings to someone else, the path to the referenced files needs to be updated.

In such a situation the option *Path* of the dialog box allows you to re-establish the path to the referenced drawing, to reload a renamed referenced file, or to replace a referenced drawing by another file.

#### **To edit or relocate the path to a previously linked reference drawing,**

1. select the external reference to be relocated from the list box
2. click the *Path* button

From now on, the link to the referenced file will be maintained as specified and the data of the referenced file will be updated from that source.

## Bind

In certain situations it is useful to make an externally referenced drawing a permanent part of a drawing, for example when you need to send the drawing to someone else or if it has been finalized and is going to be archived. Binding the referenced drawing to the current drawing has the advantage that there is no need to gather an assembly of drawings within an archive or mail.

The option *Bind* of the XLINK command allows you to transform an externally referenced drawing to an ordinary block in the drawing. Once you bind an externally referenced drawing, the link to it is broken and you will no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definition, text styles, and dimension styles into the current drawing. Within each corresponding dependent symbol name, the vertical bar symbol (|) is replaced by a new three character symbol \$n\$, where n is a number that is usually 0. For example, the layer name HOUSE|BASEMENT will be altered to HOUSE\$0\$BASEMENT.

### To bind one or more external references to the current drawing,

1. select the corresponding entries in the list box of the dialog box
2. click the *Bind* button

## Detach

Externally referenced drawings that are not needed any longer in the current drawing can be removed from the base drawing with the *Detach* option. This will remove all of its dependent layer, linetype, and block references as well.

Note: Although you can erase an externally referenced drawing with the DELETE command, it is recommended to use the Detach option of the XLINK command, as this operation will also remove the Block definition from the drawing database.

### To remove one or more external references from the current drawing,

1. select the corresponding entries in the list box of the dialog box
2. click the *Detach* button

## Open

As the Multiple Document Interface allows you to display up to four drawings on the desktop, the option *Open* of the dialog box allows you to open an externally referenced drawing simultaneously on the CAD desktop. This allows you to modify a referenced drawing, for example, if you have detected discrepancies or inconsistencies in the linked drawing that needs to be fixed.

### To open a referenced drawing in another drawing window (viewport),

1. select the external reference you want to open from the list box
2. click to the *Open* button

Once you have modified a referenced drawing, save the drawing (and, if the modification has been finished, close it). To display the revised external reference in the master drawing, call the command XLINK and reload the corresponding file (see above: *Reload*).

## Insert

The option *Insert* allows you to insert another copy of the Xref chosen from the list box into the current drawing. You may then define the insertion point, the scale factor and the rotation angle for that additional instance of the external drawing in the current drawing.

You can move, scale, rotate, and copy externally referenced drawings. But, you cannot explode an external reference.

**Note:** You should not remove an Xref from the current drawing with the DELETE command, because it does not purge the Xref block definition from the drawing database and the associated symbols will be

maintained. To remove an external reference correctly from the current drawing, it is recommended you use the Detach option of the XLINK command.

The Insert option, however, can serve to reinsert an external reference that accidentally has been deleted.

---

**See also:**

XINSERT

Attach an External Drawing File to the Current Drawing

**Command:** XINSERT

**Alias** XI

**Located:**

**1. Entered Directly at the Command Line**

The command XINSERT is processed similar to the XLINK command applied with the *Attach* option. The command is provided for convenience, allowing you to directly attach an external drawing to the current drawing.

---

**See also:**

XLINK

## Coordinate Systems

### User Coordinate System

A user coordinate system can be freely defined by the user, having the world coordinate system as a basis. The following changes can be made in a user coordinate system:

The origin can be moved to any point in the world coordinate system.

The user coordinate system can be angled or turned around one or more axis in relation to the world coordinate system.

Using the user coordinate system, for example, it is possible to change the location of the drawing plane in such a way that it always lies parallel to the objects that are to be drawn, making the construction of 3-dimensional objects much easier.

### Drawing in real world units (1:1 scale)

In a hand-drafted drawing, you draw to a scale. For example, a 1=2 or half-scale would cause all dimensions to be half the real-world size. You would use an engineering or architectural scale to draw a two-inch line, one inch long that is half the size of the real-world distance. However, full-scale represents the real-world distances, and drawing at full-scale, a two-inch distance would be drawn at two inches. If you have experience in drafting, you are used to having to choose the scale before drawing one line. While drawing, however, you always draw in full scale. For people with drafting experience drawing in full scale is somewhat unusual. Remember these important rules:

1. Draw in full scale.
2. Determine paper scale when printing or plotting.

### Coordinate Input

During the drawing process it is often necessary to enter coordinates for defining points of the drawing (starting and ending points, center points, points of contact, inserting points, base and target points)

Coordinates are entered in the following formats:

1. **Cartesian Coordinates** - Cartesian Coordinates define a point by its X, Y, and Z-values. These values are entered separated by a comma. A point is used as a decimal sign. If no Z-value is

entered it will be interpreted as Z=0.

2. **Polar Coordinates** - Polar Coordinates define a point by its distance from the coordinate origin and the angle in the XY-plane (from the X-axis). The two values are separated by the sign < (smaller than). For example, the value input *10<60* defines a point with the distance of 10 drawing units from the origin of the coordinate system and an angle of 60 degree from the X-axis.
3. **Spherical Coordinates** - Spherical Coordinates define a point by its distance from the coordinate origin, the angle in the XY-plane, and the angle to the XY-plane. For instance, the value input *10<60<45* describes a point with the distance of 10 drawing units from the origin of the coordinate system, an angle of 60 degree from the X-axis in the XY-plane, and an angle of 45 degrees from the XY-plane in direction to the Z-axis.
4. **Cylindrical Coordinates** - Cylindrical Coordinates define a point by its distance from the coordinate origin, the angle in the XY-plane and its Z-value. Distance and angle are separated by < , angle and Z-value by comma. For example, the input *10<60,5* defines a point with the distance of 10 drawing units from the origin of the coordinate system, an angle of 60 degree from the X-axis in the XY-plane, and a Z-value of 5 drawing units.

This table gives an overview of the input formats of the different coordinate types, both for absolute as for relative coordinates.

The different coordinate types can always be entered as absolute or as relative coordinate values.

**Absolute Coordinates** - Absolute Coordinates use as their reference (starting) point the origin of the current coordinate system, or, in the case of angle values, the angle to the XY-plane of the coordinate system.

Type of coordinate	Input	Example
Cartesian	X,Y,Z	3,8,6
Polar	distance<angle	7.5<45
Spherical (3D)	distance<angle,Z-coord	7.5<45,12.2
Cylindrical(3D)	distance<angle1<angle2	7.5<45<33

**Relative Coordinates** - Relative Coordinates use as their starting point the last defined point. Relative coordinates are selected by adding a sign @ before the value.

Type of coordinate	Input	Example
Cartesian	@deltaX,deltaY,deltaZ	@3,8,6
Polar	rel. distance<angle	@7<45
Spherical (3D)	rel. distance<angle,Z-coord	@7.5<45,12.2
Cylindrical (3D)	rel.distance<angle1<angle2	@7.5<45<33

## Command Reference

... provides information about the commands found in the menus.

## User Coordinate Systems



**Command: UCS**

**Alias UC**

**Located:**

1. **Pull-down Menu: Tools – World Coordinate System and**

**Tools - User Coordinate System – UCS Manager (UCS\_C) and**

**Tools - User Coordinate System – UCS Origin (UCS) and**



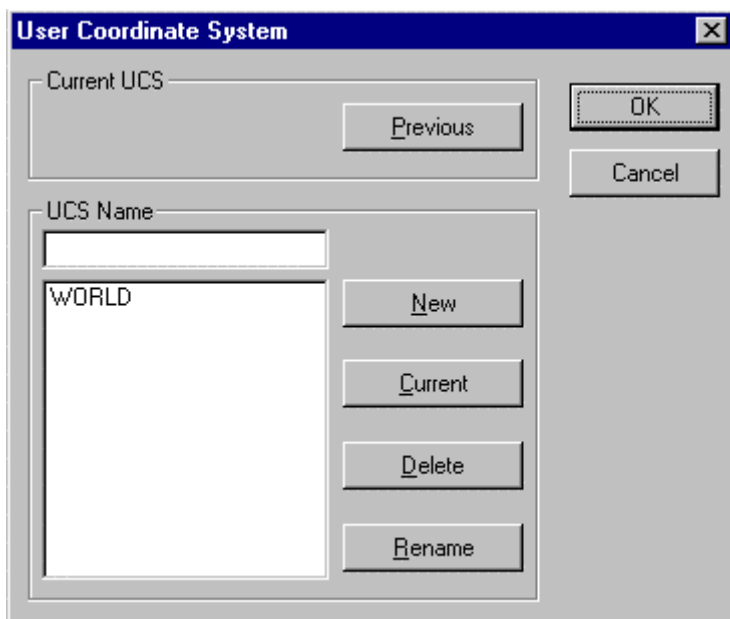
## Tools - User Coordinate System – Rotate Around Z Axis (UCS\_R-Z)

2. Left SideBar
3. Palette: View
4. Or Enter Command or Alias at the Command Line

When the UCS command is issued, an option menu is presented that lead directly to individual UCS functions.



The UCS Manager function is available by clicking on the Control option in the menu above. The UCS Manager controls the setup and definition of User Coordinate Systems.



It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate systems will be saved along with the drawing and can be used again when opening the drawing later on.

### Active coordinate system

From all the user coordinate systems in one drawing, only one can be active at any given time.

Choosing a coordinate system to activate is done in a dialog box. Using the same dialog box it is possible to rename, create or delete already existing user coordinate systems.

### Create new user coordinate system

User coordinate systems are always created in two steps. First the parameters of the user coordinate system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system, enter the command UCS. Through the message UCS **origin <0,0,0>** in the text window you are asked to enter the coordinates of the origin of the user coordinate system. The coordinates **<0,0,0>** are offered as a help. If you don't want to change the origin of the coordinate system then verify this choice through enter. Otherwise please enter new coordinates and verify this entry also with return.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented..

Control      World      Z-direction      3point      View      Rotation

The option **Control** calls up a dialog box that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog box will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option **World** chooses the world coordinate system to be the active coordinate system. This allows you to go back to the world coordinate system quickly without having to open the dialog box of the command control.

### Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .
2. The direction of the axis of the coordinate system can be determined through fixing the z-axis.

To do that, choose a point through that the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
> UCS
UCS origin <0,0,0>: Z-axis
UCS origin <0,0,0>: Enter
Point on the positive z-axis: P1
```

### 3Point

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane, nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the objectcatch function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through that the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so that they are oriented at the plane of the drawing object at that the user coordinate system should be adjusted. For example, choose the corner

points of this plane with the objectcatch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
> UCS
UCS origin <0,0,0>: 3Point
UCS origin <0,0,0>: P1
Point on the positive x-axis: P2
Point on the positive xy-plane: P3
```

## View

This option adjusts the user coordinate system parallel to the active viewing plane, the screen. Through this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

## Rotation

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly.

To call up the option rotation, please click onto its field among the options.

With one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around two or three axes, please repeat the following steps for each axis.

1. First determine the axis around that the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z) that will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. Entering or pointing an angle does this.

The angle of the rotation can be determined either by entering it manually or by the pointing of two points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
> UCS
UCS origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z axis <0>: 30
```

## Controlling the UCS

Of course it would not be very user-friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore the program offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager that is found when using the standard menu in the pull-down menu view.

Also it is possible to open a dialog window by entering the command UCS and choosing the option *Control*.

## Create a new UCS

The creation of a user coordinate system is done in two steps

1. Arrange the coordinate system with the previously described options z-direction, 3point, view

and/or rotate.

2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate system has been adjusted.

The dialog box has an option that is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and \_ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name, click onto the button **New** and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember that all changes made in the user coordinate system dialog box are registered only after clicking OK as you leave the box.

### **Renaming a user coordinate system**

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click onto the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.

Please remember that all changes made in the user coordinate system dialog box are registered only when you click on OK as you leave the box.

### **Erasing a user coordinate system**

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember that the world coordinate system cannot be deleted. Please remember that all changes made in the user coordinate system dialog box are registered only when you click OK as you leave the box.

### **Changing the active coordinate system**

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.

The **active coordinate system** is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).


1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate system.
3. The field "active user coordinate system" shows the name of the active coordinate system.

Please check there that the name of your chosen coordinate system appears there.

Changes to the active coordinate system are registered only when you click OK as you leave the dialog box.

### **Change to the previous coordinate system**

An option exists in the dialog window "User Coordinate System" to change to a previous coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Note: While a brief overview of UCS concepts has been provided here, users are encouraged to consult the **User Manual** , as well as other CAD documentation for a more in-depth exploration of the subject.

## **Plan View**

**Command: PLAN**

**Alias PLN**

### **Located:**

1. **Entered Directly at the Command Line**

The PLAN command allows you to set the plan view to the World Coordinate System (WCS), to the current User Coordinate System (UCS) or to any other named UCS.

> PLAN

Set plan view to named UCS, World coordinate system, or press <Enter> for current UCS:

---

**See also:**

PLANVIEW

UCS

3DVIEW

Set Plan View to Construction Plane



**Command: PLANVIEW**

**Alias PV**

### **Located:**

1. **Pull-down Menu: Tools – Set Plan View to Construction Plane**
2. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
3. **Left SideBar**
4. **Palette: View**
5. **Or Enter Command or Alias at the Command Line**

The PLANVIEW command sets the view planar to the current User Coordinate System (UCS). Normally, this command is used to return from an isometric 3-dimensional view of the drawing, to the standard

(from above) view of the drawing.

---

**See also:**

PLAN

3DVIEW

UCS

**Absolute Coordinates...**

Type of coordinate	Input	Example
Cartesian	X,Y,Z	3,8,6
Polar	distance<angle	7.5<45
Spherical (3D)	distance<angle,Z-coord	7.5<45,12.2
Cylindrical (3D)	distance<angle1<angle2	7.5<45<33

**Relative Coordinates...**

Type of Coordinate	Input	Example
Cartesian	@deltaX,deltaY,deltaZ	@3,8,6
Polar	rel. distance<angle	@7<45
Spherical (3D)	rel. distance<angle,Z-coord	@7.5<45,12.2
Cylindrical (3D)	rel.distance<angle1<angle2	@7.5<45<33
Directed (direction to cursor)	distance <<	20'<<

**Precision Aids**

**Units, Angles, Precision**

For entering and displaying coordinates and distances the following *unit systems* are available:

Scientific	1.234E+01
Decimal	12.340
English: Engineering	1'-0.340"
English Architectural	1'-0 3/8"
Fractional	12 3/8

For *Input Precision of Linear Units* you can choose from 0 to 8 decimal places.

You can select one of the following *Angle Measurement Units*:

Decimal degree	45.0000	d = Degree; ' = Minutes; " = Seconds
Degree/Minutes/Seconds	45d0'0"	
Grads	50.0000g	
Radians	0.7854r	
Surveyors	N 45d0'0"	

For *Precision of Angle Specification* you can choose from 0 to 8 decimal places.

The standard *Angle 0 Direction* is East, or, as on a clock face, at 3 o'clock. Any other direction may be specified.

The angle direction can be determined either as *Counter-clockwise (CCW)* or *Clock-wise*.

**Precision Aids Parameters**

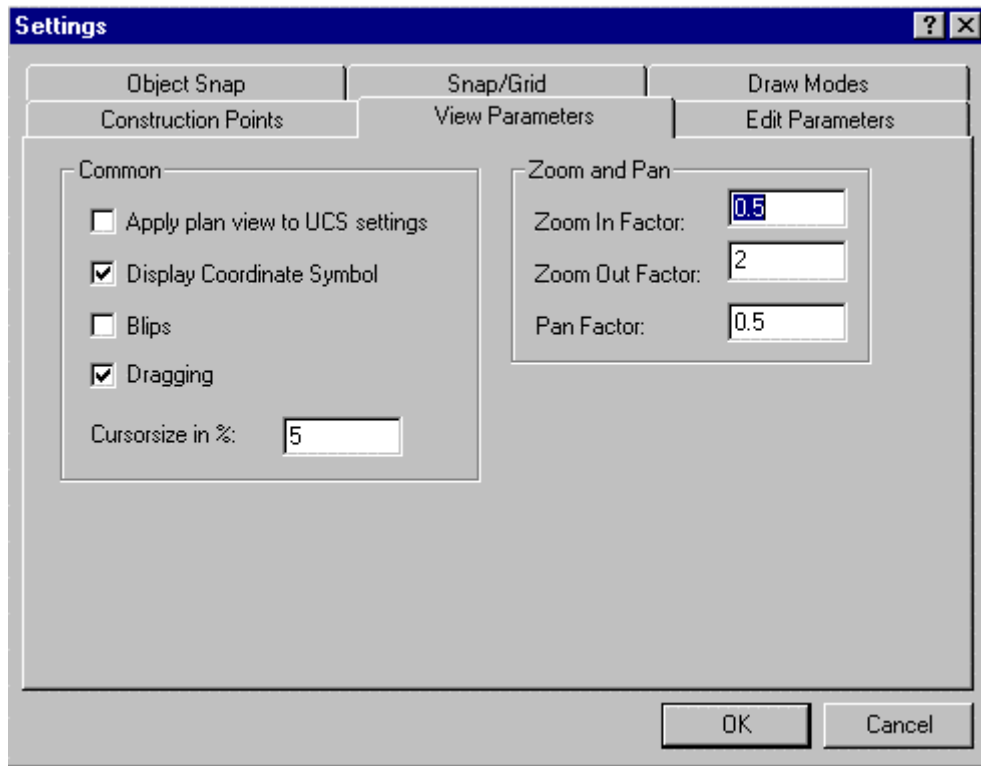
**Command: PRECPAR**

## Alias PCP

### Located:

1. **Pull-down Menu: Tools – Set Drawing Modes – Drawing With Precision**
2. **Or Enter Command or Alias Directly at the Command Line**

The PRECPAR command leads directly to various sections of the SETTINGS dialog, that control selection precision. Each tab has settings that can be adjusted to control drawing precision.



### Grid Display

Activate or de-activate the grid by clicking the check window on the upper left in the box **Grid Spacing**. A cross in this check box indicates the grid display is active.

In the input area **Factor** enter the value that defines the relation of the grid points to the snap grid. Alternatively, the user can specify a predefined value by clicking available buttons.

### Snap Grid

Activate or de-activate *Snap Grid* by clicking the check box on the upper left side in the area Snap Grid. A cross in the check box signals that the snap grid is switched on. From the dialog box, select the input fields Width and Height, and enter values for the horizontal and vertical distances between the points of the snap grid. Units of measurement for these values are, again, drawing units. The value 1.00 represents the distance of one drawing unit.

Using the palette button ( = ) for the value Width enables the user to choose a vertical value equal to the horizontal value.

The input fields marked "Origin" in this box enable the user to define the origin of the snap grid on the X and the Y-axis. The desired coordinates for the origin should always be entered.

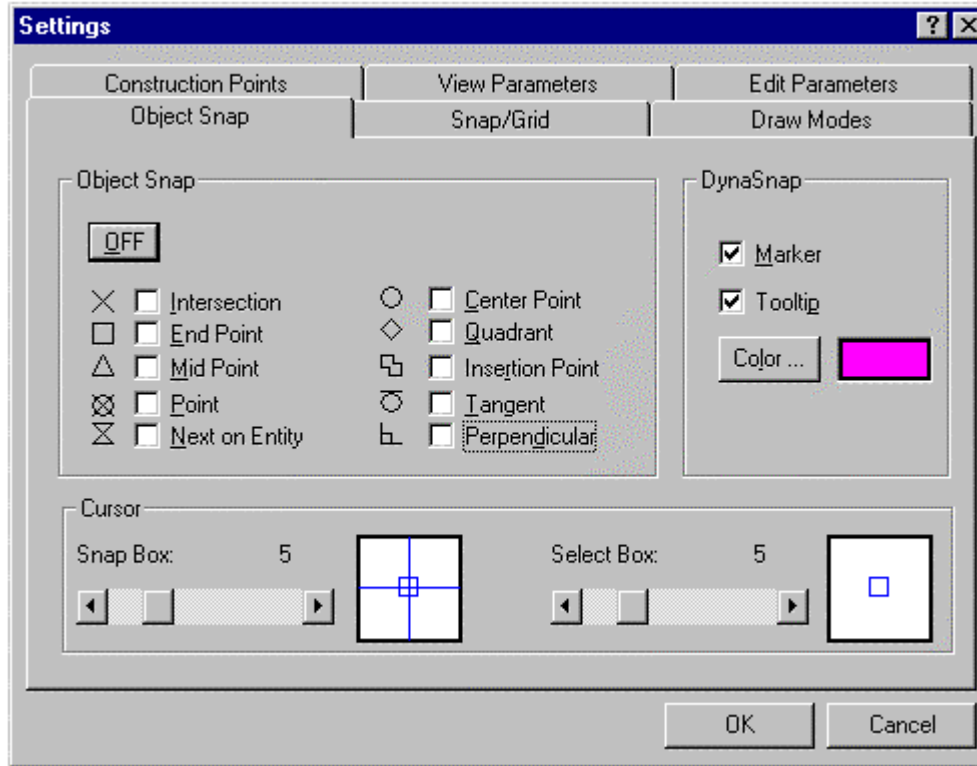
Sample

### Orthogonal Mode

Activate or de-activate **Orthogonal Mode** by clicking the check box.

ORTHOOGONAL MODE: Drawing and Modifying Objects

### Object Snap



The Object Snap tab sets the active object snaps and controls whether dynasnap (the symbols for each are shown beside each object snap selection box) and tooltips are displayed by the program, or not. The color of both the dynasnap markers as well as tooltip color is also set on this dialog.

In the lower part of the dialog, you may specify the size (in pixels) of both the cursor snap and selection box.

### Cursor

Object Snap tab allows specifying the Select Box and Snap Box size.

View Parameters tab lets you control the size of the cursor axes.

### Precision Aids Parameters

**Command: PRECPAR**

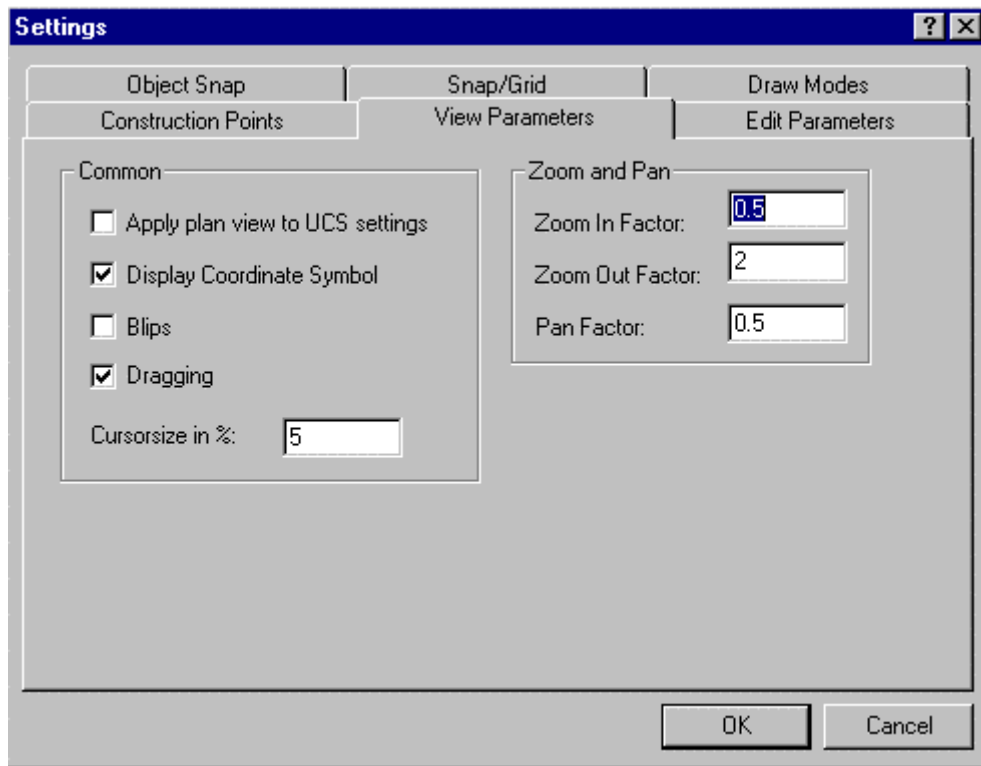
**Alias PCP**

**Located:**

1. Pull-down Menu: Tools – Set Drawing Modes – Drawing With Precision
2. Or Enter Command or Alias Directly at the Command Line



The PRECPAR command leads directly to various sections of the SETTINGS dialog, that control selection precision. Each tab has settings that can be adjusted to control drawing precision.



### Grid Display

Activate or de-activate the grid by clicking the check window on the upper left in the box **Grid Spacing**. A cross in this check box indicates the grid display is active.

In the input area **Factor** enter the value that defines the relation of the grid points to the snap grid. Alternatively, the user can specify a predefined value by clicking available buttons.

### Snap Grid

Activate or de-activate *Snap Grid* by clicking the check box on the upper left side in the area Snap Grid. A cross in the check box signals that the snap grid is switched on. From the dialog box, select the input fields Width and Height, and enter values for the horizontal and vertical distances between the points of the snap grid. Units of measurement for these values are, again, drawing units. The value 1.00 represents the distance of one drawing unit.

Using the palette button ( = ) for the value Width enables the user to choose a vertical value equal to the horizontal value.

The input fields marked "Origin" in this box enable the user to define the origin of the snap grid on the X and the Y-axis. The desired coordinates for the origin should always be entered.

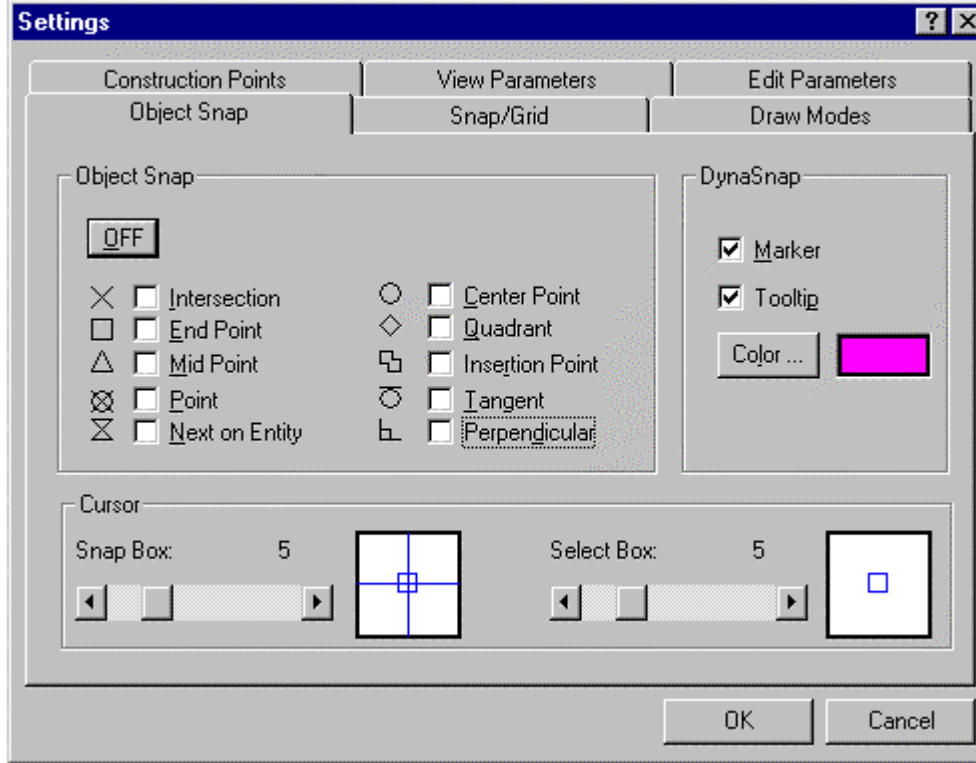
Sample

### Orthogonal Mode

Activate or de-activate **Orthogonal Mode** by clicking the check box.

ORTHOGONAL MODE: Drawing and Modifying Objects

## Object Snap



The Object Snap tab sets the active object snaps and controls whether dynasnap (the symbols for each are shown beside each object snap selection box) and tooltips are displayed by the program, or not. The color of both the dynasnap markers as well as tooltip color is also set on this dialog.

In the lower part of the dialog, you may specify the size (in pixels) of both the cursor snap and selection box.

### Cursor

Object Snap tab allows specifying the Select Box and Snap Box size.

View Parameters tab lets you control the size of the cursor axes.

## Object Snap

Many functions and commands require the definition of an exact position in the drawing. The determination of exact points can be done either by coordinate input or by indicating points. The input of coordinates has the advantage of precise placement, whereas indicating points often has an advantage in speed and ease of handling.

A very efficient method to take advantage of both techniques is to indicate points with precise references to a given geometry with the help of the Object Snap functions. The individual modes of Object Snap functions can be called for any single selection operation from the proper palette or via the cursor menu (middle mouse button) or be activated (permanently) using the *Precision Aids* dialog box (PRECPAR command).

Object Snapping allows geometrically significant points of existing drawing entities to be found. These points are then used as inputs for a new drawing step.

The Object Snap functions can be activated permanently or be called for a single operation. These techniques are described in the paragraph titled "Controlling the Object Snap Functions".

**See also:**

Mode of Operation

Object Snap Modes in Detail

## Object Snap: Mode of Operation

The object snap functions cannot be called separately, but may be used only with a command, that requests the input of a point.

The operating principal when using object snap functions is always the same:

1. Call a command or a function, that requires a point;
2. When the point is requested, answer by calling an individual object snap function or a special object snap mode. The cursor then takes the form of a small box. The command line area displays a message, that requests the point.
3. Select the object with the cursor.

The geometrically significant point that is "picked" during the activity depends on:

1. The type of the drawing entity;
2. The respective object snap mode; and
3. The specific point of the object that is chosen.

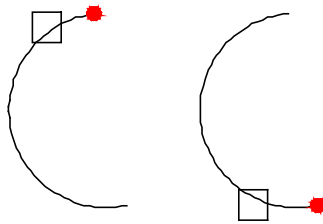
Object snap modes are occasionally not applicable. For instance, the user cannot define the ending point of a circle and while a line segment has no central point.

On the other hand, it is possible that a drawing entity contains several points to which the criteria of the object snap function correspond. A line entity or an arc, for instance, always has two ending points.

**Note:** The object snap function *Endpoint* does not differentiate in regard to the starting or ending point of an object.

An object snap function identifies, in those cases, the nearest of two or more eligible points. When selecting an object by positioning the cursor, it determines which of the possible points will be snapped to. The user should always place the cursor as close to the desired point on the object as possible.

The illustration below shows the connection between the point selected and the point snapped:



**See also:**

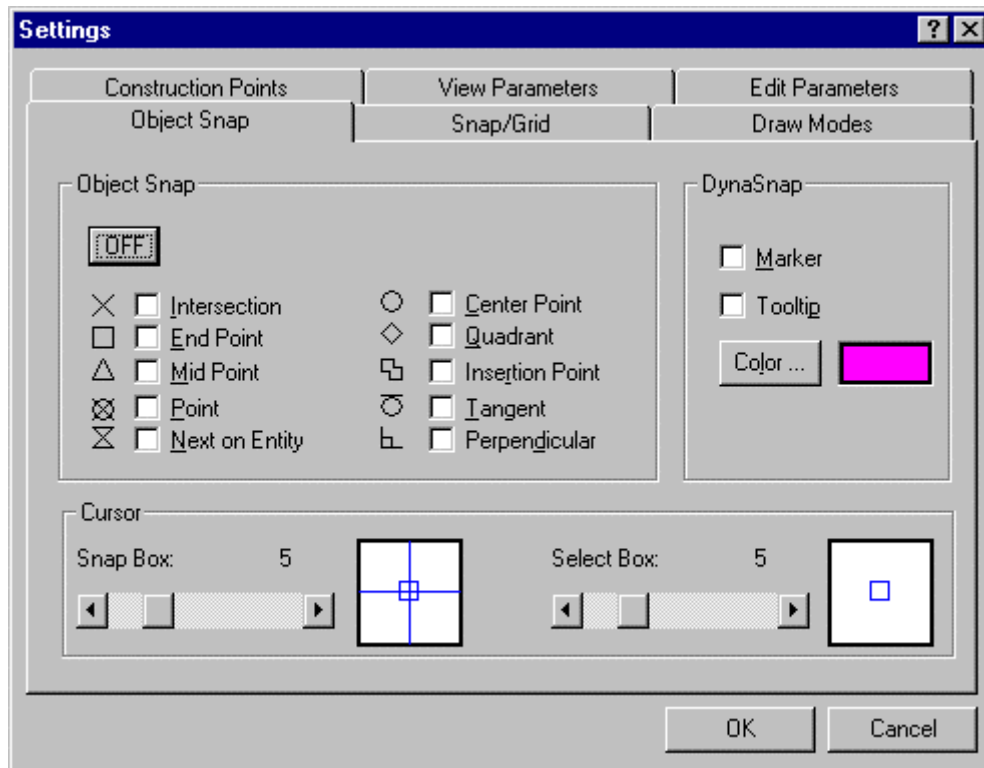
Controlling the Object Snap Functions

Object Snap Modes in Detail

## Controlling the Object Snap Functions

### Object Snap: Permanent Activation

Using the Object Snap tab on the SETTINGS command dialog, individual modes of the Object Snap functionality can be permanently activated.



As can be seen, individual snaps can be made active, either singly or in combination with other object snaps, simply by selecting the check box beside the snap. To the left of the check box, you will notice the symbol that appears onscreen when Dynasnap markers have been switched on (in the Dynasnap section to the right) and when the snap has been activated by the proximity of the cursor to the snap point. Pressing the OFF button will de-select all selected snaps, at once.

As mentioned, Dynasnaps and Tooltips (pop-up identification messages) can be switched on or off in the Dynasnap section on the right side of this dialog. Color for both can also be changed in this section, as well.

At the bottom of the dialog, you will find controls that designate the size (in pixels) for both the cursor snap and select boxes.

### Implicit Object Snap: Activating from Case to Case

For the single use activation of a certain mode of the Object Snap function you can choose between several options:

#### Snap Palette

The user may use the Snap Palette to call individual object snap modes for singular use in combination with a designing or editing command. This standard palette can be opened when needed and be placed anywhere on the drawing area. The user may now call the object selecting modes by cursor selection. The desired object snap modes can also be included in user-defined palettes with the help of the menu and dialog editor.

#### Cursor Menu

Another very effective method of calling an individual object snap mode is by activating the Cursor Menu using the middle mouse button.

#### Keyboard Input

Keyboard input of a command abbreviation is another method of activating the object snap functions. The object snap functions and their abbreviations are as follows:

<b>Object Snap Mode</b>	<b>Object Snap Input Abbreviation</b>
End	end
Mid	mid
Center	cen
Quadrant	qua
Intersection	int
Implied Intersection	imp
Perpendicular	per
Tangent	tan
Next	nxt
Insertion point	ins
Point entity	pnt

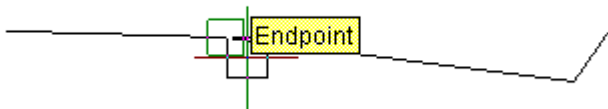
See also:

[Object Snap Mode of Operation](#)

[Object Snap Modes in Detail](#)

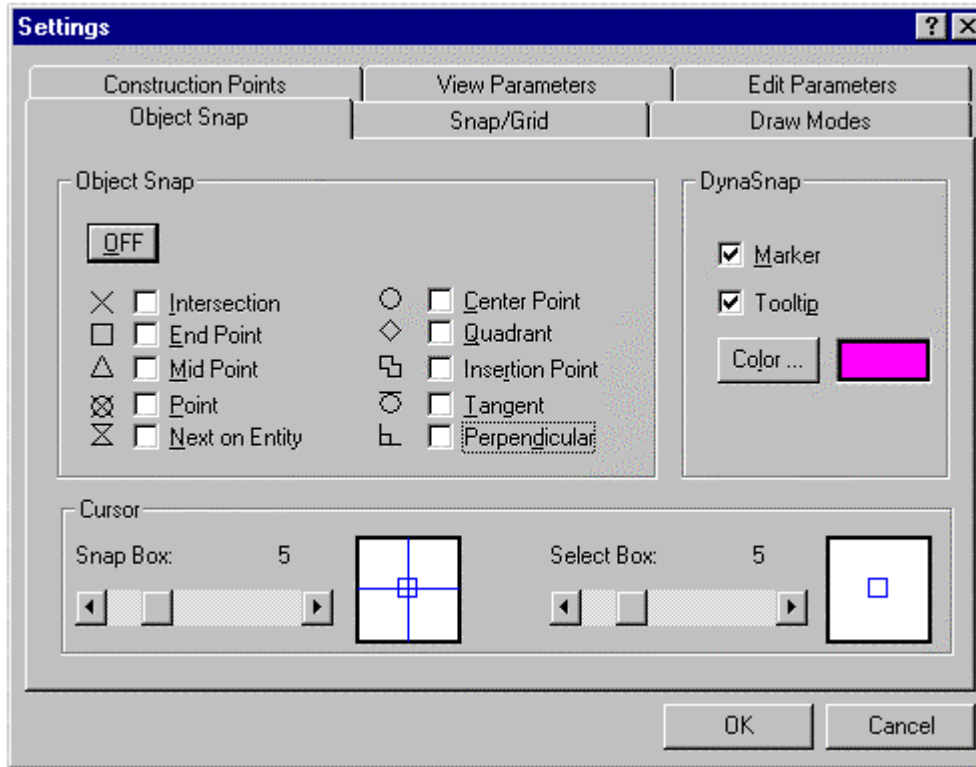
## Dynasnaps

AllenCad's "Dynasnap" (Dynamic Snap) capability works directly with currently active object snaps to indicate which snap will be applied if selection is made at the current cursor location. Dynasnap does this by displaying the symbol, a tooltip or both symbol and tooltip for the controlling Osnap next to the cursor. The user can configure which dynasnaps will display. If several Osnaps have been activated, their various symbols and tooltips will appear and disappear as the cursor is moved around on the drawing and as snaps are activated and deactivated by the cursor's proximity to snap points. In the graphic below, we can see that the endpoint of the line segment has fallen within the cursor snap box and that both a green square identifying the active snap and it's location, as well as a tooltip indicate the currently active snap.



Note that dynasnaps are displayed only while commands that require point selection are active. Only one snap can be in control at any given cursor and is generally determined by the closest snap point to the cursor. The exception to this rule is the Center Osnap, where a section of the circle itself must be within the cursor snap box. When more than one snap point is within range, the closest point will take precedence. As the cursor moves closer to another snap point, the symbol and or tooltip will change to indicate that the second snap has taken over from the first.











Dynasnap is activated and configured in the settings dialog under the Object Snap tab. The OSNAP command will take the user to this dialog directly. As can be seen on the graphic below, the choice of Marker or Tooltip can be made by clicking on the check box in the dynasnap section of the dialog. The color of the marker can also be selected by clicking on the color button.



Across in the Object Snap section of the dialog, the dynasnap symbols can be seen adjacent to their pick boxes. Below that, in the Cursor section of the dialog, the snap box slider controls the snap box size. When snap points come within this box, they become active contenders for selection.

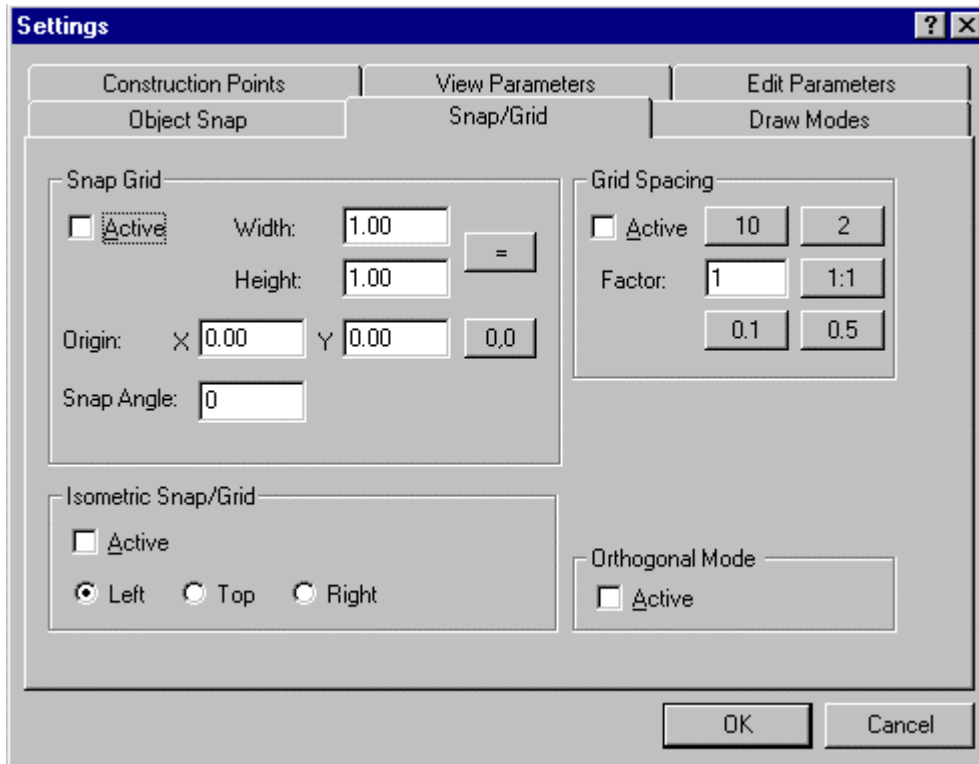
### Object Snap Modes:

Turns on running object snap:

Mark er	Mode	Supported Entities
	Intersection	Snaps to the intersection of an arc, line or, <u>polyline</u> segment.
	End Point	Snaps to the closest endpoint of an arc, line or <u>polyline</u> segment, solid, or 3dface.
	Mid Point	Snaps to the midpoint of an arc, line or <u>polyline</u> segment, solid, or 3dface.
	Point	Snaps to a point object.
	Next on Entity	Snaps to the nearest point on an arc, circle, line, point, or <u>polyline</u> segment.
	Center Point	Snaps to the center of an arc, circle, or <u>polyline</u> <u>arc</u> segment.
	Quadrant	Snaps to the quadrant point of an arc, circle, or <u>polyline</u> <u>arc</u> segment.
	Insertion Point	Snaps to the insertion point of an attribute definitions, part/block, or text.
	Tangent	Snaps to the tangent of an arc, circle, or <u>polyline</u> <u>arc</u> segment.
	Perpendicular	Snaps to a point perpendicular to an arc, circle, <u>polyline</u> segment, line, solid, or 3dface.

## Snap Grid Control

Snap grid parameters are controlled through settings made on the Snap/Grid tab of the SETTINGS command dialog.



Here, you control the overall size and location of the grid, the grid intersection spacing, whether Isometric (3-D) snap grid is active and in which orientation, and finally whether orthogonal mode is active.

## Coordinate Point Filters

In most situations, you will enter points as inputs to commands, by either typing in coordinate values, or using precision aids such as object snaps to select pre-defined points in your drawing. Coordinate filters, in the other hand, allow you to specify the x, y and z coordinates of the new point individually and to allow you to use precision aids in locating these x, y or z coordinate values. Using point filters, for instance, you can specify a new point that has the "X" value of the endpoint of a line, the "Y" value of a circle center point, and a "Z" value that you type in.

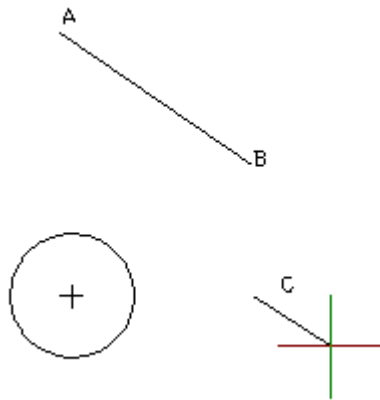


Coordinate filters are available for use on the Left Side Bar using these command buttons.

There is both an x,y filter button used for 2-dimensional drawing, as well as an x,y,z button that allows you to specify a new 3-dimensional point. You can use coordinate point filters with any command that prompts you to specify a point in the drawing, by selecting the appropriate filter button immediately after being prompted to select a point. After doing this, you will be prompted in turn to enter an x,y (or x,y,z) coordinates for the new point.

An example of the use of coordinate filters follows:





Command inputs:

IE. > LINE (Here, you initiate the LINE command)

From point: x,y (Here you pick the x,y filter button when prompted for the first point)

X-Coordinate: endp (You are prompted for X and you specify the endpoint osnap)

of object: (Here, you select the endpoint "B" of the line)

Y-Coordinate: cen (You are then prompted for Y and you specify the center osnap)

of object: (Here you select the circle whose center has the Y value that you want – this establishes point "C", the first point in your line)

To point: (You are then prompted for the second point in the line. You can again use point filters as necessary.)

#### Notes:

1. You can simply type in an x,y or z value at any coordinate prompt, rather than using precision aids.
2. You can use the ELEVATION variable to set a standard z value to new points being drawn.

## Operational Modes

### Orthogonal Mode

#### Orthogonal Mode



Orthogonal Mode limits the movement of the cursor so that it will only move parallel to the axes of the current coordinate system. This makes it much easier to design and place parallel, or "running", lines or objects.

In the standard World Coordinate System (WCS) the orthogonal lines will run horizontally or vertically from the base point.

If Orthogonal Mode is activated, it is only possible to indicate points that are located on a line running parallel to one of the axes of the coordinate system. Such a line is also called an "orthogonal" line.

**Note:** Orthogonal Mode is effective only when "picking" points using the Cursor. Input of non-orthogonal

points is possible by keyboard input while the orthogonal mode is activated.

### Toggle Orthogonal Mode



**Command:** TORTH  
**Alias** TO

#### Located:

1. **Pull-down Menu:** Tools – Set Drawing Modes – Orthogonal Mode
2. **Properties Bar**
3. **F-8 Function Key** (by default)
4. **Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Orthogonal Mode ON / OFF. When Ortho mode is on, lines that you draw will be forced parallel to the axes (IE straight up and down or sideways).

## Object Snaps

### Object Snap

Many functions and commands require the definition of an exact position in the drawing. The determination of exact points can be done either by coordinate input or by indicating points. The input of coordinates has the advantage of precise placement, whereas indicating points often has an advantage in speed and ease of handling.

A very efficient method to take advantage of both techniques is to indicate points with precise references to a given geometry with the help of the Object Snap functions. The individual modes of Object Snap functions can be called for any single selection operation from the proper palette or via the cursor menu (middle mouse button) or be activated (permanently) using the *Precision Aids* dialog box (PRECPAR command).

Object Snapping allows geometrically significant points of existing drawing entities to be found. These points are then used as inputs for a new drawing step.

The Object Snap functions can be activated permanently or be called for a single operation. These techniques are described in the paragraph titled "Controlling the Object Snap Functions".

#### See also:

Mode of Operation

Object Snap Modes in Detail

### Command OSNAP

**Command:** OSNAP

**Alias** OS

#### Located:

1. **Left SideBar (individual icons)**
2. **Entered Directly at the Command Line**

For precise drafting, many commands require that you select an exact position in the drawing. You can pick the precise location by typing the coordinates, or by picking a point in conjunction with an object snap. Object snap allows you to pick geometrically significant points on existing entities.

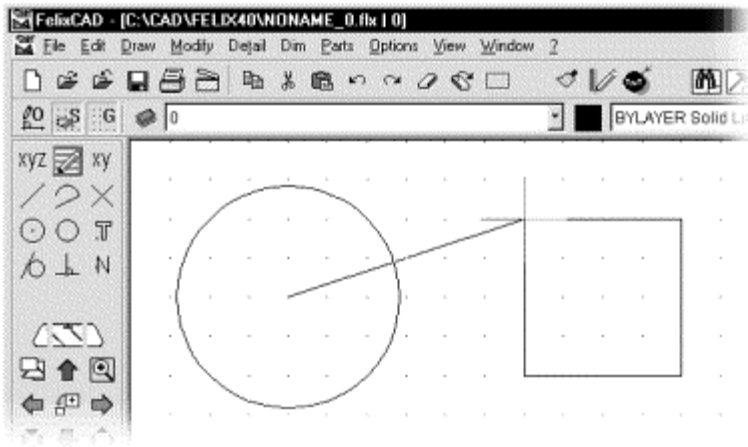
Object snap mode can be activated permanently or called for a single pick. The OSNAP command lets you turn on object snap modes; transparent commands - such as mid and qua -- temporarily activate an object snap mode.

The procedure for using the object snaps is always the same:

1. Start a command that requires a point, such as "From point:"
2. When the point is requested, reply with an object snap mode; the command prompts for the point with "of".
3. Select the object with the cursor.

Here is an example of using object snap. A line is drawn from the center (cen) of the circle to the corner (int) of the rectangle:

```
> line  
From point: cen  
of [pick]  
To point: int  
of [pick]  
To point: [Enter]
```



### Drawing a line using object snap modes

Some object snap modes make no sense. For example, you cannot define the end point of a circle (use the qua mode instead), nor can you pick the center of a line segment (use the mid mode instead).

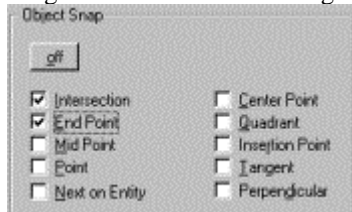
On the other hand, you can use several osnap modes on entities. For example, a line can use the end mode (twice, one for each end), mid mode, and per (perpendicular) mode. An arc can use end mode (again, twice), mid, cen, per, qua (quadrant points) and tan (tangent) modes.

The end object snap mode does not differentiate between starting or ending point of an entity; you must be careful with your pick point. The object snap selects the nearest of the two possible points.

The illustration below shows the connection between the point selected and the point snapped:

### Running Object Snap

Using the Precision Aids dialog box, object snap modes are turned on. This is called running mode.



Click the checkboxes for the desired mode(s). A check mark indicates the mode is active. Clicking the off button deactivates all running snap modes.

When several object snap modes are on, you differentiate between them by the way you select an entity. For example, End Point and Intersection are on; pick a point nearest an end point, then the end is selected. If you pick a point near the intersection of two entities, then the intersection is selected. Properly

configured, this feature can speed the drawing of entities significantly.

### Clearing Object Snaps

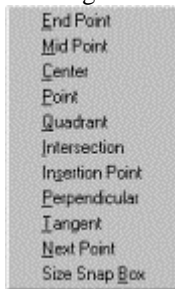
Object snaps can all be switched off by either pressing the "OFF" button on the Osnap dialog, or clicking on the "No" icon on the Snap Palette. Individual Osnaps can be deselected on the Osnap dialog.

### Temporary Object Snap

You can choose between several options for a one-time activation of an object snap mode.

You can use the Snap Palette to activate an object snap mode just one time during a command.

An effective method of calling an object snap mode is to bring up the cursor menu. Press the middle mouse button (or hold down the Ctrl key and press the left mouse button on a two-button mouse). The following menu is displayed, from which you select an object snap mode:



Use the keyboard to type a command abbreviation. The object snap modes and their abbreviations are shown in this table:

Object Snap Mode	Abbreviation
End point	end
Mid point	mid
Center point	cen
Quadrant	qua
Intersection	int
Implied Intersection	Imp
Perpendicular to	Per
Tangent to	Tan
Next point on entity	Nxt
Insertion point	Ins
Point entity	Pnt

### Object Snap Modes in Detail

There is a difference between object selection and object snap modes. Object selection treats a group of entities, such as a rectangle, chain, and polyline, as a single object. In contrast, object snap modes treat these as individual entities. For example, you can snap to each segment and vertex of a polyline. This is also true for polygons and 2D and 3D faces.; the edges are interpreted as by object snap as individual lines. In this way, the object snap modes can be used for the edges of planes as well as for segments of chains and polylines.

For simplicity, when this text refers, for example, to the "end point of a line" the term also refers to polyline segments and the edge of a plane.



#### ENDpoint

The End object snap snaps to the endpoint of a line or of an arc. The endpoints of planes and faces can be accessed.



#### MIDpoint

The Mid mode snaps to the center of a line or an arc. Te edges of faces may also be snapped.

**CENterpoint**

The Cen mode snaps to the center point of a circle, arc, and polyarc.

**QUAdrant**

The Qua mode snaps to the nearest quadrant of a circle or an arc. Quadrant refers to the points situated on the circumference at 0, 90, 180 and 270 degrees. Only the next visible quadrant of an arc can be snapped.

**INTersection**

The Int mode snaps to the intersections of a pair of lines, arcs, and other entities -- or any other combination of them. This includes dashed and centerlines where the intersection point may not have line segments actually touching.

**IMPLied**

The Int object snap mode only returns an intersection point of entities that have a real intersection. In contrast, the Imp mode snaps -- in addition -- to an implied (intersection point. This is the point where the extension of two entities would intersect.

**PERpendicular**

The Per mode snaps to the perpendicular to an entity. This mode requires that a starting point already been selected. For example, when a perpendicular is desired, the ending point for the perpendicular is picked. The result is a perpendicular line from the ending point to the chosen entity.

**TANgent**

The Tan mode snaps to the tangent point of a curve. This mode requires that a starting point has already been selected. For example, when a tangent to a circle is desired, the ending point for the tangent line is picked. The result is a tangent line from the ending point to the chosen entity.

**NeXT**

The Nxt mode snaps to the next situated point of an object when at least one point of this object lies within the borders of the selecting cursor box. This ensures that entities that are designed or modified by use of the object snap mode share a point with the selected object.

(If you select a point in 3D space without using this object snap mode, it is possible that the selected point is not on the desired object. The two objects would not have a common point, or a closure.)

**INSertion**

The Ins mode snaps to the insertion point of a part, an attribute, or a text object.

**PoiNT**

The Pnt mode snaps to a point entity, which was created by the Point comm

## Object Snap: Mode of Operation

The object snap functions cannot be called separately, but may be used only with a command, that requests the input of a point.

The operating principal when using object snap functions is always the same:

1. Call a command or a function, that requires a point;
2. When the point is requested, answer by calling an individual object snap function or a special

object snap mode. The cursor then takes the form of a small box. The command line area displays a message, that requests the point.

3. Select the object with the cursor.

The geometrically significant point that is "picked" during the activity depends on:

1. The type of the drawing entity;
2. The respective object snap mode; and
3. The specific point of the object that is chosen.

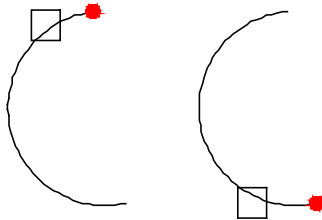
Object snap modes are occasionally not applicable. For instance, the user cannot define the ending point of a circle and while a line segment has no central point.

On the other hand, it is possible that a drawing entity contains several points to which the criteria of the object snap function correspond. A line entity or an arc, for instance, always has two ending points.

**Note:** The object snap function *Endpoint* does not differentiate in regard to the starting or ending point of an object.

An object snap function identifies, in those cases, the nearest of two or more eligible points. When selecting an object by positioning the cursor, it determines which of the possible points will be snapped to. The user should always place the cursor as close to the desired point on the object as possible.

The illustration below shows the connection between the point selected and the point snapped:



**See also:**

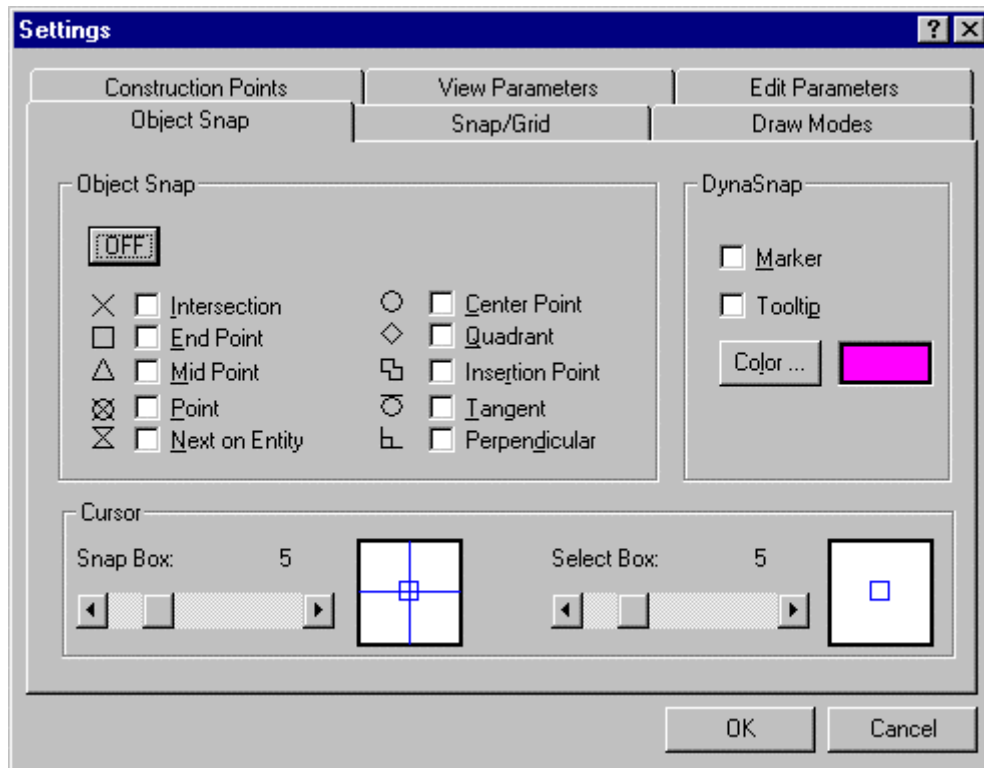
Controlling the Object Snap Functions

Object Snap Modes in Detail

## Controlling the Object Snap Functions

### Object Snap: Permanent Activation

Using the Object Snap tab on the **SETTINGS** command dialog, individual modes of the Object Snap functionality can be permanently activated.



As can be seen, individual snaps can be made active, either singly or in combination with other object snaps, simply by selecting the check box beside the snap. To the left of the check box, you will notice the symbol that appears onscreen when Dynasnap markers have been switched on (in the Dynasnap section to the right) and when the snap has been activated by the proximity of the cursor to the snap point. Pressing the OFF button will de-select all selected snaps, at once.

As mentioned, Dynasnaps and Tooltips (pop-up identification messages) can be switched on or off in the Dynasnap section on the right side of this dialog. Color for both can also be changed in this section, as well.

At the bottom of the dialog, you will find controls that designate the size (in pixels) for both the cursor snap and select boxes.

### **Implicit Object Snap: Activating from Case to Case**

For the single use activation of a certain mode of the Object Snap function you can choose between several options:

#### **Snap Palette**

The user may use the Snap Palette to call individual object snap modes for singular use in combination with a designing or editing command. This standard palette can be opened when needed and be placed anywhere on the drawing area. The user may now call the object selecting modes by cursor selection. The desired object snap modes can also be included in user-defined palettes with the help of the menu and dialog editor.

#### **Cursor Menu**

Another very effective method of calling an individual object snap mode is by activating the Cursor Menu using the middle mouse button.

#### **Keyboard Input**

Keyboard input of a command abbreviation is another method of activating the object snap functions. The object snap functions and their abbreviations are as follows:

Object Snap Mode	Object Snap Input Abbreviation
End	end
Mid	mid
Center	cen
Quadrant	qua
Intersection	int
Implied Intersection	imp
Perpendicular	per
Tangent	tan
Next	nxt
Insertion point	ins
Point entity	pnt

See also:

Object Snap Mode of Operation

Object Snap Modes in Detail

### Object Snap Mode - End



#### END Endpoint

The Object Snap function "End" snaps to the end point of a line or of an arc.

Planes or the edges of solids are also interpreted as lines, and end points of planes and solids can be trapped by the mode "end point".

### Object Snap Mode - Mid



#### MID Point

Snaps to the center of a line or an arc.

Planes, or the edges of solids, are also interpreted as lines, and may be snapped by the mode "mid-point".

### Object Snap Mode - Center



#### CEN - Center Point

Snaps to the center of a circle or an arc.

### Object Snap Mode - Quadrant



#### QUA - Quadrant

Snaps to the nearest quadrant of a circle or an arc.

Quadrants refer to the points of a circle or an arc, situated on the circumference at 0, 90, 180 and 270 degrees. These are the intersecting points of the axes of a coordinate system upon whose origin lies at the



center of the circle or the arc with the circumference of this circle or arc. Only the next visible quadrant of an arc can be snapped.

NOTE: For further operations, the command ROTATE will not rotate the quadrants with the object.

### Object Snap Mode - Intersection



#### **INT - Intersection Point**

Snaps to the intersection points of lines, arcs, circles, or any other combination of them.

### Object Snap Mode - Implied Intersection

#### **IMP – Implied Intersection**

This is a newly-added object snap that is at present available only as a keyed-in temporary object snap. Using the IMP option will snap the cursor to the theoretical intersection of two converging linear objects.

### Object Snap Mode - Perpendicular



#### **PER - Perpendicular to...**

This mode requires that a starting point for a function has already been selected. Activating the Object Snap mode will relate this point to the snapped point.

For instance, when a perpendicular is desired, the ending point for the perpendicular is picked, and the Object Snap mode Perpendicular is chosen. The result will be a perpendicular line from the ending point to the chosen entity.

### Object Snap Mode - Tangent



#### **TAN - Tangent**

Similarly, this mode requires that a starting point for a function has already been selected. Activating the OBJECT SNAP mode will relate this point to the snapped point.

For instance, when a tangent to a circle is desired, the ending point for the tangent line is picked, and the Object Snap mode Tangent is chosen. The result will be a tangent line from the ending point to the chosen entity.

### Object Snap Mode - Next



#### **NXT - Next Point on Entity**

This mode snaps to the next situated point of an object, when at least one point of this object lies within

the borders of the selecting cursor window. This ensures that entities that are designed or modified by use of the OBJECT SNAP function share actually a point with the selected object. If the user selects a point on the space without using this OBJECT SNAP function, it is possible that the selected point is not on the desired object. The two objects would not have a common point, or a "closure".

### Object Snap Mode - Insertion Point



#### **INS - Insertion Point**

Snap to the insertion point of a part insertion, an attribute or a text object.

### Object Snap Mode - Point

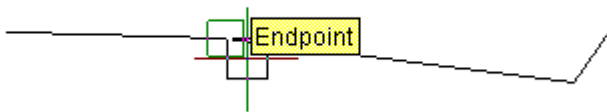


#### **PNT- Point**

Snap to a point that was created by the drawing command POINT.

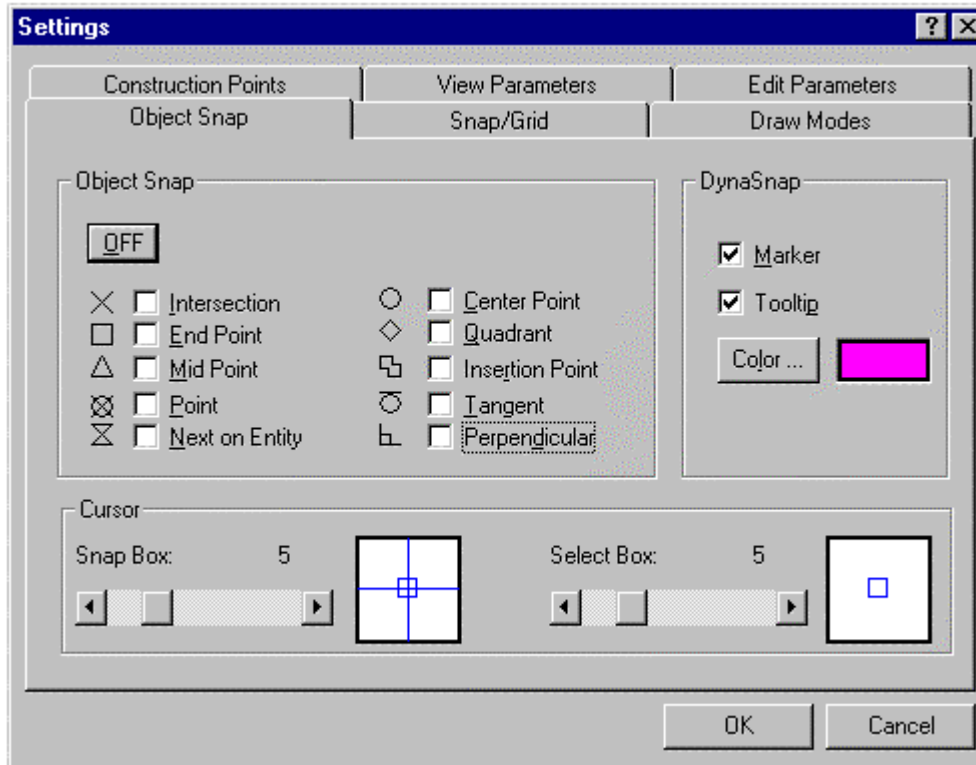
### Dynasnap

AllenCad's "Dynasnap" (Dynamic Snap) capability works directly with currently active object snaps to indicate which snap will be applied if selection is made at the current cursor location. Dynasnap does this by displaying the symbol, a tooltip or both symbol and tooltip for the controlling Osnap next to the cursor. The user can configure which dynasnap will display. If several Osnaps have been activated, their various symbols and tooltips will appear and disappear as the cursor is moved around on the drawing and as snaps are activated and deactivated by the cursor's proximity to snap points. In the graphic below, we can see that the endpoint of the line segment has fallen within the cursor snap box and that both a green square identifying the active snap and its location, as well as a tooltip indicate the currently active snap.



Note that dynasnap are displayed only while commands that require point selection are active. Only one snap can be in control at any given cursor and is generally determined by the closest snap point to the cursor. The exception to this rule is the Center Osnap, where a section of the circle itself must be within the cursor snap box. When more than one snap point is within range, the closest point will take precedence. As the cursor moves closer to another snap point, the symbol and or tooltip will change to indicate that the second snap has taken over from the first.











Dynasnap is activated and configured in the settings dialog under the Object Snap tab. The OSNAP command will take the user to this dialog directly. As can be seen on the graphic below, the choice of Marker or Tooltip can be made by clicking on the check box in the dynasnap section of the dialog. The color of the marker can also be selected by clicking on the color button.



Across in the Object Snap section of the dialog, the dynasnap symbols can be seen adjacent to their pick boxes. Below that, in the Cursor section of the dialog, the snap box slider controls the snap box size. When snap points come within this box, they become active contenders for selection.

### Object Snap Modes:

Turns on running object snap:

Mark er	Mode	Supported Entities
	Intersection	Snap to the intersection of an arc, line or <u>polyline</u> segment.
	End Point	Snap to the closest endpoint of an arc, line or <u>polyline</u> segment, solid, or 3dface.
	Mid Point	Snap to the midpoint of an arc, line or <u>polyline</u> segment, solid, or 3dface.
	Point	Snap to a point object.
	Next on Entity	Snap to the nearest point on an arc, circle, line, point, or <u>polyline</u> segment.
	Center Point	Snap to the center of an arc, circle, or <u>polyline</u> arc segment.
	Quadrant	Snap to the quadrant point of an arc, circle, or <u>polyline</u> arc segment.
	Insertion Point	Snap to the insertion point of an attribute definitions, part/block, or text.
	Tangent	Snap to the tangent of an arc, circle, or <u>polyline</u> arc segment.
	Perpendicular	Snap to a point perpendicular to an arc, circle, <u>polyline</u> segment, line, solid, or 3dface.

## SNAPBOX

This variable (Snap Cursor Box) sets the size of the snap area in pixels for object snap functions.

### Properties

<i>Type</i>	Integer
<i>Default</i>	5
<i>Status</i>	Read / Write
<i>Saved</i>	In the CFG file

### Commands

The transparent command 'PRECPAR allows the user to specify an appropriate value for his current needs.

## Grid Mode

### Snap Grid



Snap Grid (located on the Properties Bar) covers the drawing area with an invisible grid. With *Snap Grid* activated, the cursor can only select points that are positioned directly on the grid. For the user, this means that starting- and ending-point, center, and any other specific point that can be defined will always be exactly on a grid point.

The distance between grid points, as well as the position of its zero point, can be selected by the user. The grid will always follow the axes of the current coordinate system.

## GRIDUNIT

Specifies the horizontal and vertical distances (x, y) between the grid dots when grid display is turned on

(see system variable GRIDMODE).

#### Properties

<i>Type</i>	2D point
<i>Default</i>	0.0,0.0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Command

'PRECPAR

#### See also

System variable GRIDMODE

### Grid Display



Grid Display (located on the Properties Bar) covers the drawing area with a visible latticed point grid and offers a visual impression of the distances, angles and other relations of the objects with themselves.

## GRIDMODE

Turns grid display on or off. The dots of the grid are displayed in the XY distances as specified by the system variable GRIDUNIT and in the area specified by the system variables LIMMIN and LIMMAX.

#### Values

0	Off
1	On

#### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Command

'PRECPAR

### SNAPBASE

Determines the origin (x,y) for the snap grid.

#### Properties

<i>Status</i>	Read / Write
<i>Type</i>	2D Point
<i>Default</i>	0.00,0.00

#### Command

'PRECPAR

## SNAPBOX

This variable (Snap Cursor Box) sets the size of the snap area in pixels for object snap functions.

#### Properties

<i>Type</i>	Integer
<i>Default</i>	5
<i>Status</i>	Read / Write
<i>Saved</i>	In the CFG file

#### Commands

The transparent command 'PRECPAR allows the user to specify an appropriate value for his current

needs.

## SNAPISOPAIR

Sets the current isometric plane for the isometric drawing style. Has effect only if SNAPSTYL is set to isometric (see below).

### Values

0	Sets the crosshairs to the left plane (default)
1	Sets the crosshairs to the top plane
2	Sets the crosshairs to the right plane

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1   2
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

[PRECPAR](#) 'SETTINGS > Snap/Grid tab.

### See also

System variables [SNAPSTYL](#), [SNAPMODE](#), [SNAPBASE](#), [GRIDMODE](#)

## SNAPUNIT

Determines the horizontal and vertical distance of the points on the snap grid.

### Properties

<i>Type</i>	2D Point
<i>Default</i>	1.00, 1.00
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Command

[PRECPAR](#)

## SNAPSTYL

Toggles the snap style for the current viewport between standard and isometric drawing style. The style setting can be:

### Values

0	Standard (default)
1	Isometric

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

[PRECPAR](#) [SETTINGS > Snap/Grid tab.](#)

### See also

System variables SNAPISOPAIR, SNAPMODE, SNAPUNIT, SNAPBASE, GRIDMODE

## Modes Controlled by Systems Variables

## PICKFIRST

With the global variable PICKFIRST system variable, you specify whether you wish to select objects before or after you issue a command that uses the object selection set. Values of zero and one are valid for this variable.

- 0 - Turns off Variable PICKFIRST; you select objects after you issue a command
- 1 - Turns on Variable PICKFIRST; you select objects before you issue a command

## PLINETYPE

This system variable controls polyline format, both in the creation of new polylines with the PLINE command, as well as conversion of existing polylines. Both standard format and lightweight (LWPOLYLINE) format polylines are supported. Valid settings for this variable are as follows:

- 0 Polylines in older drawings are not converted when opened; PLINE creates old format polylines
- 1 Polylines in older drawings are not converted when opened; PLINE creates optimized polylines
- 2 Polylines in older drawings are converted when opened; PLINE creates optimized polylines

This command is entered at the command line and the program will prompt with the currently set value for the variable. You may enter a new value or accept the current value by simply pressing the Enter key.

## Deleting and Undeleting

### Delete



**Command: DELETE**

**Alias DEL**

#### **Located:**

- 1. Pull-down Menu: Edit – Delete**
- 2. Top Function Bar**
- 3. Palette: Edit**
- 4. Or Enter Command or Alias at the Command Line**

During the creation of drawings, or, later during the editing, changing or detailing of drawings, it may be necessary to delete certain parts of the drawing.

After calling the command, the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for object selection. These same options will alternately appear in a dialog menu, depending upon your preferences specified in Desktop Configuration.

WPolygon	CPolygon	Fence	Last	Previous	Remove	Window	Crossing	Single	All
----------	----------	-------	------	----------	--------	--------	----------	--------	-----

When deleting, as many objects can be selected as the user desires. When the object selection is complete, the selected entities will be deleted without another confirmation, by simply pressing ENTER.

---

**See also:**

Command UNDELETE

Undelete: Withdraw Deletions



**Command: UNDELETE**

**Alias UD**

**Located:**

1. **Pull-down Menu: Edit – Undelete**
2. **Top Function Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

Entities that have been deleted by mistake can be restored by the command UNDELETE.

Call the command

- By clicking the UNDELETE symbol;
- By selecting the option **Undelete** in the menu *Edit*; or
- By keyboarding UNDELETE in the command line area.

This will return objects deleted by the last command that removed objects from the drawing.

---

**See also:**

Command DELETE

**Undo Control**

**Command: UNDO**

**Alias U**

**Located:**

1. **Pull-down Menu: Edit - Undo**
2. **Or Enter Command or Alias at the Command Line**

The program archives all steps made during the task in a list called the Undo List. The steps registered in this list can be withdrawn by using the UNDO function sequentially, each UNDO removing the next sequential step in the task.

Certain commands are not registered in the Undo List. These are commands and functions that control the viewport (Zoom, Pan, Window), set the parameters, and other transparent functions. These steps cannot be withdrawn by using UNDO.

The Undo List is kept only with the current session of the program. It will not be saved with the drawing.

The command can be repeated until all working steps stored in the Undo List have been canceled. The program will then send out the message "Undo List is empty".

**Options**



Canceling the last work step (the first step appearing in the Undo List) is the normal mode of this function. The function can be altered to perform groups of UNDO steps as well, by following this procedure:

- Enter UNDO in the Command Line area.
- Define the number of steps to cancel by entering a number; or
- Select an option from the option bar.

## Control

The option field "Control" allows access to these options for controlling the UNDO process:

### All

By selecting this switch you indicate that all (chosen) working steps are selected for the UNDO function.

### None

No steps are registered in the UNDO LIST. The command is switched off.

### Once

This option fixes the number of the steps you can cancel at the value 1. Only the last working step is registered in the UNDO LIST.

### Erase

Selecting this switch causes erasure of the current UNDO LIST.

### Comment

This option is a Switch that will trigger a prompt asking for the confirmation of the user's intentions. After selecting this field the option bar will show the possibilities ON and OFF. The active status will always be indicated as the preset value. The user may change this value, or go with it.

The options *Group-B*, *Group-E* and *Group-D* allow the user to select groups of steps. A number of defined procedures can be canceled with a single UNDO.

## Group-B

This switch defines the **Beginning** of the selected group. This "beginning" may be canceled, and any previous registrations in the Undo List remain in the list.

## Group-E

This option defines the **End** the group of working steps.

## Group-D

This option specifies that the group of steps selected for UNDO is **Deleted**. The individual steps remain in the UNDO LIST and can be canceled one by one.

The two following options, **Mark** and **Back**, allow the user to "mark" the listed steps of a task, and cancel all unmarked steps.

## Mark

This option defines the current state of the drawing as a state that can then be restored by the option **Back**.

## Back

This option restores the task to the previously marked state.

---

## See also:

- Command U
- Command REDO

## Editing

## Modify Object Properties...



Command: PROEDIT

Alias PED

### Located:

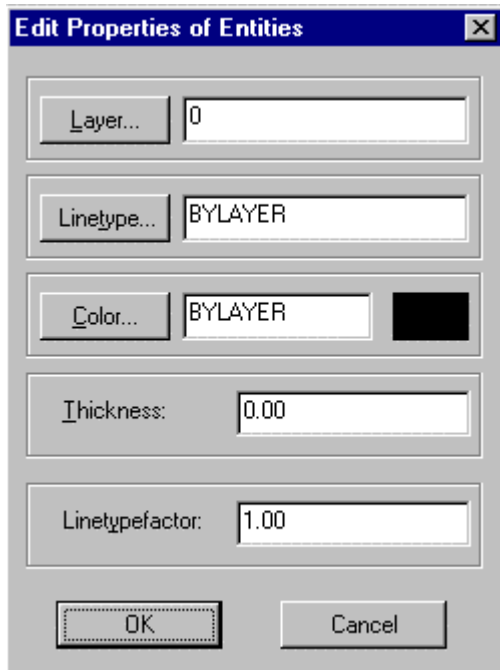
1. Pull-down Menu: Edit – Modify Object Properties
2. Left Side Bar
3. Palette: Edit
4. Or Enter Command or Alias at the Command Line

The function **Modify Properties...** invokes the dialog boxes that allow you to set the properties of entities, and make changes in **Color**, **Layer**, and **Linetype**.



As with the other commands in this set, first select one or more objects, using the selection dialog shown above, if required.

Then press RETURN, or use the right-hand cursor button.



The dialog box that appears displays the current properties Layer, Linetype, Color, Thickness and Linetype Factor (also called Linetype Scale Factor ) of the entities selected. You may then edit these properties.

## Change Properties (Command Line Version)

**Command:** CHPROP , QPROPEDIT

**Alias** CHP, QPR

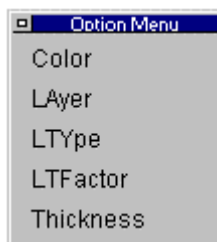
**Located:**

### 1. Entered Directly at the Command Line

These commands modify the properties of drawing objects.

They are the command line variants of the PROPEDIT command.

When you issue the command, you will be asked to select objects, and be presented with the object selection dialog to help you make this selection. Next, you will be prompted at the command line to identify which properties you want to change. A dialog appears (as below) on screen to help you enter your selection.



After making your selection, you will be asked to enter a new value for the property you want changed.

As with most commands, command line input can be used, instead of using the dialog. You simply type in the option abbreviations as are shown bold in the following text screen recording. The requests of these commands are as follows:

> CHPROP

Select objects:  
Specify property to change (Color, LAYER, LType or Thickness):

---

See also:

PROPEdit

## Modify Text...



Command: TEXTEDIT

Alias ED

### Located:

1. Pull-down Menu: Detail – Modify Text
2. Left SideBar
3. Palette: Detail
4. Or Enter Command or Alias Directly at the Command Line

You can modify an existing text line in the drawing by selecting **Modify Text...** from the menu *Detail*.

Select a text entity in the drawing. The dialog box *Text Edit* is displayed that is similar to the dialog to create text (with exemption that only one line can be edited).

Simply edit the existing text in the window meet the requirements. The dialog box also presents the parameters attached to the selected text. These parameters can be modified as required, for the replacement text.

If the check box labeled *Interactive* is marked, you will be prompted to relocate the text when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

> TEXTEDIT

Select a text:  
<Dialog box appears to edit the selected text line>  
Insertion point:

---

**See also:**

TEXT  
RTEXT  
FONT

## **Command MTEXTEDIT**

### **Command: MTEXTEDIT**



Alias MTE

#### **Located:**

1. **Pull-down Menu: Detail – Edit Multiline Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

This command predictably is used for editing multi-line text. When the command is issued, you are prompted to select the multi-line text that requires editing. This command will only work on text that has been entered with the MTEXT command. You must use TCORRECT or TEXTEDIT on other text entities.

When you select an MTEXT entity, the full multi-line entity is brought into the MTEXT dialog window, where you can edit any of the parameters that were originally available when the text was first created.

## **Array**

### **Command: ARRAY**

Alias RA

#### **Located:**

1. **Pull-down Menu: Modify - Array**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Array** represents a variant of Copy, except that Array will create multiple copies, and they can be arranged at random, or in rows, or in vertical order, or on a circle, or on an arc.

The entities created by the command are exact copies of the selected entities in which all entity attributes such as layers, line types and color are copied.

After entering the command, select an option from the Option Bar:

Individual  
Rectangular  
Circ-NRot  
Circ-Rot

Options



### Individual Array

This mode allows the user to copy the Array as many times as is required during one command sequence and place them at random on the drawing area.

The use of this option is very similar to the command Copy.

After choosing the option, select the entities to be copied using the selection modes described in the Option Bar. Always confirm object selection by pressing RETURN.

Determine the base point for the copying. A particularly discreet point in the geometry or in one of the selected objects (edges, corners, intersecting points etc.) will be useful. Next, either input the coordinates of the target point(s) or "pick" the points for placement of the Array. The program will create a copy of the Array after each definition of a target point.

As long as the command remains active the user can produce and place as many copies as are required. Terminate the function by pressing ESC or RETURN.



### Rectangular Array

This option will arrange the Array copies in rows or columns. The use of this function is the same as was the use of the **Individual** mode.

Determine the number of the columns and rows in which the Array copies should be arranged. Define the distance between the columns and the rows.

Note that, when planning the arrangement, the rows and columns must be in the positive X or Y-axis. This infers that the original Array is situated in the lower left-hand corner of the quadrant plus-XY. If it is not located there, it may be useful to move the original Array before attempting to begin defining the copying positions. Use the command Move to do this.

### Circular Arrays

The Array can also be reproduced in a circular path. The circle (arc) itself is not displayed on the screen. The reproduction of the Array is described as follows:

1. The center of the circle (arc)
2. The number of times the Array is to be reproduced;
3. The included angle of the arc (360 if the path is to be a full circle) to be filled;

There are two ways to reproduce the Array on a circular path:



### Placing the Arrays Without Rotation

This Option, will arrange the reproductions on the circle (arc) in the same alignment to the principal axes as the original Array.

Define the center of the circular path. Then determine **either**;

The number of the entities to be arrayed (including the original); and  
the included angle of the circle (arc),

or

The included angle of the circle (arc); and  
the included angle between the Arrays to be reproduced (measured **between** the reference points of the objects).

Note that, during the planning of the reproduction of the Array on a circle that the circle (arc) runs **counter-clockwise** from the starting point. For a **clockwise rotation** enter negative values for the angle inputs.



### Placing the Arrays With Rotation

This option will rotate the Arrays when placing them on the circular path. The Arrays will be rotated in the same direction as the circle arc that describes the path of reproduction. The rotation angle will correspond to the angle between the Arrays (measured between the reference points of the Arrays).

After selecting the command, choosing the objects in the Array, and choosing the option **Ci.Rot** from the Option Bar, define the center of the circular path and then determine **either**;

the number of the entities to be arrayed (including the original); and  
the included angle of the circular arc,

or

the included angle of the circular arc; and  
the included angle between the entities to be arrayed (measured between the reference points of the objects).

As parameter value for the Arc angle there is always used the standard value 360 degree, i.e. the copies are arrayed on a full circle, if this value has been confirmed.

Please note during the planning of the arraying on a circle of multiple copies that the circle (arc) runs from the starting point that is the original object anti-clockwise. For a clockwise rotation please enter negative values for the angle inputs.

## Copy Object Properties



**Command: COPYPROP**

**Alias CP**

### Located:

1. **Pull-down Menu: Edit – Copy Object Properties**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

This command helps you to change the properties (layer, linetype and color) of selected objects to those of a selected entity.

### To change the properties of a selection:

1. Select a single reference entity.
2. Select one or more objects to be changed.

### Command line example:

```
> COPYPROP  
Select entity for reference properties:  
Select objects to modify. Select objects:
```

## Command COPYSELECT



**Command: COPYSELECT**

**Alias CST**

### Located:

1. **Pull-down Menu: Edit – Copy Selection to Clipboard**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

The command COPYSELECT copies selected objects from the drawing, placing them onto a **AllenCad-specific** clipboard. The selected objects remain in the drawing.

The command COPYSELECT is useful especially to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

**To copy objects onto the clipboard:**

1. First specify the base point for later insertion of the objects.
2. Then select the objects to be copied to the clipboard.

**Command line example:**

```
> COPYSELECT  
Insertion base point:  
Select objects:
```

**Note:**

Do not confuse this command with the COPY command found in the *Modify* menu that makes copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another drawing. They will remain on the clipboard until you place new objects on it with the COPYSELECT command.

---

**See also:**

Command PASTE  
Command CUTSELECT

## Paste Selection from Clipboard



**Command: PASTE**

**Alias PST**

**Located:**

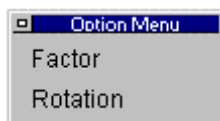
1. **Pull-down Menu: Edit – Paste Selection From Clipboard**
2. **Top Function Bar**
3. **Palette: Edit and Modify**
4. **Or Enter Command or Alias at the Command Line**

The PASTE command copies objects from the **AllenCad-specific** clipboard to point you select in the current drawing. These objects have typically placed on the clipboard by a preceding COPYSELECT or CUTSELECT command.

Use the command to copy graphic data from other drawings into your current drawing.

**To paste the objects from the clipboard into the current drawing:**

1. Choose the PASTE command.



2. You are presented with the option dialog. If you plan to scale or rotate your



objects during insertion, select your option now. You will be offered the opportunity to scale and rotate as a conclusion to the command later if you do not choose at this point, however you will not be able to visualize the scaled or rotated objects in advance, if you wait until later.

3. Specify the insertion point by entering coordinates of the insertion point, OR
4. Drag the highlighted objects into position on the drawing. The objects of the clipboard contents appear highlighted (as well as scaled and/or rotated - if you selected this in the last step), and attached to the cursor during the exercise. This feature allows you to visualize object placement prior to selecting the insertion point with a mouse click.
5. Specify the scale factor for insertion – you can specify a scale factor at this point (if you have not done so already) and have the objects inserted at a larger or smaller scale than they were originally. Note that the relative position of these objects to each other remains the same (is scaled too) during the operation).
6. Specify the rotation angle (if you have not done so already) – you can choose to rotate the object by any input angle during the exercise.

**Example:**

```
> PASTE
Insertion point:
Scale factor <1>:
Rotation angle <0>:
```

When inserting the objects from the clipboard you can also specify the scale factor and/or rotation angle before you indicate the insertion point by choosing the appropriate option from the context bar when prompted for the insertion point:

```
> PASTE
Insertion point: Factor
Scale factor <1>: .5
Insertion point: Rotation
Rotation angle <0>: 90
Insertion point:
```

**Note:**

The objects selected with COPYSELECT remain on the clipboard, ready to be placed in another drawing, until you place new objects on it with the COPYSELECT command.

To copy objects within the current drawing it is recommended that you use the COPY or ARRAY command found in the *Modify* menu.

---

**See also**

Command COPYSELECT

Command CUTSELECT

Mirror



**Command: MIRROR**

**Alias Mi**

**Located:**

1. **Pull-down Menu: Modify - Mirror**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Mirroring drawing objects produces identical copies of selected entities that are placed in a "mirrored" position about an axis specified by the user.

The selected objects have, after "mirroring", the same relative position to the symmetry axis as they had prior to " mirroring".

The mirror axis can be defined freely, or parallel to the X-axis or the Y-axis

In order to use the command the user must select an object to be mirrored. Select the object or objects, remembering that the selection of multiple objects will result in the treatment of the objects as one.

The next prompt will ask for the definition of the mirroring axis.

### Options:

#### Mirroring With a Free Axis

The Option **Axis** allows the user to freely define the mirror axis. It is the standard option for mirroring objects and is used when no other option has been selected.

Select this option, then define the position of the mirror axis. This can be made by indicating two points or by inputting their coordinates. The mirror axis can have any alignment with regard to the objects to be mirrored as well as with regard to the axes of the coordinate system.

If the mirror axis should run through points of an existing geometry (for instance, along a solid edge), the user should employ the object snap options for precise definition.

#### Mirroring on a Horizontal Axis

To choose a horizontal-mirroring axis, select the option **Horizontal**. This option creates a mirror axis that runs parallel to the Y-axis of the coordinate system. Its exact position is indicated by a single point.

#### Mirroring on a Vertical Axis

To choose a vertical-mirroring axis, select the option **Vertical**. This option creates a mirror axis that runs parallel to the X-axis of the coordinate system. Its exact position is indicated by a single point.

#### Note: Mirroring Text

If there is text associated with the selected object or objects, it is likely that the user does not want to mirror that text. In order to mirror the text it is necessary to activate an option called **Text Mirroring**. Activating this option will result in the text also being mirrored.

#### Offset



**Command: OFFSET**

**Alias O**

#### Located:

1. **Pull-down Menu: Modify – Offset Through Point**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Offset** creates congruent pictures of lines, circles, arcs and 2D-polylines. This command enables a copy of a selected entity to be created and placed a distance from the original entity. The original entity(s) stays in position.

The operating mode of the command depends on the type of drawing entity:

- For lines and polylines identical copies are created and moved in the direction of the X or Y-axis.
- For circles and arcs copies are also created, but, in addition, they will be scaled by a selected

factor. As a base point for scaling the center of the original circle or arc is used. In this way, concentric circles or arcs are created.

## Options

### Offset for Lines/ 2D-Polylines

After entering the command, select the line(s) or polyline(s) that are to be offset by "picking" with the cursor. Then you either select the distance for the offset by picking, or select **Distance** from the Option Bar, and enter the offset distance. Then answer the prompt "Side to Offset?" with a point on the desired side.

### Offset for Concentric Circles/ Arcs

Concentric circles or arcs can be created with a radius or diameter smaller or larger than the original entity.

After having selected the objects to be offset, the position of the offset is determined as follows:

- By indicating, or picking, a point;
- By input of coordinates of a point on the circle or arc; or
- By using the option **Distance** from the Option Bar (see below).

### Distance (Command DOFFSET)

#### Located:

1. **Pull-down Menu: Modify – Offset Distance**
2. **Or Enter Command or Alias Directly at the Command Line**

The option **Distance**, selected from the Option Bar, allows the offset to be created a specific distance from the original entity.

Next, determine the distance between the original entity and the offset. Do by either a numeric value input (in drawing units) or by cursor selection.

Then, point (and click) with the cursor to the side of the original entity on that the offset is to occur.

---

#### See also:

DOFFSET

## Cut Selection to Clipboard



**Command: CUTSELECT**

**Alias CT**

#### Located:

1. **Pull-down Menu: Edit – Cut Selection to Clipboard**
2. **Top Function Bar**
3. **Palette Edit**
4. **Or Enter Command or Alias at the Command Line**

With the command CUTSELECT selected objects are copied to a **AllenCad-specific** clipboard (as with the command COPYSELECT)

However, unlike COPYSELECT, with CUTSELECT, the originally selected objects are deleted from the

drawing.

The command PASTE will insert the objects from the PCAD clipboard into any open drawing. PASTE will request the insertion point.

---

**See also:**

Command COPYSELECT  
Command PASTE  
Command DELETE

Delete



**Command: DELETE**

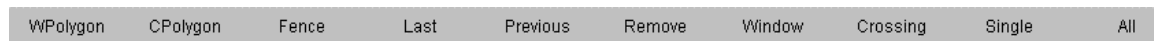
**Alias DEL**

**Located:**

1. **Pull-down Menu: Edit – Delete**
2. **Top Function Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

During the creation of drawings, or, later during the editing, changing or detailing of drawings, it may be necessary to delete certain parts of the drawing.

After calling the command, the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for object selection. These same options will alternately appear in a dialog menu, depending upon your preferences specified in Desktop Configuration.



When deleting, as many objects can be selected as the user desires. When the object selection is complete, the selected entities will be deleted without another confirmation, by simply pressing ENTER.

---

**See also:**

Command UNDELETE

## Delete Partial



**Command: DELPARTIAL**

**Alias DPL**

**Located:**

1. **Pull-down Menu: Modify – Delete Partial**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Delete Partial** will cause a part of a line, circle, or arc to be erased.

These entities can also be **divided** by this command. The result of such a division will be two independent entities that can be edited, or have another operation performed on it, separately.

### Deleting Part of a Line

Deleting part of a line is the standard option for this command. to perform this deletion, determine the end points of the part of the line to be deleted. The line will be divided at this point, and the portion of the line between the two points will be deleted.

### Deleting Part of a Circle or Arc

If the entity is a **circle**, you have only the two-point option available that removes a section of the circle circumference, as a 360o arc is not supported. It must therefore be some angular dimension less than 360o. Arcs, on the other hand, can be either broken into two at a specified point (the one-point option), or else a section can be removed, also leaving two arc objects (the two-point option). With arcs or circles, removal of a section is measured counter anti-clockwise from picked points 1 and 2.

---

#### See also:

BREAK

Break

#### Command: BREAK

Alias B

#### Located:

1. Pull-down Menu: Modify - Break
2. Or Enter Command or Alias Directly at the Command Line

The BREAK command operates identically to the command DELPARTIAL – One Point option. Like this command, the command BREAK breaks objects at a point or points you select. BREAK has been added for convenience of those users who are familiar with other CAD programs.

The main difference is that the command BREAK assumes that you might want to use the pick point when selecting the entity as the first point of the part to delete:

> **BREAK**

Select entity (line, circle, arc, 2D-polyline):

Second point (or F for first point):

However, you can respond to the prompt for the second point the Option *First* (short *F*) to explicitly specify both break points independent from the entity selection:

> **BREAK**

Select entity (line, circle, arc, 2D-polyline):

Second point (or F for first point): **F**

First point:

Second point:

Also, you have the choice to break an object into two pieces without erasing any part of it. To do this, use the command and specify the same point for both the first and the second points; type @ at the request for the second point that signifies the previous point entered. This is useful especially if you want to break an entity at an intersection point with another entity:

> **BREAK**

Select entity (line, circle, arc, 2D-polyline):

Second point (or F for first point): **F**

First point: **int**

of object:

Second point: **@**

---

See also:

DELPARTIAL

## Trim



**Command: TRIM**

**Alias T**

### **Located:**

1. **Pull-down Menu: Modify - Trim**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The TRIM command cuts off sections of linear objects, using another linear object as the "cutter".

The first prompt for this command will be a request to identify the entities to make up the "cutting edge". These entities will be the ones that will result in the "trimmed" intersection of lines. Pick these entities with the cursor.

The next prompt will ask you to select the entity to trim. When you pick an object, that portion of the object on the "selected" side of the cutting object, is trimmed off and deleted.

Note: when trimming multiple objects at once, using the same "cutter", consider using the object selection tool "fence" to trim all linear objects at once that cross the "cutter" along its selected length.

## Fillet



**Command: FILLET**

**Alias F**

### **Located:**

1. **Pull-down Menu: Modify – Fillet or  
Modify – Edit Polyline – Fillet Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Often two solid edges do not intersect exactly, in acute or right angles. With the Fillet command solid edges can be filleted quickly and precisely.

The first prompt will request the selection of the two lines whose intersection is to be filleted. These can also be lines of an object such as a rectangle.

### **Options:**

The following options will be shown in the Options Bar.

#### **Polyline**

Instead of placing a radius between two separate linear entities as with the standard FILLET command, FILLET-POLYLINE will apply the radius at all vertices in the selected polyline. Note: this command acts upon a polyline only once. To change the radius you must UNDO the first fillet command or else reconstruct the polyline by exploding it, deleting the arcs and INTERSECTing the segments and then using POLYEDIT-JOIN to reconnect the segments to form the original polyline.

## Radius

The option **Radius** will define the radius value for the fillet. The program will always offer the value of the last radius used in the current command operation. A radius value of zero means no radius will be applied to the fillet and that entities will simply be trimmed off where they cross each other.

## Trim (Off)

**Trim** functions as a switch. It decides whether the parts of lines that lie beyond the fillet are erased or whether they will be left as separate entities.

If the switch in the Options Bar displays the term **Trim**, you can **switch off** the function **Trim** by clicking the switch. It will then change to **Trim (Off)**.

If the switch in the Options Bar displays **Trim (off)**, by clicking the switch the function will change to **Trim**.

## Chamfer



Command: CHAMFER

Alias CH

### Located:

1. **Pull-down Menu: Modify – Chamfer or  
Modify – Edit Polyline – Chamfer Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

When executing the command, the first prompt will ask for the entities to be chamfered. Select those entities by picking them with the cursor.

In the Options Bar, the option **Distance** will appear.

There are two choices: either select, and enter, a distance for the chamfer, or choose to "pick" the distance for the chamfer. The distance back from the point of intersection for the chamfer must be specified for both of the intersecting entities, since there are often non-symmetrical chamfers.

The option **Trim (Off)**, or **Trim**, functions in exactly the same manner as described with the command **Fillet**, as does the option **Polyline**.

The option **Angle** specifies the relative angle that will be applied to the chamfer, a given distance back from the intersection, on object 1.

---

### See also:

FILLET

Intersect



Command: INTERSECT

Alias IN

### Located:

1. **Pull-down Menu: Modify - Intersect**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Intersect** is a special case of trimming that causes intersecting lines to be trimmed following the intersection. Essentially this command is the same as issuing the FILLET command with a specified radius of zero.

Enter the command by selecting the option **Intersect** from the menu *Modify* or by typing the command INTERSECT.

Select the two lines to be trimmed. Make the selection by picking, and be sure to make the pick on the segment of each line that is to remain.

---

**See also:**

TRIM

FILLETCommand FILLET

## **Flip**



**Command: FLIP**

**Alias FL**

### **Located:**

1. **Pull-down Menu: Modify - Flip**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Flip** contrasts to the **Mirror** command, in that the original object does not remain in its original position; it is simply "flipped" about an axis specified by the user.

The selected objects have, after "flipping", the same relative position to the symmetry axis as they had prior to "flipping", but are now "flipped" in a new direction.

The reflection axis can be defined freely, or parallel to the X-axis or the Y-axis

The first prompt will be for object selection.

Next, select one of the options for choosing the axis from the option bar:

### **Axis**

The option **Axis** allows the user to freely define the flip axis. It is the standard option for axis definition, and is used when no other option has been selected.

Define the position of the axis by the indicating two points or by the input of their coordinates. The axis can have any alignment toward either the objects or the axes of the coordinate system.

If the mirror axis should run through any of the points of existing geometry of the object(s), you should specify object snap functions to define the points precisely.

### **Horizontal Flip**

To flip the marked object about a horizontal axis choose the option **Horizontal**. This option creates a flip axis that runs parallel to the Y-axis of the coordinate system. Only a single point is required to define this axis.

### **Vertical Flip**

To flip the marked object about a vertical axis choose the option **Vertical**. This option creates a flip axis that runs parallel to the X-axis of the coordinate system. Only a single point is required to define this axis.



## Move



**Command: MOVE**

**Alias M**

### Located:

1. **Pull-down Menu: Modify - Move**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command is used to move objects within the coordinate system. An object can be moved in any combination then, of the x, y, or z directions.

The first prompt will be to select the object to be moved. Multiple objects may be selected, and they will be moved together in exactly the same relationship as they are currently placed. Select the object or objects, and confirm the selection by pressing RETURN.

### Options

#### Basepoint

It is the standard reference point for moving objects. It denotes the point that the user is choosing as the base from that to begin the command.

#### Rel. Point

The prompt will then ask for distances in the X, Y, and Z direction. Enter these values (if there is no Z value, enter only the X and Y values). The move will occur when the values are verified.

#### Quick Move

This method of accomplishing the move is to simply pick up the object(s) with the cursor, and "rubberband" them to the new desired location. This works well if no particular precision is required.

## Modify Polylines



**Command: POLYEDIT**

**Alias PE**

### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Edit Entire Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Polylines may be modified in the following fashion:

- Editing of a Polyline as a whole;
- Filletting of a polyline;
- Chamfering of a polyline; and
- Editing individual polyline segments by editing their control points.

### Polyline Editing

The command allows the user to:

- Open or close the Polyline;
- Join the polyline with other drawing entities such as line, arc or polyline;
- Change the width of the whole polyline; or
- Undo the last step.

The first option panel will display the Option Open or Close, depending on whether the selected polyline is currently opened or closed. Note that the option "Back" is only available after the execution of the first operation under the command.

### Close / Open

The option **Close** joins the start and end points of the polyline with a straight polyline segment. The width of this segment equals the ending width of the end point of the last drawn polyline.

The option **Open** is a possible option if the polyline to be edited has already been closed during drawing, or in a successive working step. The polyline segment between the start and end points of the polyline will in this case, be deleted.

### Join

**Join** allows lines, arcs and other polylines to be joined with the selected polyline. The prerequisite to this is that the polyline to be edited and the object to be joined meet in one ending point.

After the calling of the option *Join*, indicate the drawing entity to be joined to the previously selected polyline. The polylines chosen are now joined as one entity.

### Width

**Width** offers the opportunity to change the width of the polyline. Polygons that have different start and end widths as single segments, can be made a uniform width for all segments.

Define the desired width of the polylines by numeric input or indicating a width with the cursor. As a "help" parameter for this input, the value used last for the width of the polyline segment to be joined will be displayed. This value may be used by pressing RETURN.

### Undo

**Undo** will remove the last operation. The POLYEDIT command is continued, and the user will see the request "Input:".

---

### See also:

VTXEDIT

If the vertices only are to be edited that option may be picked from the sub-menu, or keyboarded by typing VTXEDIT.

Vertex Editing



**Command: VTXEDIT**

**Alias VX**

### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Vertex Editing**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Editing of single segments of a polyline is done through the command **Vertex Editing**. This command allows the user to change the vertices (the start and end points) of the polyline. Vertices of a polyline

segment can be inserted, deleted, and moved. Also, the width of the polyline segments can be changed. Select the function by selecting the option **Vertex editing** in the submenu *Modify Polyline* of the menu *Modify*, or by typing the command VTXEDIT.

### **Vertex Selection**

Select the polyline containing the vertex to be edited. The selection of the polyline will also cause the selection of the vertex containing the start point as the first vertex to be edited.

The vertex selection may be changed by "stepping" through the vertices using the **Next** and **Previous**. The user should experiment with these selections until he or she understands how they work.

The selected control point will always be marked by a "rubber band" line joining the selected point with the current cursor position.

After selecting the control point to be edited, select one of the editing options from the Options Bar:

### **OPTIONS:**

#### **Insert**

The option **Insert** allows the addition of vertices, and thereby of further segments, to an existing polyline.

Define (either by picking with the cursor or by input of coordinates) the position of the additional vertex.

The width of this new segment will correspond to the width values of the polyline, at the bordering control points, before the addition of the new segment.

After inserting a segment the command VTXEDIT stays active. The user may select another operation or complete the use of the command by pressing ESC.

#### **Move**

The option **Move** enables the user to relocate a vertex of the polyline to a new position.

First determine the vertex to be relocated. Then define (by "picking" with the cursor or by input of coordinates) the new position of the chosen vertex. The vertex will then be moved to this position. Other segments of the polyline will remain unchanged.

The command will remain active. Finish editing the polyline segments by pressing ESC.

#### **Delete**

Define the vertex to be deleted in the same fashion as was used with the previous options. Then select the option **Delete** from the Options Bar.

The segment having its start point in the selected vertex will be deleted. The previous (in order of drawing) segment will be lengthened to the next control point.

The command VTXEDIT remains active. Select another option or finish editing the polyline segments by pressing ESC.

#### **Width**

Width allows the width of single segments of a polyline group to be changed.

Define the vertex to be deleted in the same fashion as was used with the previous options. Then select the option **Width** from the Options Bar.

Input, or select with the cursor, the desired width of the polyline segment. The width of the selected segment will be changed according to the inputs.

The command VTXEDIT remains active. Select a another operation or finish editing the polyline segments by pressing ESC.

---

**See also:**

POLYEDIT

## Commands Measure and Divide

**Command: MEASURE, DIVIDE**

**Alias MR, DI**

**Located:**

1. **Pull-down Menu: Draw – Measure / Divide**
2. **Or Enter Command or Alias Directly at the Command Line**

The commands MEASURE and DIVIDE order points or multiple blocks (parts) along a 2D-Object. This object can be an arc, a line, a circle or a 2D-Polyline. Before a block may be used with these commands, **it must exist in the drawing database before the command is given**. This may be accomplished, either through insertion or part definition. When the command is given, the block can be inserted aligned, or not aligned to the object. Both commands work in a similar fashion. While the command MEASURE asks for a distance, DIVIDE requests for the number of points with the distance between points or parts dependent upon the length of the linear entity selected.

### Command sequence for MEASURE and DIVIDE

After typing in "measure" or "divide" (or selecting it from the menu DRAW), the following request appears:

/↻> measure

Select a line/circle/2d-polyline or an arc to measure:

After selecting the entity it is highlighted and the following question appears:

Block or distance:

Here you can input either a distance directly, (by putting in a real value or selecting two points), or you can input "Block" (or "B". "BI" etc), to use a block. In the first case the points at each distance from the start to the endpoint of the object are inserted and so the command is completed. If you select the option "Block" you will be asked for the block name:

Name of block to insert:

Now type in the name of an **existing** block. You will be asked for the offset angle for the alignment of the block along the object.

Offset angle for alignment the block along the object (or ENTER for no alignment):

Here you can press return to avoid the alignment, or to put in any offset angle that is added to the tangential angle of the object at the point of insertion. Lastly, you will be asked for the distance again.

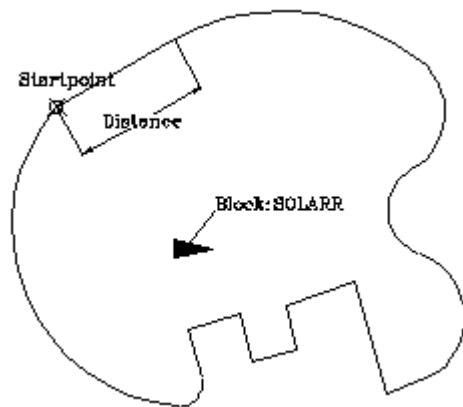
Distance for dividing the object:

You have to put in the distance here, either by giving a real value or by selecting two points. Now the Block at each position is inserted and the command will terminate.

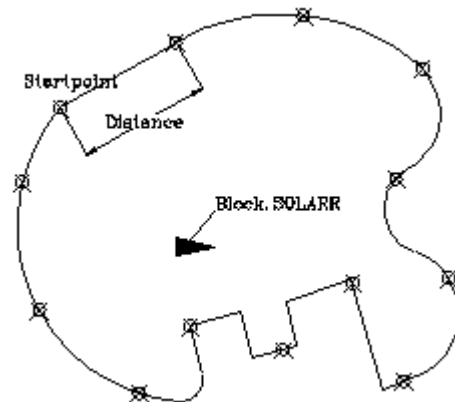
The sequence of the command DIVIDE is the same as above, except that you are asked for the number of points or blocks to insert, instead of a distance. In this case the command will compute the length between the insertion points itself.

### Example:

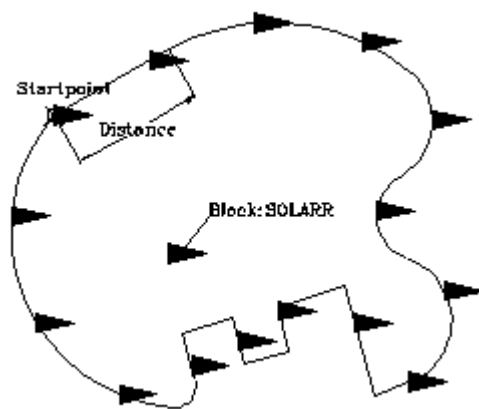
The following pictures show at the sample of a polyline, how the commands MEASURE and DIVIDE are working:



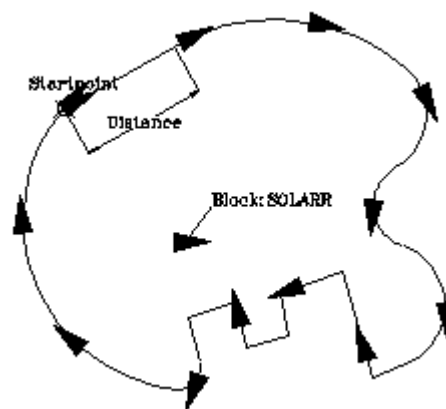
The initial state: The entity to measure is the polyline. Moreover the block SOLARR is already defined in the drawing.



After measuring the polyline using the distance, we receive points at each distance along the polyline, starting at the polyline start point



After dividing the polyline in ten pieces the block SOLARR is inserted ten times along the polyline, but it is not aligned.



In this case the polyline is divided into 12 pieces with the option block, and aligned with an angle of 0 degrees, this means the block is inserted locally tangential to the polyline.

### Technical notes

The command Divide inserts number+1 entities along the object unless a circle is selected. So it may be necessary to delete the first or last inserted entity, if a closed polyline is selected. This is much easier, however, then creating an additional entity manually.

The command is not an internal command and so it available for the usage inside the (command...) function. To use it inside of a LISP-application use the related LISP-Function (MesDiv EntName Dist|Nr BName OffsAngle) instead that is defined in MesDiv.LSP in the PCAD-Applic directory.

---

**See also:**

POINT

## Rejoin broken Lines or Arcs



**Command: REJOIN**

**Alias RJ**

**Located:**

1. **Pull-down Menu: Modify - Rejoin**
2. **Palette: Modify**
3. **Or Enter Command or Alias at the Command Line**

Rejoin will unite separated co-linear lines segments or arc segments that share the same center and the same radius. The resulting joined line or arc will be a single entity.

A common function for this command is to reunite separated line or arc segments that have been divided by the command DELPARTIAL (Delete partial).

Select the command, then pick the entities to be rejoined.

> REJOIN Select first entity (line or arc): *P1*

Select second entity (line or arc): *P2*

---

**See also:**

**Delete partial**

**Rotate**



**Command: ROTATE**

**Alias RO**

**Located:**

1. **Pull-down Menu: Modify - Rotate**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Rotate will serve to rotate an entity or entities through an angle, about an axis.

As with the other modification commands, Rotate requires the selection of the object/ objects to be rotated. There are several methods suggested by the Option Bar of selecting the object(s). Choose a method, and make the selection.

## Options

### Basepoint

The command will prompt for the Basepoint. The Basepoint is the point about that the rotation will occur. It is handy to pick a corner, or a center, but any point may be picked that will give the desired result. Enter, or pick, the Basepoint.

Note that the Basepoint may not be a point on the described geometry of the object or objects, but may fall off of their geometry.

The next prompt will ask for the angle of rotation. The user may elect, through the Option Bar, either the **Angle of Rotation**, or a **Reference Angle**.

### Angle of Rotation

The option Angle of Rotation is the standard option. It is used if no other option has been selected.

Angle will prompt the user for a rotation angle. Enter that angle, and rotation will take place to the specified angle.

### Reference Angle

The Reference Angle will ask the user to specify the angle at that the object is now placed, and then prompt for a New Angle, to which the object will be rotated. Rotation will always be counter-clockwise. As before, rotation will occur about the base point.

## Scale



**Command: SCALE**

**Alias SC**

### Located:

1. **Pull-down Menu: Modify - Scale**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The **Scale** command allows the user to change the "scale" of the objects that is, to enlarge or reduce them while keeping all proportions the same.

The first prompt will be to select the object(s) to be scaled. Use one of the options presented in the Options Bar for this. Remember that if more than one object is selected, the objects will be treated as one by the command.

The next prompt will be to identify the **base point**, and that should be done as it was in the two previous commands. This will bring up the Options Bar with the following options:

Sc.Factor | Reference

## Options

### Scaling Factor

The option **Scaling Factor** is the standard option for scaling objects; and is used if no other option has been selected.

Note that it is important to realize that the **base point**, if is a part of the selected object, will hold its original position during the scaling, while all the other points will be moved by the same relative values in direction of the X- and Y-axis. As in other cases, however, the user may select any other point, whether on or off the geometry of the object(s). In this case, the distance from the base point to all of the other points that comprise the object will be changed by the same scale factor.

To continue, enter a scaling factor. The use of a scaling factor bigger than 1 results in enlarging the

selected objects by this factor. A scaling factor between 0 and 1 results in reducing the objects by that factor.

For instance, the factor <3> enlarges the selected objects three times, while the factor <0.25> results in reducing the objects to one-quarter of their original size.

### Reference

This option allows an exact change of the size of an object to a particular scale.

For instance, if the user were to enlarge an object with a side length of 3.45 drawing units up to 6.90 drawing units, he or she would first have to calculate the scaling factor. **Reference**, however, facilitates this calculation by the input of a reference length (here 3.45) a precise changing of the size up to the desired scale (here 6.90), without the user having to calculate a scaling factor.

To use this variation of scaling, select the option **Reference** from the Option Bar. Enter the **Reference length**, and then enter the **New length** that is desired.

## Lengthen



**Command: LENGTHEN**

**Alias LEN**

### Located:

1. **Pull-down Menu: Modify - Lengthen**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Lengthen** provides the ability to extend the length of existing lines or arcs.

Unlike the command **EXPAND**, **Lengthen** extends an entity to a specific value or in a particular ratio (percentage value) of the original size.

Entities will be only lengthened in their existing direction. All entities will always be lengthened on the end that lies closest to the point where the cursor was positioned during the selection.

### Options:

#### Dynamic

Lengthen normally works in the **Dynamic** mode. This mode is active after the selection of a drawing entity and needn't be selected from the Option Bar. After any lengthening operation the mode will return to Dynamic, no matter which of the options has been used.

In the Dynamic mode the line or the arc will be lengthened to a point or to an angle indicated the cursor. A preview will show, during this operation, the result of the command.

In addition to the standard mode Dynamic, the following modes are available for lengthening lines and arcs from the Option Bar:

1. **Total** allows the entity to be lengthened to a dimension that is defined. The dimension will apply to either the length of a line or to the circumference of an arc. To assist the user, the current length of the selected entity is displayed in parentheses. If the option **Angle** been selected the included angle of the arc is displayed instead of the length, and the input should also be the additional angle required.
2. **Percent** will permit the entity to be lengthened by a defined ratio of the original size. This percentage value relates to either line length or the circumference of an arc. For instance, the input of the percent value 200 doubles the length of the line or the arc, the value 50 diminishes it to the half of the original value. If the option **Angle** has been selected, the input of the percentage value will apply to the arcs original angle.



3. **Relative** lengthens a line or an arc by a length that is defined by indicating as distance between two points. This is an advantage particularly if the entity is to be lengthened by a straight line.

Expand



**Command: EXPAND**

**Alias EXP**

**Located:**

1. **Pull-down Menu: Modify - Expand**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Expand** allows the user to extend objects to a selected common boundary edge. In this way, it is almost identical to the EXTEND command.

During the execution of the command the drawing entities to be expanded are extended. They do not change direction. A line will remain a line, an arc will remain an arc. For chains and polylines, only the last segment will be extended to the selected boundary.

Select the entities that are to serve as boundary edges for the entities to be expanded. These boundary entities will remain unchanged. Object selection functions are presented in the Option Bar.

Next, select the entities to be expanded. In doing this, make sure that the selecting window of the cursor is on the part to be trimmed.

The expansion of selected entities is always executed from the endpoint that lies nearest to the cursor when it selected the entity.

If more than one boundary edge has been selected, the entity is expanded to the first boundary edge first. "Picking" a second time will result in the expansion being continued to the second boundary edge.

Stretch



**Command: STRETCH**

**Alias SR**

**Located:**

1. **Pull-down Menu: Modify - Stretch**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The **Stretch** command is useful for the enlargement or reduction of objects in only one direction (in contrast to scaling). Subsequent stretching of the same object in a second direction is not supported. Standard geometric entities (circles, rings, etc.) are not distorted, only shifted in position relative to the other objects being stretched.

Selecting entities for stretching must be done very carefully to ensure the desired effects, as **selection is by crossing and crossing polygon only**. During stretching, the method of selection has an influence on the question of which entities are moved and which entities are stretched.

- All entities included totally in the selecting windows will be moved;
- All entities crossing the selecting window will be stretched (lengthened); and
- All other drawing entities will retain their position and shape.

After having completed object selection, choose an option for stretching from the Option Bar..

### Options:

#### Basepoint

The option **Basepoint** is the standard option for moving entities. It is used if no other option has been selected.

The option is useful for moving selected drawing entities to a particular position and for "stretching", or moving, entities on a border, without changing the relationship to the border. The target position may be a point defined by coordinates of a position determined by the geometry of the object.

Next, determine the end point of the stretch. The base point for the stretch will be moved to the end point.

#### Displacement

The option **Displacement** allows stretching selected entities by a straight line (a vector). After selecting the objects, select the option from the Option Bar. Enter the value for the move, a straight line, in the form X,Y, (Z).

#### Thickness

#### Command: THICKNESS

##### Located:

1. **Pull-down Menu: Tools – Entity Extrusion – No Thickness  
And Set Thickness**
2. **Or Enter Command or Alias Directly at the Command Line**

By applying the Thickness property to drawing entities, you can create a three-dimensional effect of certain entities. Such extrusion of entities creates the illusion of depth.

Thickness is displayed with the following objects:

1. Line
2. Arc
3. Circle
4. 2D-Polylines
5. 2DFace

The Thickness property is also displayed, if the entities listed above are drawn with a dash-dot pattern or if these are wide polylines.

As commands like NGON, RECTANGLE, ELLIPSE, RING, or CHAIN create 2D-polylines, Thickness setting is applied to them.

Thickness can also be assigned to other drawing objects (3D-polylines, 3D-faces, text, attributes, attribute-definitions, block insertions, dimensions, hatchings, ...), but the program will not display the extrusion of those objects.

To set a current preference for subsequent drawing, set the THICKNESS system variable. Type in the system variable name and enter a numeric value. To turn off applying Thickness for further drawing, type in THICKNESS and set the value to zero.

To alter the Thickness of drawing object, use the PROPEdit command.

#### Applying Object Selection and Object Snap on extruded entities

When selecting entities that are displayed with extrusion thickness or when snapping to them, keep these two rules in mind:

1. *Object Selection:* When you are requested to select objects, the program will recognize picking extrusion lines as if the base entity has been selected.
2. *Object Snap:* When you snap to extrusion line of entities that have a thickness, the program will find the appropriate point of the base entity.

### **Treatment of extruded objects in Rendering and Hidden Line Removal**

Objects that have a thickness are recognized in the Render and Hidden Line Removal commands (ARENDER, RENDER, HIDE, FHIDE).

## **Explode Complex Objects...**



**Commands: EXPLODE, XPLODE**

**Alias EX. XPL**

### **Located:**

1. **Pull-down Menu: Edit – Explode Complex Objects**
2. **Palettes: Edit and Parts**
3. **Or Enter Command or Alias at the Command Line**

Parts and other complex objects, like polylines, crosshatches, and dimensioning, inserted in the drawing constitute one single drawing object.

In this mode, it is not possible to edit any of the individual entities that make up the part. If, for any reason, there is a requirement to edit one of the entities comprising a part, then the part must be "exploded" (broken into its individual entities), the element(s) edited, and then the component parts reassembled again.

The explosion of complex objects is executed by the command EXPLODE, or by selecting and activating the option **Explode Complex Objects...** in the *Edit* menu.

When the object selection is complete, the objects (parts, polylines, hatches, or dimensions) will be exploded.

Now you may choose any of the entities for individual editing.

### **Notes:**

1. It is important to realize that, if there are attributes attached a part to be exploded, these attributes will be lost. Instead, the attributes defining names will appear.
2. Exploding a part does not change the part definition in the drawing. Another part or parts with the same name and original definition can be re-inserted into the drawing at any time.
3. To change the part definition in the drawing, you must modify the part components after exploding, then create a new internal or external part with the PARTEXP or PARTDEF commands, using the old part name. The program will ask you to confirm the re-definition when you do so.

Rename

**Command: RENAME**

**Alias RNM**

### **Located:**

1. **Pull-down Menu: Edit – Drawing Database Tables – Rename Table Items**

## 2. Or Enter Command or Alias at the Command Line

The RENAME command allows you to alter the names associated with blocks, layers, linetypes, text styles, dimension styles, user coordinate systems, and views.

### To rename a drawing database table entry

- First, specify the drawing database table that contains the item to be renamed.
- Next, enter the name of the named object you want to rename.
- Finally, type in the new name of the item.

### Example

```
> RENAME
Rename symbol (BBlock, Dimstyle, LAyer, LType, Style, Ucs, or View): LA
Current name: dimensions
New name: dim
```

### Tip

You can rename these named symbols used within the current drawing from within the *Drawing Xplorer*.

### Command DRAWORDER

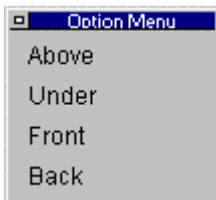
#### Command: DRAWORDER

#### Alias DO

#### Located:

1. Pull-down Menu: Edit – Change Object Display Order
2. Or Enter Command or Alias at the Command Line

The **Command\_DRAWORDER** command changes the drawing and plotting order of any object in the AllenCad drawing database. In addition to moving objects to the "front" or "back" of the sort order, you can order objects relative to another object (that is, above or below a selected object). After issuing the command, you are asked to pick entities to re-order. You are then presented with the following dialog, once this selection is complete:



You simply select the option that you want and then select the entities that you want the original entities placed in relation to.

AllenCad differs from AutoCAD, in that object database order is changed directly, using the Command\_DRAWORDER command. Different object sort methods are therefore not supported, and a SORTENTS system variable is not required in AllenCad.

See Also HIDE, FHIDE

## Obtaining Drawing Information

Entity Info...



**Command: EINFO**

**Alias LI**

**Located:**

1. **Pull-down Menu: Edit – Entity Info**
2. **Left SideBar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

The EINFO command prompts you to select objects for identification. It then opens the text window and displays object information as in the following EINFO report on a selected line entity:

```
> EINFO
Select objects:
  1 selected.
Select objects:
*** 1 selected. ***
Type:      LINE
Layer:     0
Line type: Layer's line type
Color:     Layer's color
Handle ID: "A"
Length:    0.44
Thickness: 0.00
Start Point (UCS):  5.09, 5.31, 0.00
End Point (UCS):    5.53, 5.35, 0.00
Delta (UCS):0.44, 0.04, 0.00
Start Point (WCS):  5.09, 5.31, 0.00
End Point (WCS):    5.53, 5.35, 0.00
Delta (WCS):        0.44, 0.04, 0.00
Projected length (in xy-plane): UCS: 0.44      WCS: 0.44
Angle in xy-plane:   UCS: 6  WCS: 6
Angle to xy-plane:   UCS: 0  WCS: 0
```

**Tip:** You may use this information in external reports, etc, by highlighting the text and using Windows' Copy (Ctrl C) and Paste (Ctrl V) key definitions to copy the text from the text window first to the Windows Clipboard (Ctrl C) and then to the external application (Ctrl V).

## Identify Coordinate



**Command: 'ID**

**Alias ID**

### Located:

1. **Pull-down Menu: Edit – Identify Coordinate**
2. **Or Enter Command or Alias at the Command Line**

The transparent command **ID** (identify coordinate) determines the coordinates of a selected point and then displays that information.

The function **Identify Coordinate** may also be selected from the menu *Edit*.

### Using the command

Indicate the point to be determined with the Cursor. If it is a geometrically significant point of a drawing object (for instance, corner point, center of a circle, intersecting point of two lines etc.) use the Object Snap functions.

The coordinates will appear in the display.

```
> ID
Point: P1
X: 1.27 Y: 4.45 Z: 0.0
```

Distance / Angle



**Command: 'DIST**

**Alias D**

### Located:

1. **Palette: Edit**
2. **Left Sidebar**
3. **Pull-down Menu: Edit – Distance / Angle**
4. **Or Enter Command or Alias at the Command Line**

The transparent command **Distance / Angle** will determine the distance or angle between two points and displays them in the text window:

1. The absolute distance (range);
2. The Distance on the X-, Y- and Z-axis;
3. The Angle on the XY-plane; or
4. The Angle on the XY-plane.

The points may be arbitrary points that do not have to be part of a single drawing object.

The function **Distance / Angle** operates as follows:

1. Determines the two points required by selecting them with the Cursor, using the Object Snap functions or by input of the coordinates; then
2. The distance and angle between the two points will be displayed.

The term *Distance* is understood to mean the absolute distance between points.

Example:

```
> DIST
First Point:
Second Point:
Distance: 0.37 Angle in XY plane: 41 Angle from XY plane: 0
```

Delta X: 0.28 Delta Y: 0.24 Delta Z: 0.00

The terms *Delta X*, *Delta Y* and *Delta Z* designate the distance between the two points in the direction of the respective axis of the coordinate system. All length values are given in drawing units.

*Angle in the XY-plane* measures the angle to the second point starting from the X-axis. It always displays the lesser-valued angle, no matter what the rotation direction.

*Angle from the XY-plane* measures the angle between the first and the second point from the XY-plane to the Z-axis. It is assumed that the first point lies on the XY-plane.

## Area / Circumference



**Command: AREA**

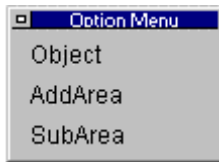
**Alias AR**

### Located:

1. **Pull-down Menu: Edit – Area / Circumference**
2. **Left SideBar**
3. **Palette: Edit**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

This function defines the circumference and the area of one or more drawing objects and displays the values. For objects that do not have width, the length instead of the circumference will be displayed. The areas of single objects can be also added or subtracted.

As soon as the command is issued, you are presented with the following option dialog:



These options are described below:

### Using Area / Circumference

To determine the circumference and the area of one or more objects proceed as follows:

Define the drawing object to be examined by tracing its outline or using the Object option. The preferred method of selection is by using the cursor and Object Snap functions, but inputting the coordinates of the points is also okay. Pressing RETURN at the request **Next Point:** will finish the selection. It is not necessary to select the first point again to close the boundary, as the program assumes a closure between first and last points. It also does not matter which direction you trace the outline in, so long as you complete the outline definition in either a continuous clockwise or anti-clockwise direction.

### Object: Regular Areas

"Regular" geometric areas (of circles, arcs, closed polylines) can be evaluated by choosing the **Object** option.

### Add or Subtract Areas

Select the first partial area or determine the points for describing the area geometry, as above. The determined values will be displayed in the text window.

Select the option *AddArea* to add a partial area, or *SubArea* to subtract a partial area, from the total area.

The values (circumference /length and area) for the total area will be displayed in the text window, after additions or subtractions are made.

When partial areas are added or subtracted from an area, the total area after the addition or subtraction is regarded as the new (initial) parameter value for subsequent additions or subtractions unless the user explicitly selects a contrary option from the Options Bar.

The Options menu will display only that option that is not active at the moment.

---

**See also:**

System variable AREA

## Command TABLES



### Command TABLES

#### Alias TBL

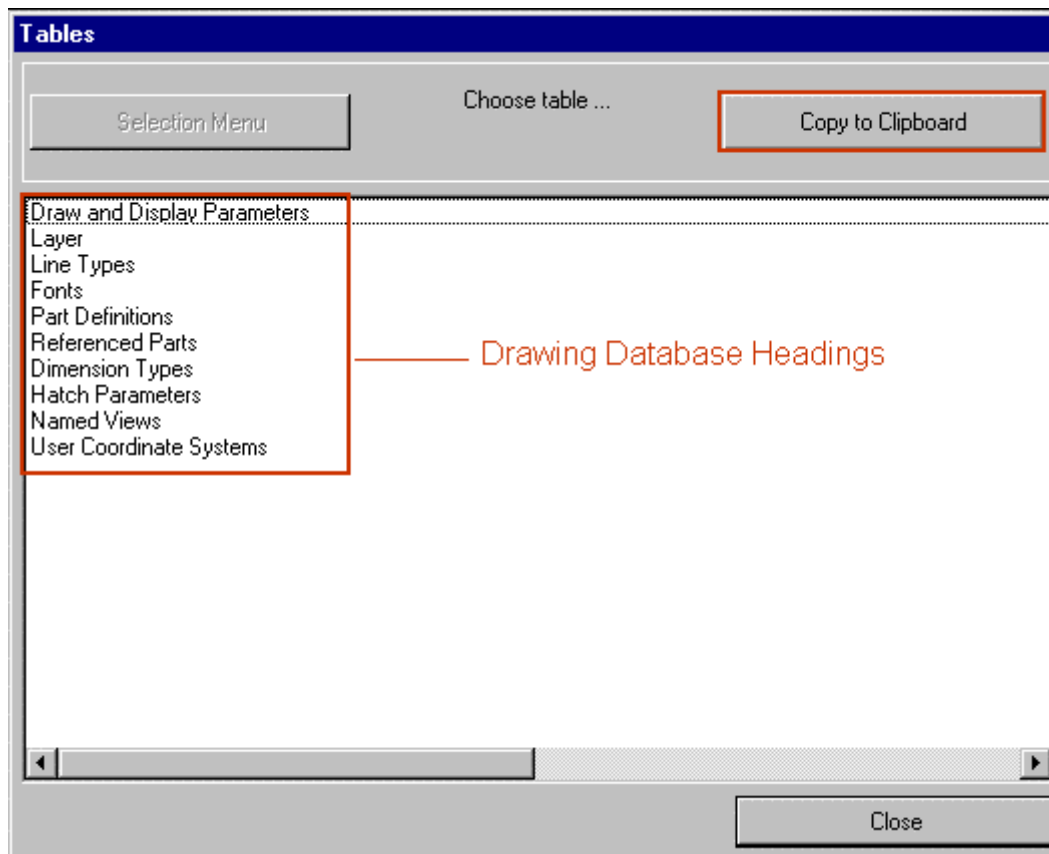
##### Located:

1. **Pull-down Menu: Edit – Drawing Database Tables – Tables Info**
2. **Left SideBar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

Through the use of the TABLES and associated commands, AllenCad offers the user simple control and auditing capabilities over the entire drawing database. The TABLES command is located on the Edit pull-down menu as the Drawing Database Tables – Tables Info selection. It can also be entered at the command line if preferred.

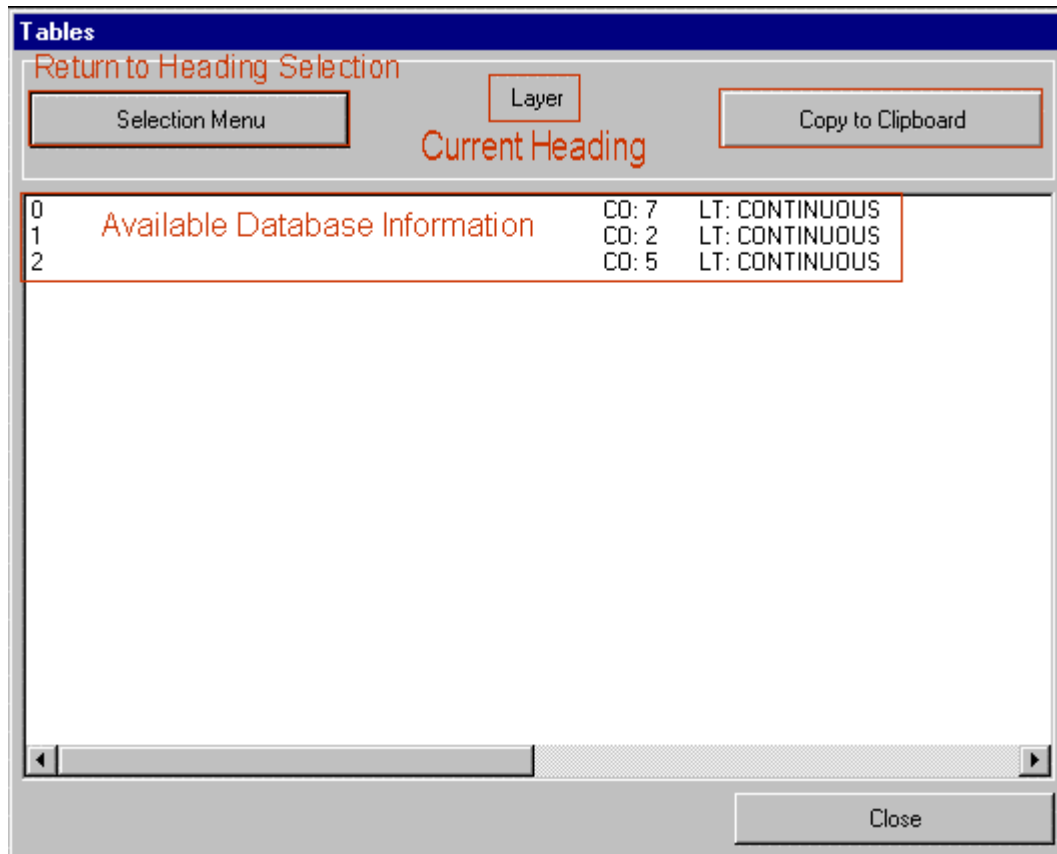
Upon execution, the following dialog is opened:





Here, information on drawing parameters used can be listed out and copied to the clipboard, to be used in reports or even imported into the drawing, itself.

If we select the option LAYER, for example, we see a listing of all layers and layer parameters in the current drawing:



This listing is copied to the clipboard by pressing the Copy To Clipboard button and you can return to the original listing of drawing and display parameters by clicking on the Selection Menu button.

## Detailing the Drawing

### Text

#### Text Objects

A text entity is defined by the following properties. These properties are assigned during the creation of the text entity. Any of them may be changed after the creation of the original entity.

The properties and characteristics of text objects are:

A Character string that represents the content of a text object. All characters or special characters may be used for this string.

The Font (typeface) - the program contains a series of standard fonts that can be used in creating text objects. Font files are marked by the extension .fsh.

The Text height (height of characters) defined in drawing units.

The Alignment of the text objects - determines the horizontal and vertical position of the character string relative to the insertion point of the text object. The character string can run right or left from the insertion point, or the insertion point may be situated centered in the character string. The vertical alignment of the text objects may be above, center, base line, or below. Any text object can be rotated during insertion as well.

The Insertion point of the object. The insertion point (and the alignment) of a text object determines its position in the drawing. The insertion point can be defined by coordinate input or by selecting the point with the cursor.

Related Commands:

TEXT	Drawing text characters
DTEXT	Direct Text Entry
TEXTEDIT	Modifying text
FONT	Creating and modifying text style definitions
FCOMPIL	Compiling source shape files to binary font files
MTEXT	Creating multi-line text
MTEXTEDIT	Editing multi-line text
RTEXT	Reference Text

Note: For control of onscreen text display see TEXTMINDISP.

Text...



**Command: TEXT**

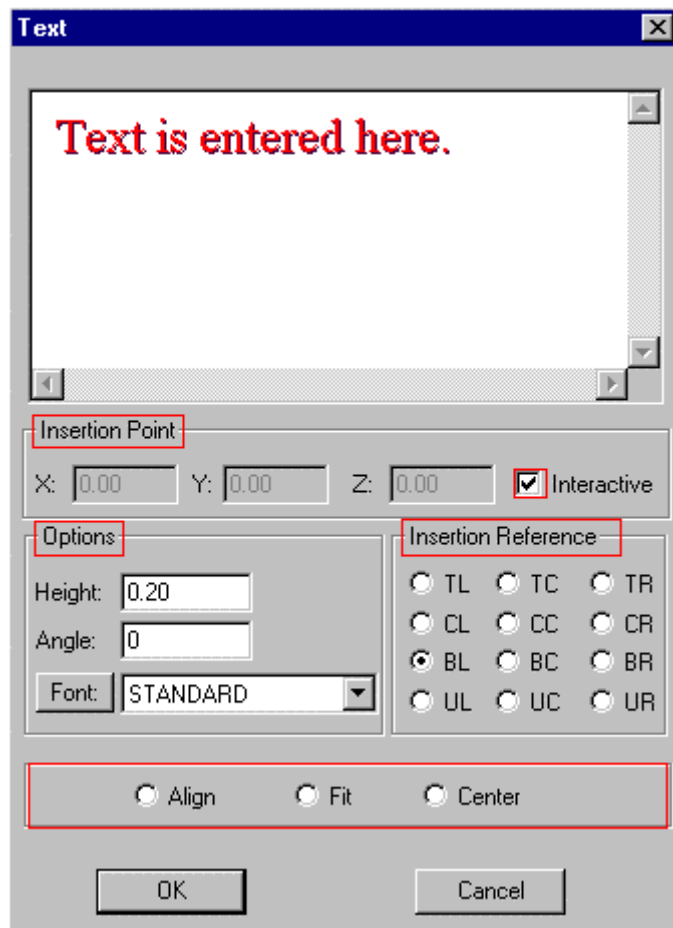
**Alias TX**

**Located:**

- 1. Pull-down Menu: Detail – Text**
- 2. Left SideBar**
- 3. Palettes: Draw and Detail**
- 4. Or Enter Command or Alias Directly at the Command Line**

The TEXT functions are used for inserting text into drawings. They provide labeling, create legends, descriptive notes, and other text elements in a drawing. Most of these commands and options are found under the menu *Detail*.

The command TEXT comes up with a dialog box that allows you to input text and to set several parameters for the text to be added to the drawing:



### Text input

Text is entered into the input field in the upper area of the dialog box, as indicated. The cursor will be positioned in this field upon the opening of the dialog box, so the user may start the input of the text immediately.

The text editor in this dialog box offers the basic functions for editing text input. Incorrectly entered text can be marked and erased just as with a word processor.

When the text entry is completed, the user should make any of the changes that are desirable and within the capabilities of this dialog box. The changes possible are discussed below.

When a line or block of text is complete, press ENTER.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's TEXT command dialog window, either through picking on the window and using Windows' "Ctrl V" function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry

### Insertion point

The Dialog Box offers two methods to define the insertion point. The insertion point may be entered using (absolute) coordinates (boxes are available for input of the X-, Y- and Z-coordinates). The second option is to define the insertion point interactively. The option **Interactive** (shown highlighted) must be checked to activate this option. Unchecking interactive will enable absolute coordinate entry.

## Options:

The following options aid the user in determining the height, the rotation angle, and the font (typeface) of the text object to be inserted.

### Height

Define the size of the text in the *Height* edit box. Enter the desired text height in drawing units.

### Angle

Define the insertion angle of the text object by this input field. Any rotation is possible, and the text can also be aligned to vertical or sloping objects. Parameter value for this is a rotation angle of 0 degrees (the text object is inserted horizontally). For a counter-clockwise rotation, enter a positive value; and for a clockwise rotation, enter a negative value between 0 and 360.

### Font

A window allows the user to define the font. The list of available fonts is displayed in this window. Select the desired font from this list.

## Alignment:

The alignment of the text object(s) can be specified by the following options:

### Insertion Reference

Defining an **Insertion Reference** point places the text in a relative position to the selected insertion point. For this, a reference point on the text object is defined, as described below.

If the reference point, for instance, is situated at the lower left corner of the text object the text will be placed such that the insertion point is at the lower left corner of the text, with text running to the right of the insertion point. If bottom right is selected, the text will lie to the left of the insertion point.

In total there are twelve reference points available for the user's choice. They are represented by the initials of the locating corner (or center) of the text block. UL, for instance, represents the Upper Left corner of the block. Select the position for the reference point by selecting the appropriate radio button.

### Align

Align allows text objects to be rotated and scaled during insertion. The rotation angle and the scaling factor are defined by indicating two points on the drawing. These are the procedures for using Align:

- Activate the option by clicking the box *Align* at the lower left side of the Dialog Box *Text*. The options insertion point, text height, angle and position will not be available any longer. The parameters required for alignment, result from the definition of two points during the insertion of the text object. The reference point of the text object is positioned left justified, lower corner;
- Confirm the use of the option Align by clicking OK;
- Enter the first point for insertion of the text object. This point is interpreted by the program as both the insertion point and the first point for the rotation angle and scaling factor;
- Enter the second point that completes the definition of the rotation angle and the scaling factor, and sets the text height.

### Fit

Fit enables the simultaneous rotation and expansion of a text object during the insertion. The text object is not scaled that is, the text height remains unchanged, while the string is expanded or compressed to fit. The procedure is:

- Activate the Fit option in the dialog box;
- After inputting the text string(s) confirm the settings by clicking OK;
- The prompt will be for the first point - it may be specified or "picked" - do this; then
- The prompt will be for the second point - specify or "pick" this point.

The text will be fit between the two points, rotated so that it follows the line between the points.

### **Center**

Center enables the user to insert a text object with both vertical and horizontal alignment centered. Text height and rotation angle may be chosen.

For onscreen display of text characters, see TEXTMINDISP.

## **QTEXT command (Command Line Version)**

### **Command: QTEXT**

#### **Alias QT**

#### **Located:**

#### **2. Entered Directly at the Command Line**

The QTEXT function is used to create text entities in the current drawing.

The command QTEXT works like the command TEXT but with one difference: QTEXT is performed at the command line.

>QTEXT

Specify start point or select option (Justify, Style): **Justify**

Specify justification (Align, Fit, Center, Middle, Right, TL, TC, TR, CL, CC, CR, UL, UC, UR): **Middle**

Insertion point: **10,10**

Text height <0.20>: **0.40**

Insertion angle <0>:

Text: **3D-Trafo**

---

#### **See also:**

TEXT

QTEXT is the command line version of the TEXT command.

### **Command DTEXT**

#### **Command: DTEXT**



Alias DT

#### **Located:**

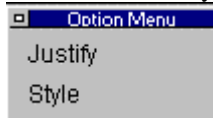
- 1. Pull-down Menu: Detail – Single Line Text**
- 2. Palette: Detail**
- 3. Or Enter Command or Alias at the Command Line**

Unlike the other text input commands, DTEXT places your text on the drawing as you enter it.

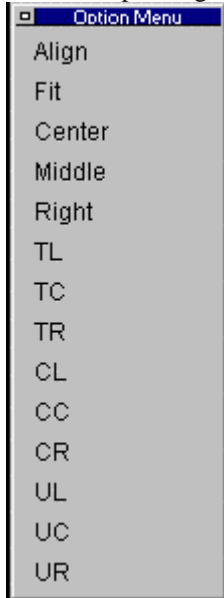
AllenCad's DTEXT command operates essentially the same way that AutoCAD's DTEXT command does. When you issue the command, you are prompted at the command prompt to pick the start point for your text on the drawing.

```
> *** Cancel ***  
> dtext  
Specify start point or select option (Justify, Style):
```

At the same time, you are offered two options:



These will either be offered at the command line as well as in the option menu as shown, or else on the toolbar, depending upon your desktop preferences. Choosing Justify will raise another dialog:



On this dialog you can select the justification you want the text to have.

Choosing Style will raise an input prompt at the command line requesting the required text style name.

Next, you will be prompted to enter the required text height, if other than the current default.

```
> dtext  
Specify start point or select option (Justify, Style):  
Text height <0.20>: |
```

Note that as with most dimension prompts, you can also respond by picking two points on the drawing that will define the height to apply to the text.

Next, you are asked to input the angle (orientation) at which the text is to run. Again, picking two points on the drawing will establish the text orientation.

```
Specify start point or select option (Justify, Style):  
Text height <0.20>:  
Insertion angle <0>: |
```

Lastly, you will type in your text. Note that it appears immediately on your drawing as you type. Pressing the Return key will move you to a position one line below, on which to type a second or subsequent lines of text. Pressing Return key twice in a row will terminate text entry and pressing the Esc key will cancel the current command.

## Modify Text...



Command: TEXTEDIT

Alias ED

### Located:

1. Pull-down Menu: Detail – Modify Text
2. Left SideBar
3. Palette: Detail
4. Or Enter Command or Alias Directly at the Command Line

You can modify an existing text line in the drawing by selecting **Modify Text...** from the menu *Detail*.

Select a text entity in the drawing. The dialog box *Text Edit* is displayed that is similar to the dialog to create text (with exemption that only one line can be edited).

Text Edit

Text:  
This is the text we wish to modify.

Insertion Point  
X: 3.43 Y: 5.07 Z: 0.00 ☐ Interactive

Options  
Height: 0.24  
Angle: 0  
Font: STANDARD

Insertion Reference  
☐ TL ☐ TC ☐ TR  
☐ CL ☐ CC ☐ CR  
☒ BL ☐ BC ☐ BR  
☐ UL ☐ UC ☐ UR

☐ Align ☐ Fit ☐ Center

OK Cancel

Simply edit the existing text in the window meet the requirements. The dialog box also presents the parameters attached to the selected text. These parameters can be modified as required, for the replacement text.

If the check box labeled *Interactive* is marked, you will be prompted to relocate the text when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

```
> TEXTEDIT
Select a text:
<Dialog box appears to edit the selected text line>
Insertion point:
```

---

See also:

TEXT



RTEXT  
FONT

## Command MTEXT



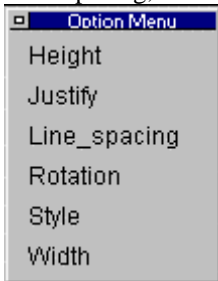
### Command MTEXT

Alias MT

#### Located:

1. Pull-down Menu: Detail – Multiline Text
2. Palette: Detail
3. Or Enter Command or Alias at the Command Line

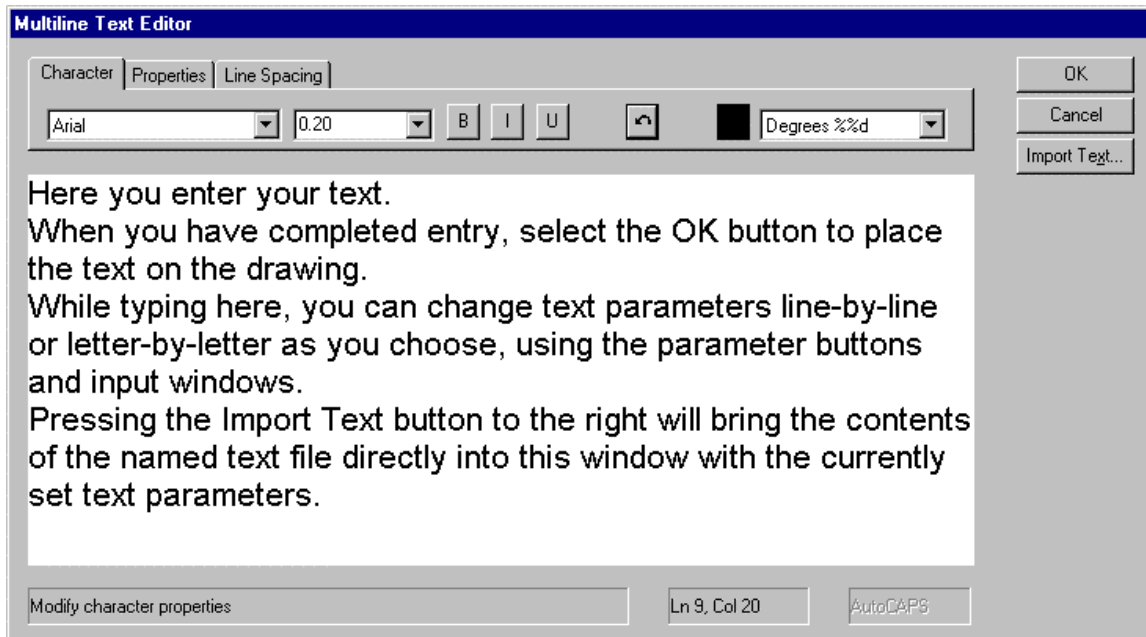
The MTEXT command provides AllenCad with full multiline text functionality. After issuing the command you are asked to specify the location of the new text by picking an upper left and lower right corner on the drawing. This window roughly locates the placement for the text but is essentially used to determine the line length. Text will be wrapped into the next line if the available width is insufficient. While specifying the two window points, you are presented with options to change Height, Justification, Line Spacing, Rotation, Style and Width.



You may choose to set these values here, however these options are also available in the dialog that follows.

Next, you are presented with a windowed dialog box in which you enter your text. Three tabs allow you to exercise control over all text parameters and a **Text File Import function** enables you to import an external text file into the window. These duplicate and add to the earlier options that were presented and you can set your preferences in either manner.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's MTEXT command dialog window, either through picking on the window and using Windows' "Ctrl V" function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry



#### MTEXT Dialog showing Character Options.

**Note:** to allow the text to fill the indicated space in your drawing and to line wrap as needed within that designated width, be sure to continue the line entry beyond the visual extents of the entry window. Pressing the Enter button as we have done in the above example will allow you to see all the text without scrolling left-right, however it may not give the desired effect when this text is transferred to the drawing.

Most of these windows and buttons you are probably fairly familiar with if you do much word processing. Viewed left to right, these windows and buttons:

1. Specifies the required font
2. Toggles between bold and regular type
3. Italicizes text
4. Underlines Text
5. Removes the latest entry or reverses the latest action
6. Indicates the color to be applied to text. Click on this to change the setting
7. The last button allows you to call up special symbols (diameter, degree, plus/minus)



#### MTEXT Dialog showing Property Options.

Here, you can specify a named text Style, set Justification, override text Width originally set using the picked window points, and finally, Rotation, where you can specify an angle or bearing at which to place the entered text.



#### MTEXT Dialog showing Line Spacing options.

Here you can specify line spacing distance -either a minimum (At Least) or (Exactly)

See Also MTEXTEDIT , QMTEXT

## Command MTEXTEDIT

**Command:** MTEXTEDIT



**Alias** MTE

### Located:

1. **Pull-down Menu:** Detail – Edit Multiline Text
2. **Palette:** Detail
3. **Or Enter Command or Alias at the Command Line**

This command predictably is used for editing multi-line text. When the command is issued, you are prompted to select the multi-line text that requires editing. This command will only work on text that has been entered with the MTEXT command. You must use TCORRECT or TEXTEDIT on other text entities.

When you select an MTEXT entity, the full multi-line entity is brought into the MTEXT dialog window, where you can edit any of the parameters that were originally available when the text was first created.

## Reference Text...



**Command:** RTEXT

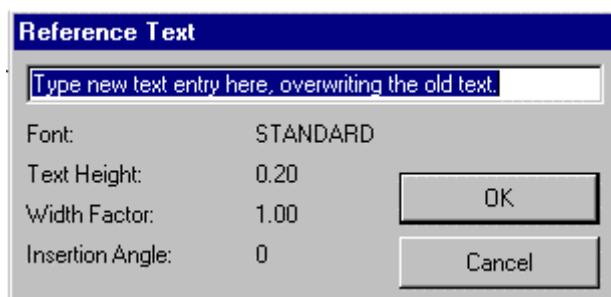
**Alias** RX

### Located:

1. **Pull-down Menu:** Detail – Reference Text
2. **Palette:** Detail
3. **Or Enter Command or Alias at the Command Line**

This command will permit you to select regular text entries (not mtext entries) and enter new text with the same characteristics as the text entry selected.

When you issue the command, you will be asked to select the text that you want duplicate the properties of. Once you have done this, the following dialog will appear:



Simply overwrite the text that appears in the dialog box and press OK. Lastly, you will be asked to specify the location of the new text entity in the drawing. Simply drag the text to the location required and click to anchor it. The new text will have the same font, height, width factor, and rotation angle as displayed in the dialog, along with the same layer and color.

The command scheme is:

```
> RTEXT
Select reference text:
<Dialog box to edit text>
Insertion point:
```

---

**See also:**

TEXT

Correct Text



**Command: TCORRECT**

**Alias TR**

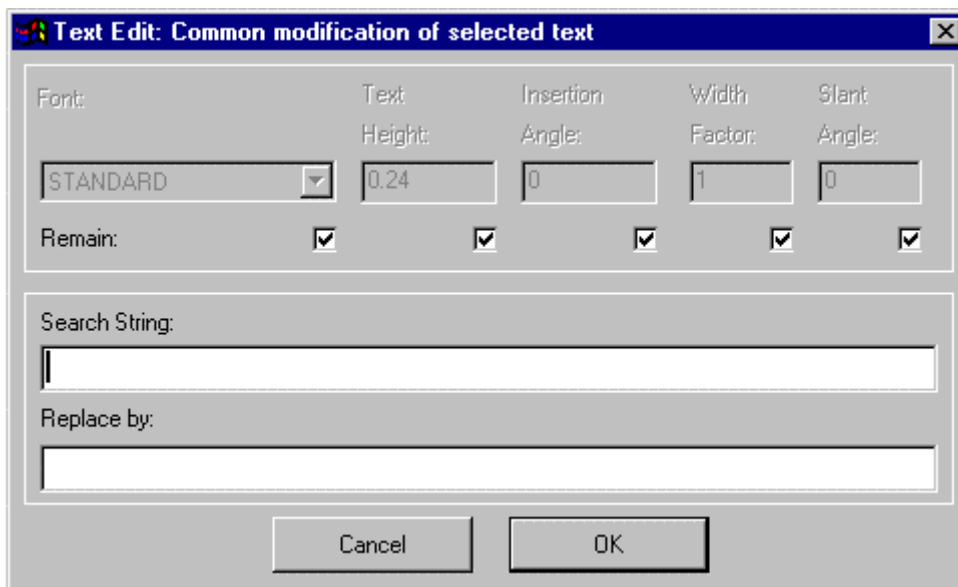
**Located:**

1. **Pull-down Menu: Detail – Correct Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

The function **Correct Text...** allows the user to edit multiple text entities and attribute definitions. It is possible to correct texts or text attributes as well as to change the font or the point size.

Call the function Correct Text by keyboarding TCORRECT or select Correct Text... from the pull-down menu *Detail*.

First, you are requested to select the text lines or attribute definitions. You may use the usual object select functions. Finish the object selection by ENTER.



As can be seen from the dialog above, this command will replace all instances of an existing text string in the selection set, with a replacement string. Alternately, or in addition, you can change existing text

parameters in the selection set by un-checking the appropriate "Remain" box and entering new values.

---

**See also:**

TEXTEDIT

**Font... (Text Style Definition)**



**Command: FONT**

**Alias FT**

**Located:**

1. **Pull-down Menu: Detail – Fonts**
2. **Palette: Detail**
3. **Or Enter Command or Alias Directly at the Command Line**

The command FONT loads fonts from the font files and customizes them to the requirements of the drawing.

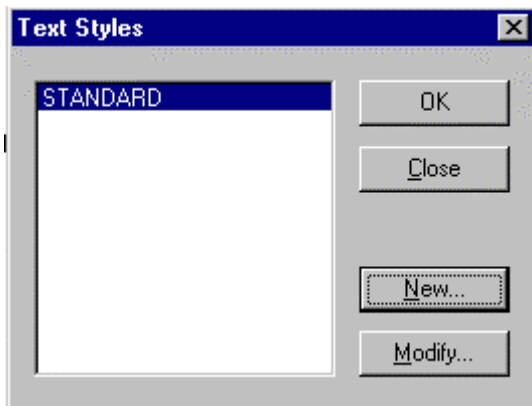
Fonts define the shapes of the text characters that make up each character set. In AllenCad, you can use:

1. AllenCad's own compiled shape (FSH) fonts,
2. AutoCAD's own compiled shape (SHX) fonts
3. True Type fonts.

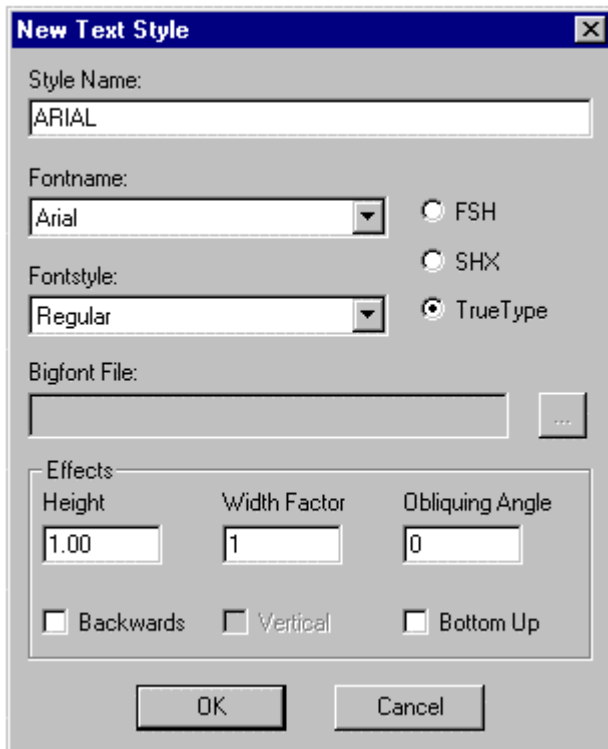
All installed True -Type fonts, provided with your Windows system, are listed by font family name.

SHX fonts and FSH fonts are listed by the name of the file in which the fonts are stored.

All fonts to be used in the program must first be assigned to a text style. When you give the FONT command, you are therefore presented with the following dialog:



Here, you create a new text style or choose an existing one to modify, by assigning a different font. With the required style name highlighted, press OK and you will be presented with the font dialog:



In this case, we have chosen to create a new style called Arial and to apply the Arial true type font to this style. To do this, we click on the True Type radio button and select the down arrow on the Fontname box to display the available choices. We scroll to Arial and select it.

Next, we may need to select a fontstyle. Again we click on the down arrow in the fontstyle window and choose regular text, as we do now wish it to be bold, italic or otherwise.

If we wish to use a Bigfont file, we press the browse button to the right of the window and select our bigfont file from the available choices.

Other text effects are available at the bottom of the dialog. We can assign a text height, width factor or Obliquing (Tilting) Angle. If we want the text to run right to left in a mirror image, we can select the Backwards option. The Bottom Up option will give us text that is upside down. .

**Note:** Some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

Confirm the settings of the properties by selecting OK. Fonts added or changed in this manner can now be used for creating or editing text objects when the modified text style is current.

**Note:** Defined text styles are saved with the drawing file only with a reference to the corresponding font file. They are available any time that the drawing is opened, if the corresponding font file is found in the programs directories.

---

#### See also:

Adding Additional Fonts

Command QFONT

Command TEXT

## Compiling a Source Shape File to a Binary Font File

**Command: FCOMPILE**

**Alias FC**

**Located:**

1. **Pull-down Menu: Detail – Compile Font**
2. **Or Enter Command or Alias Directly at the Command Line**

This program uses binary font files that permit defining text styles in the drawing (see command FONT).

Defined text styles can then be set and used with the commands TEXT, TEXTEDIT, ATTDEF, ATTDEDIT.

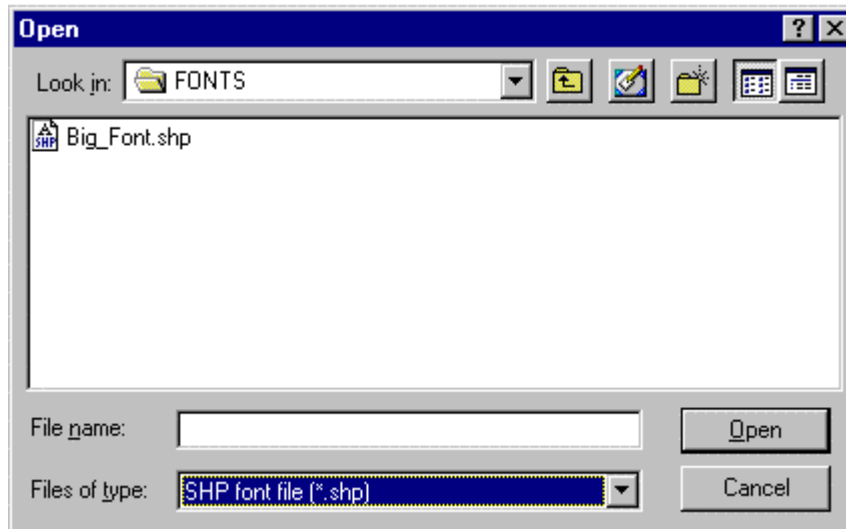
Binary font shape files (*.fsh*) provide less loading time and improved memory usage. A number of these FSH files are delivered with the program.

Application-specific or customized font shapes can be defined in plain text in **.shp** or **.fnt** shape files.

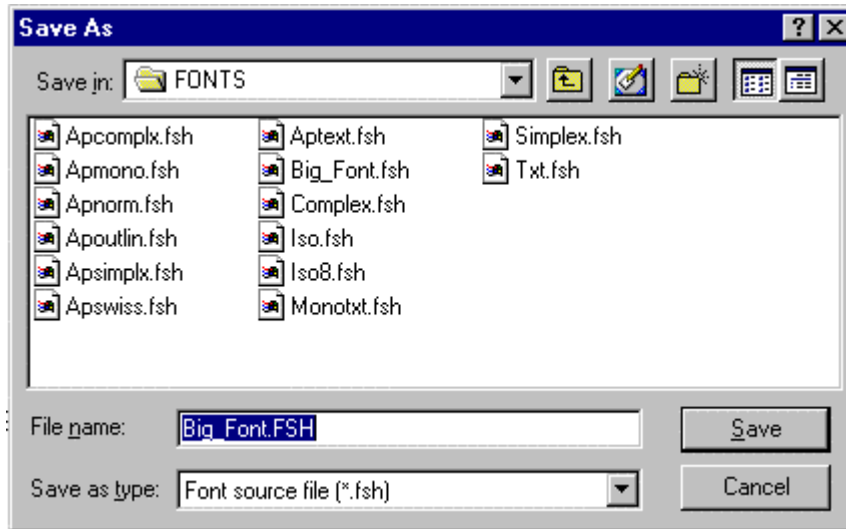
Before they can be used in a drawing, **.shp** or **.fnt** files must first be compiled.

**To convert a source shape file to a binary font file**

1. Call the command FCOMPILE



2. Specify the *.shp* or *.fnt* file name in the file dialog box



3. In the next dialog box, enter the name of the *.fsh* binary font file to be created for later use.

---

**See also:**

FONT  
QFONT

## Text Style Definition (Command Line Version)

**Commands:** QFONT or STYLE

**Alias** QF

**Located:**

**1. Entered Directly at the Command Line**

The command QFONT, or STYLE, allows the user to load text styles from the specified font file and customize it to individual requirement.

These commands are command line variants of the FONT command.

```
> STYLE
Style name (or ?) <STANDARD>:
New text style definition.
Font file <TXT.FSH>:
Text height <0.00>:
Text style width factor <1.00>:
Text style obliquing angle <0>:
Backward alignment (Yes or No) <N>:
Upside-down alignment (Yes or No) <N>:
DIM set as current text style.
```

---

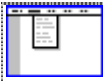
**See also:**

Command FONT

QFONT and STYLE are the command line versions of the FONT command.

## Dimensioning Variables





## Dimensioning

For detailed information, select one of the following items:

### Dimension Drawing Commands:

DIMLIN	Linear Dimension (Combination between DimHor and DimVer)
DIMHOR	Horizontal Linear Dimension
DIMVER	Vertical Linear Dimension
DIMALI	Aligned Linear Dimension
DIMROT	Rotated Linear Dimension
DIMORD	Ordinate Dimension
DIMDIA	Diameter Dimension
DIMRAD	Radius Dimension
DIMCENT	Center Mark
DIMA3P	Angular Dimension 3 Points
DIMA4P	Angular Dimension 4 Points
DIMSER	Serial Dimensioning
DIMCON	Continuing Dimensioning
DIMBAS	Baseline Dimensioning

### Dimension Editing Commands:

DIMUPD	Update Dimensions
DIMOBL	Dimension Orientation Editing
DIMTNEW	Edit Dimension Text
DIMTMOVE	Move Dimension Text
DIMTROT	Rotate Dimension Text
DIMTHOME	Re-Align Dimension Text

### Dimension Style Commands:

DIMTYPE	Modify Dimension Type
DIMSAVE	Save Dimension Type
DIMREST	Restore Dimension Type

Angle Dimensioning (3 Points)

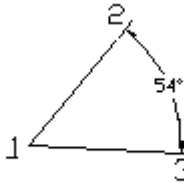


**Command: DIMA3P**

**Alias D3P**

**Located:**

1. **Pull-down Menu: Dim – Angular 3 Points**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Angular- 3-Point

This function uses 3 measurement points that are placed at the apex and legs of an angle. This function not only allows the user to calculate the interior angle between the two legs, but also the exterior angle.

The user will be asked to select the vertex position. Either type in the coordinates, or select with the cursor. It is suggested that the user use the Object Snap options whenever applicable.

Define a point on each of the legs of the angle.

Define the location of the dimensioning arc by selection with the cursor. The dynamic preview function will assist the user in gaining a visual impression of the result. The position chosen for the dimensioning arc will also decide that angle, (interior or exterior) is to be dimensioned.

After confirmation, the program will display the measured value. This can be confirmed with RETURN, or amended as required.

```
> DIMA3P
Identify vertex: end
of object: P1
First point: P2
Second point: P3
Position dimension arc: P4
Dimensioning text <60>: RETURN
```

Angle Dimensioning (4 Points)

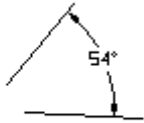


**Command: DIMA4P**

**Alias D4P**

**Located:**

1. **Pull-down Menu: Dim – Angular 4 Points**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



#### Angular 4-Point

The difference between these two angle functions is that the angle 4P dimension is based on four points of measurement. These could be placed at the end points of two lines, or at the tangent of the end points of an arc.

Both the interior and exterior angles can be measured and dimensioned.

The first prompt will ask for the appropriate objects. (lines, polylines, arcs or circles.)

```
> DIMA4P
Select line, polyline or arc: P1
Select line, polyline or arc: P2
Dimensioning text <60°>: RETURN
```

Note: It is not possible to measure the angle between two parallel lines.

#### Command DIMALI



#### Command DIMALI

Alias DAL

#### Located:

1. **Pull-down Menu: Dim – Aligned**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias at the Command Line**



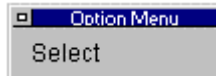
Aligned

Aligned dimensioning is a type of linear dimensioning that measures and labels the absolute distance between two points, regardless of the relative position of the axes. This makes it possible to dimension lines, edges and spaces etc. that are not located on the principal axes of the coordinate system (IE. not parallel to x, y or z-axes).



Aligned

The user will be asked to define two points of measurement (DEFPOINTS). As an alternative the program offers the user the chance to select a specific dimensioning object function. Activate this function by



choosing Select from the options bar. This allows lines, polylines, circles and arcs to be selected. Point to the desired object with the cursor. In the case of lines, polylines and arcs the start and end points of the object will be interpreted automatically and used for the measurement calculations. The first point of a circle will be defined as a point situated along the perimeter of the circle. The second point will be obtained by drawing a straight line from the first point, through the center point of the circle, to the opposite side of the perimeter of the circle.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line. This can be done either by entering the relevant coordinates, or with the cursor. The absolute measurement between the two points will be displayed. Pressing the RETURN key will accept the calculated value. The dimension will be inserted into the drawing and shown on a horizontal plain.

> DIMALI

1st Point: P1

2nd Point: P2

Dimension line positioning: P3

Dimension text <2.5000>: RETURN

Or

> DIMALI

st Point: SELECT <CR>

select line, polyline, circle, arc: P1

Dimension line positioning: P2

Dimension text <2.5000>: RETURN

## Baseline Dimensioning

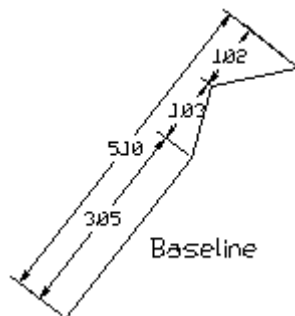


**Command: DIMBAS**

**Alias DBA**

**Located:**

1. **Pull-down Menu: Dim – Baseline**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



**Baseline** adds a second measurement line to the drawing, parallel to the already existing dimensioning lines. This allows the user to read the total dimension between two end points, extending over a number of individual dimensions within a chain.

This type of dimensioning assumes that a form of linear dimensioning already exists. The appendixes dimension will inherit the same properties as the existing measurements.

With the cursor, mark the starting point of the dimensioning chain, taking care to utilize the same direction of measurement as laid down during the creation of the individual dimensions within the chain.

Select the second point to be used for the baseline dimension by entering the coordinates at the keyboard or with the cursor. Whenever possible enlist the help of OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as this second line will automatically be drawn at a given distance from the existing dimensioning lines.

```
> DIMBAS
Select a dimensioning line: P1
Second Point: P2
Dimensioning text <20.50>: Return
```

## DIMCENT



**Command:** DIMCENT

### Located:

1. **Pull-down Menu:** Dim – Center
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**

This command places a cross at the radius point (center) of selected arcs or circles.

## Continuing Dimensioning

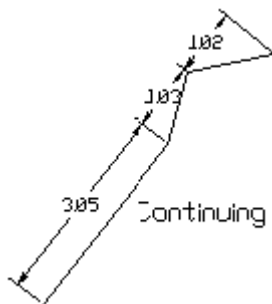


**Command:** DIMCON

**Alias** DCN

### Located:

1. **Pull-down Menu:** Dim – Continuing
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**



**Continuing** dimensioning is a variation of linear dimensioning in which existing dimensioning is continued or extended. Here a second dimension is linked to an existing dimension so as to create a dimensioning chain. Continuing dimensions can only be created, when a drawing already contains an existing linear dimension that can be used as a basis for the chain.

A continuing dimension always inherits the same properties as the existing dimensioning. For this reason it is only possible to connect dimensioning of the same type. For instance, a new horizontal dimension can

only be added or linked to an existing horizontal dimension.

A message will appear in the text window requesting the selection of an existing dimension. Select the desired dimension, taking care to place the selection cursor as close as possible to the point at that the continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the continuing dimension will be the point that lies closest to the selection point at the time the object is selected

Choose the second point by entering the relevant coordinates, or by selecting with the cursor. Whenever possible, employ the object snap-to function.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as this will automatically be placed in line with the existing dimensioning.

```
> DIMCON  
Select a dimension: P1  
2nd Point: P2  
Dimension text <2.50>: RETURN
```

### **Diameter Dimensioning**

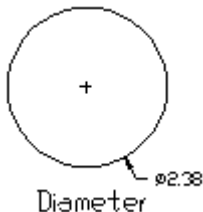


**Command: DIMDIA**

**Alias DDA**

#### **Located:**

- 1. Pull-down Menu: Dim – Diameter**
- 2. Palettes: Create Dimensions and Dim**
- 3. Or Enter Command or Alias Directly at the Command Line**



Diameter dimensioning is used to display dimensions of circles and arcs. Unlike the horizontal or vertical dimensioning functions, no measurement lines are shown. In this case a single extension line is used to show the relationship between the value and the object itself. The diameter symbol signifies that this is a special dimensioning feature used only in circle and arc dimensioning.

The first prompt will ask for the specific circle or arc. Select with the cursor. Please note that the point at that the user selected the object will be defined as the point at that the dimensioning extension line will connect to the element.

QUADRANT OSNAP will help the user to select specific points on the circle or arc. After selection, the program will display the corresponding diameter measurement. This can be confirmed with the RETURN key or amended as required.

The second prompt will ask for the position of the extension line. Select a coordinate at the required distance from the object

The alignment of the extension line will have been fixed by the position of the two selected points and cannot be altered. The extension line will be drawn along an imaginary straight alignment that extends

from the middle point of the object to the measurement point as defined during object selection (DEFPOINT).

The extension line and measurement text will be displayed and a cross (+) will automatically appear at the center of the circle of arc.

```
> DIMDIA
Select circle or arc: P1
Dimensioning text <24.00>: RETURN
Extension length: P2
```

## DIMEXO

Real

Determines an offset of extension line starting point from the measured entity.

### Horizontal Dimensioning

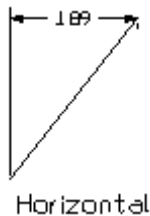


**Command: DIMHOR**

**Alias DH**

#### Located:

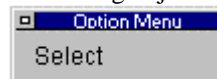
1. **Pull-down Menu: Dim – Horizontal**
2. **Palette: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Draws horizontal dimensions. Horizontal dimensioning is linear dimensioning that both measures and labels the distance between two points parallel to the X -axis.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button . Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the x-axis and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMHOR
1. Point: P1
2. Point: P2
```

Dimension line positioning: **P3**  
Dimension text <2.50>: **Return**

or

> DIMHOR  
1. Point: **Select**  
Select line, polyline, circle, arc: **P1**  
Dimension line positioning: **P2**  
Dimension text <2.50>: **Return**

## Linear Dimensioning

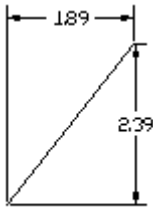


**Command: DIMLIN**

**Alias DL**

### Located:

1. **Pull-down Menu: Dim – Linear**
2. **Palette: Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

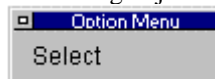


Linear

The DIMLIN command creates a horizontal or vertical dimensioning depending on the position of the dimension line. This command is a combination between both orthogonal modes horizontal and vertical.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button . Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points is measured along the X-axis for horizontal and Y-axis for vertical dimensioning and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

> DIMLIN  
1. Point: **P1**  
2. Point: **P2**  
Dimension line positioning: **P3**  
Dimension text <2.50>: **Return**

or



> DIMLIN  
 1. Point: **Select**  
 Select line, polyline, circle, arc: **P1**  
 Dimension line positioning: **P2**  
 Dimension text <2.50>: **Return**

## Modify Dimension Orientation



**Command: DIMOBL**

**Alias DOB**

### Located:

1. **Pull-down Menu: Dim – Modify Orientation**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to modify the orientation of existing linear dimensions.

Call the command and enter the angle for the dimension orientation. Select one or several dimensions, to be modified in its orientation.

## Ordinate Dimensioning

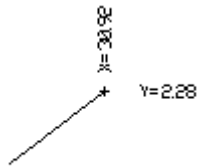


**Command: DIMORD**

**Alias DOD**

### Located:

1. **Pull-down Menu: Dim – Update Dimension**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



### Ordinate

Ordinate can be used to determine the X and Y coordinates of a point and enter it into the drawing. These coordinates can either represent a point within the drawing, or be that of a geometrically significant point of a drawing entity.

The X and Y options will be displayed within the options bar. By selecting one of the options the user can determine whether the X or Y coordinate value is to be defined. Click the corresponding button, bearing in mind that the X value is set as the default and will therefore automatically be used if no other option is selected.

The user will then be prompted to select the point to be measured. Type in the coordinate values, or select with the cursor. If the location represents a specific point on an entity, enlist the help of OSNAP.

The program will ask for LEADER LENGTH; this requires that the user enter the EXTENSION LINE position and also determines the physical location of the measurement value text. The extension line

originates at the point to be measured and connects to the measurement value text.

Use the cursor to select the position at that the measurement value is to be displayed. This figure can be positioned at any alignment and distance from the dimensioning line. To help, the program draws a line from the selected coordinates to the current cursor position. This illustrates the distance and alignment between the two positions. Select the desired position with the cursor.

If the dimension is not placed horizontally or vertically in relation to the measurement point, then the extension line will appear in orthographic mode and not as a straight line. An example of this is shown in the above diagrams.

The program will display the calculated coordinate. The value can be confirmed by pressing the RETURN key, or amended as required. The user may type in any alphanumeric sequence, thus enabling the point to be titled as required.

```
> DIMORD
Coordinate point: X
Coordinate point: P1
Extension length: P2
Dimensioning text <20.00>: RETURN
```

## DIMPOST

String (Default: "")

Determines a text prefix, suffix or both, for dimension label created with following dimension commands. These strings surround the dimension measurement value computed. The string <> used in the variable setting serves to separate prefix and suffix specification; otherwise the string supplied is used as suffix. Example: A variable value of "Approx. <> m" might result in the dimension text *Approx. 12 m*

### Radius Dimensioning

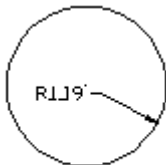


**Command: DIMRAD**

**Alias DRA**

#### Located:

1. **Pull-down Menu: Dim – Radius**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Radius

Radius dimensioning is carried out in the same way as diameter dimensioning. The only difference is that a radius dimension is calculated instead of a diameter measurement.

```
> DIMRAD
Select circle or arc: P1
Dimensioning text <24.00>: RETURN
Extension length: P2
```

## Restore a Dimensioning Style



## Command: DIMREST

Alias DRE

### Located:

1. **Palette: Dim**
2. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to reinstate a stored style file so that it can be reinstated as the current style. All of the stored style settings will be retrieved and used by the system for future dimensioning. Any dimensions that were created prior to the definition of the new style can also be updated (DIMUPD) so that they, too, adhere to the conventions of the new style settings.

This is done by using the DIMUPDATE command, explained in the following section.

Restoring a dimensioning style is achieved up by entering the DIMREST command.

The program will follow this command by displaying the name of the current dimensioning style in the text window. The user will then be requested to choose the dimensioning style that is to be used as the current default. The program options bar offers the user two methods by that to select the required dimensioning style.

- Either type in the name of the stored dimensioning style, remembering that the user can list all of the dimensioning styles already contained within the particular drawing; or
- Use the cursor to select the desired dimensioning style.

### Select the Style by Typing the Name.

Enter the style name at the appropriate input field. By clicking the list button shown in the options bar the user can display all of the available dimensioning styles.

In order to achieve a clearer overview, it is possible to list all of the file names that begin with a particular character chain by entering the chain at the input field. The standard DOS default \* can also be used to substitute a complete character chain.

When used in a search routine this will cause all of the existing file names to be listed. If the user wishes to search for specific dimensioning style files, type in the desired character chain followed by the \* character. For example, ar\* would result in a list of all files beginning with AR.

```
> DIMREST
Current dimensioning style: Architecture03
Name of the dimensioning style: List <?>
List dimensioning styles <*>: arch*
ARCHITECT01
ARCHITECT02
ARCHITECT03
```

Listing the existing dimensioning styles will complete the **Save Dimensioning Style** function. Pressing RETURN will restart the option and save the current dimensioning settings.

```
> DIMREST
Name of the dimensioning style: Architektur03
```

### Selection by Pointing

After the program has displayed the current dimensioning style, choose the **SELECT** option field and choose the desired dimensioning style with the cursor. The program confirms the user selection and verifies it by displaying the new dimensioning style in the text window.

```
> DIMREST
Current dimensioning style: Architecture
Name of the dimensioning style: Select
Select a dimension: P1
```

---

**See also:**

**Command DIMTYPE**

**Rotated Dimensioning**



**Command: DIMROT**

**Alias DRO**

**Located:**

1. **Pull-down Menu: Dim – Rotated**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Rotated dimensioning is also a type of linear dimensioning that measures the distance between two points, at a specified angle.

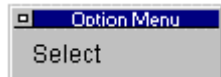
When dimensioning the distance between two points (or an object with linear extension) at an angle greater or less than 90°, the space allocated to the dimension will always be smaller than the absolute dimension measurement. This procedure is equivalent to looking at an object from a specific angle and establishing the visible length from a different viewpoint.

The user will be asked to define the angle at that the dimensioning is to appear in relation to the axial alignment of the coordinate system. This can be done by entering the coordinates at the keyboard, or by using the cursor to click on two points.

The first figure entered defines the angle between the direction of the negative Y-axis and the first dimension line as seen in a clockwise direction. The apex of this angle is determined by the position of the first point of measurement.

If the user defines the angle at that the dimensioning is to take place by selecting two separate points, then a perpendicular line will be drawn at 90° to the extension lines.

After having determined the dimensioning angle, the user will be asked to define two points of measurement (DEFPOINTS). As an alternative, the program offers the user the chance to select a specific

dimensioning object by clicking the Select function from the Options Bar . This will allow lines, polylines, circles and arcs to be selected. Point to the desired object with the selection cursor.

The start and end points of a line, polyline or arced object will be interpreted automatically and used for the measurement calculation.

The first point of a circle will be defined by the program as the next point situated along the circle perimeter. The second point is obtained by drawing a straight line from the first point, through the center

intersection of the circle and on to the opposite side of the circle perimeter.

In contrast to objects that have a linear extension, this method will always measure and display the exact diameter of a circle. Only the position of the dimensioning extension lines will be affected when using rotated dimensioning.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line.

The distance between the two points will be displayed at the designated angle orientation. Pressing the RETURN key will accept the calculated value and cause it to be inserted horizontally into the drawing.

```
> DIMROT
Angle of rotation <0>: 60
First Point: P1
Second Point: P2
Dimension line positioning: P3
Dimension text <2.50>: RETURN
```

or

```
> DIMROT
Angle of rotation <0>: 60
First Point: Select
Select line, polyline, circle or arc: P1
Dimension line positioning: P2
Dimension text <2.50>: RETURN
```

## Command DIMSAVE



**Command: DIMSAVE**

**Alias DS**

### **Located:**

- 1. Palette: Dim**
- 2. Or Enter Command or Alias Directly at the Command Line**

The DIMSAVE command allows the user to combine the individual settings or parameters and store them in a named dimensioning style. This facility enables the user to store and recall a number of different styles, each of that can be used for an individual working environment.

Architectural dimensioning, tolerance dimensioning or country and language specific options are examples of this (such as for individual units of measurement or labeling).

The use of pre-defined dimensioning styles removes the need to constantly change the dimensioning settings, simplifies the actual drawing procedure and in general makes the job of dimensioning easier.

It should be noted that dimensioning styles can only be stored together with the current drawing and are, therefore, only available for use from within that drawing. In order to use pre-defined dimensioning styles in other drawings, they should be created and stored in one of the template drawings, or loaded from a pre-defined prototype drawing.

### **Procedure for Saving**

Select the command **Modify Dimension** from the menu *Detail*. From the palette, select the icon representing the Dialog Box *Dimension*. Enter the proper settings and values from this dialog box. In the first field in this box, name the dimensioning template with a descriptive name.

Before entering the name of a dimensioning style, it is possible to list the file names that have already been allocated. To do so type a question mark (?) in the input field and press RETURN.

Dimensioning style names may be up to 31 characters long and can consist of any alphanumeric input

except special symbols. Up to a maximum of two umlauts can be used in any one name, but these should be avoided if at all possible. No differentiation is made between upper and lower case characters as all entries are automatically converted to upper case.

Confirming the selected name will cause the dimensioning style to be stored.

In order to achieve a clearer overview, it is possible to list all of the file names that begin with a particular character chain by entering the chain at the input field. The wildcard pattern option \* can also be used to substitute a complete character chain. When used in a search routine this will cause all of the existing file names to be listed. If the user wishes to search for specific dimensioning style files, type in the desired character chain followed by the \* character. For example, ar\* would result in a list of all files beginning with AR.

```
> DIMSAVE
Name of the dimensioning style or ? for a complete list: ?
List of dimensioning styles <*>: ar*
ARCHITECTURE01
ARCHITECTURE02
```

Listing the existing dimensioning styles will complete the save dimensioning style function. Pressing RETURN will restart the option and save the current dimensioning settings.

```
> DIMSAVE
Name of the dimensioning style, or ? for a complete list:
ARCHITECTURE03
```

---

See also:

**Command DIMSTYLE**

## **Serial Dimensioning**

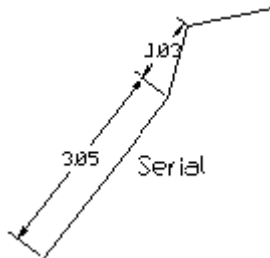


**Command: DIMSER**

**Alias DSR**

### **Located:**

- 1. Pull-down Menu: Dim – Serial**
- 2. Palettes: Create Dimensions and Dim**
- 3. Or Enter Command or Alias Directly at the Command Line**



Serial dimensioning is carried out in the same way as described above for Continuing dimensioning. However, where only one additional dimension can be added to an existing dimension with Continuing dimensioning, Serial dimensioning enables the user to link as many dimensions as required. This assumes that at least one dimension already exists that can be used as a basis for the rest.

It is only possible to add dimensioning of the same type since the additional dimensioning will inherit the properties of the existing dimensioning:

Select the dimensioning that is to act as the basis for the series, taking care to place the selection cursor as close as possible to the point at that the Continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the series dimensioning will be the point that lies closest to the click point at the time of object selection.

In addition to Select and End that are the same for Serial dimensioning as for Continuing, two other selections are possible from within the series dimensioning function bar:

#### **Continue and Baseline.**

1. **Continue** allows the user to link one additional dimension and should be selected when creating serial dimensioning. As this is the default option it only needs to be confirmed with the RETURN key. Alternatively, click the appropriate button as displayed in the options bar.
2. **Baseline** allows the user to create baseline dimensioning, parallel to the existing linear dimensioning. This function is described in the next section. Select the second point by typing in the coordinates or selecting with the cursor. Whenever possible use OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as it will automatically be placed in line with the existing dimensioning.

```
> DIMSER
Select a dimension: P1
Series <continuous> RETURN
2nd Point: P2
Dimension text <2.50>: RETURN
Series <continuous>: RETURN
2nd Point: P2
Dimension text <2.50>: RETURN
...
ESC
```

By re-selecting the **Continue** option, it is possible to repeat the above steps for as many times as dimensions are required. The creation of additional dimensioning can be terminated by pressing the ESC key.

### **Undo Repositioning**



**Command: DIMTHOME**

**Alias DTH**

#### **Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Realign DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This command undoes the moving of the dimensioning text (done with DIMTMOVE) and returns it to the original position.

After calling the command, select the dimensioning objects that dimensioning text was moved.

Close the object selection by pressing RETURN.

---

#### **See also:**

Move Dimensioning Text

**Command DIMTMOVE**



**Command: DIMTMOVE**

**Alias DTM**

**Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Move DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to move the dimensioning text of existing dimensioning objects. It doesn't move the associated dimension arrows.

After calling the function, select a dimensioning object. Move the dimensioning text by "pointing" a new insertion point for the text object.

---

**See also:**

Undo Repositioning

Command DIMTNEW



**Command: DIMTNEW**

**Alias DN**

**Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Modify Dim Text Value**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This command allows the user to edit and modify the dimensioning text value.

After calling the function, enter the new dimensioning text value. Select one or several dimensioning objects to be changed.

Press RETURN to close the object selection. The dimensioning text of the selected objects will be modified accordingly.

**Rotate Dimensioning Text**



**Command: DIMTROT**

**Alias DTR**

**Located:**

1. **Pull-down Menu: Dim – Modify Dimension Text – Rotate DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

Using this function, it is possible to rotate the dimensioning text of a previously generated dimension.

After calling the function, first enter the rotation angle, then select the one or more dimensioning text objects. Press RETURN to finish the object selection.

The selected dimensioning text will be rotated around its insertion point.

**Dimensioning Styles**





**Command: DIMTYPE**

**Alias DDR**

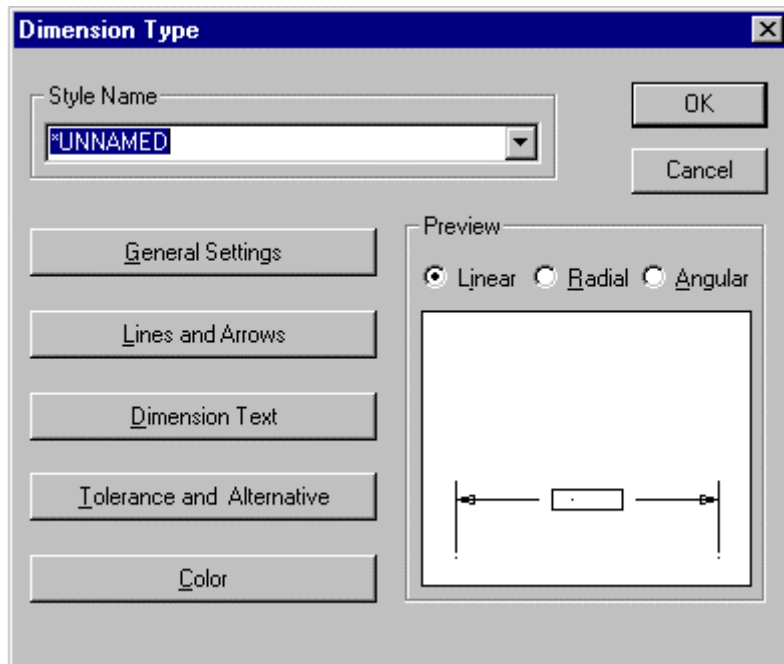
**Located:**

1. **Pull-down Menu: Dim – Dimension Type**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

Dimensioning format is controlled by the Dimensioning dialog box. Use the style name, input and selection dialog to either store the current settings as a dimensioning style, or to select a pre-defined style as the default.

The buttons described below each open additional dialog windows in which individual dimensioning parameters can be defined.

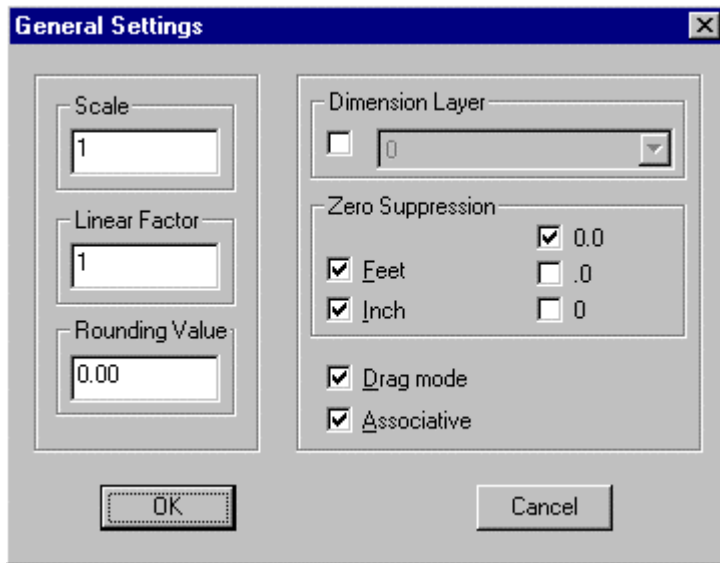
### **Dialog Box Options:**



When you issue the DIMTYPE command, you are presented with the opening dialog. Here, you see five bars on the left side that lead to specific configuration areas. On the right, you can click on the Linear, Radial and Angular radio buttons, to see a representation of current settings in the window below, for each dimension type.

Clicking on each bar will produce the dialogs detailed below:

### **General Settings**



### Scale

Scale value enlarges or reduces the size of all dimensioning elements (apart from the dimension value itself) and allows the dimensioning elements to be scaled to suit the aspect ratio of the drawing.

### Linear factor

Linear factor converts the measured drawing units into a dimension figure. The default value is 1.

### Precision

Precision defines the number of decimal places to which the measurement will be carried.

### Dim. Layer

The layer in which the dimensioning values are stored is identified by the name entered in the dimensioning layer selection field. Select the layer required, or enter a new layer name. It is prerequisite that the option has already been activated by clicking the appropriate control box. A check mark indicates the current status of the option.

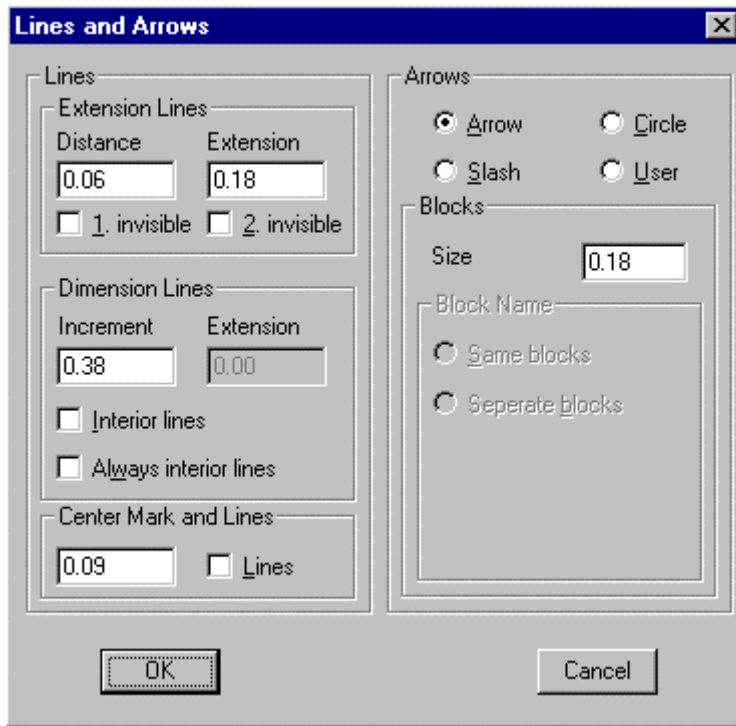
### Zero Suppression

The Zero suppression field decides the treatment of the display mode for zeroes.

### Drag mode/ Associative

The last two control boxes activate or deactivate either the "rubber band" mode and / or the associative character of the dimensioning.

### Lines and Arrows



Extension lines are the lines that extend the object dimensions outside the perimeter of the measured object, assuming that dimensioning is to take place outside of these perimeter constraints. These lines, and the nature of the arrowheads to be used, are defined in the Dialog Box *Lines and Arrows*.

## Lines

The fields in this area decide the characteristics of the extension and dimension lines.

### Extension Lines

This area defines the spacing required for the extension lines.

**Distance** allows the user to choose the space between the object and the point at that the extension lines begin. A value = 0 indicates that the lines will start directly on the object itself.

**Extension** dictates how far the extension lines should extend past the dimensioning line.

The control boxes marked **Invisible** can be used to suppress either one or both of the extension lines in the drawing.

### Dimension Lines

Dimension lines either connect the object edges, or the extension lines of the object that is to be measured. The dimensioning text is written along this line.

**Increment** sets the spacing between successive dimensioning lines that use the same extension lines.

**Extension** allows the user to enter the measurement by that the extension lines should run past the dimensioning lines. This value can only be entered when the **Slash** option has been selected as the dimension arrow type.

The field **Interior Lines** or **Always Interior Lines** decides whether dimensioning lines should be allowed to extend past the extension lines.

### Center Mark and Lines

This area defines the existence and size of a cross-mark indicating a circle center point. It also defines whether or not dashes or strokes should be added to the center point cross or not.

## Arrows

This field allows the user to select the type of arrowhead to be used.

## Blocks

This area describes the length of the arrowhead. Only the size selection of this area is normally active. Block Name is active only when the arrow style selected is User. The size of the Block to be used is defined here.

## Block Name

Active only when the arrow style selected is User, this field decides the Block Name. The name commonly in the field is ARROW.

## Separate Blocks

This field either allows or suppresses the use of divided or separate arrow blocks. When activated, it is also possible to define that elements should be used at either end of the dimensioning line.

## Dimension Text

**Dimension Text**

Text position

☐ Text always inside

☐ Text outside, lines inside

Text always horizontal

☒ inside ☒ outside

Vertical text position

☐ above

☒ relative 0.00

Dimension Text Style

STANDARD

Dimension Text

Height 0.18 Factor 1

Distance to dimension line 0.09 ☒ Frame

Prefix <> Suffix

OK Cancel

These dialog window fields determine the position and look (size, style, spacing) of the dimensioning text.

## Text position, Text always horizontal, and Vertical text position

These fields serve to define the position of the text with respect to the dimension lines.

## Dimension text style, Dim text

These fields define the style and size of the text, and the distance separating text lines.

The prefix and suffix fields enable the user to type in a character string that will appear before or after the dimensioning text. For example, this field may be used to display the unit of measurement.

The <> character decides whether the character string will appear before or after the dimensioning text. For example, **approx. <> ft.** entered in this field would produce the following dimensioning : **approx. 12 ft.**

## Tolerance and Alternative Dimensioning

The tolerance settings control the tolerance, both the maximum and minimum values, allowed. The control fields activate or deactivate the display of these elements. A check mark in either of the control windows shows the current status of the individual options.

Values for the positive or negative tolerance are entered in the field marked Values. These fields also enable the user to type in a line of text (String).

The **Alternative mode** options control the use of a second, alternative dimensioning method. This mode, when activated, will allow dual dimensioning in metric and imperial values.

#### Factor

The conversion factor between the two units of measurement can be entered in the factor field.

#### Decimals

The number of decimal places selected will appear in the decimal display.

#### Alternative string

Alternative string should contain a description or title of the second unit of measurement.

#### Colors:

The dialog window allows colors to be assigned to the dimensioning lines, extension lines and dimensioning text (dimension value, alternative dimensioning, tolerances, limit values, unit of measurement etc.)

Select the appropriate element and allocate a color from the displayed selection box. 256 standard colors are available in addition to the BYLAYER and BYBLOCK options.

## DIMUPD



**Command: DIMUPD**

**Alias DU**

### **Located:**

- 1. Pull-down Menu: Dim – Diameter**
- 2. Palettes: Modify Dimensions and Dim**
- 3. Left SideBar**
- 4. Or Enter Command or Alias at the Command Line**

After changing the current dimensioning style, all new dimensions will be created in line with the new criteria. Using the Update Dimension function, it is possible to update all of the existing dimensions so that they comply with the new dimensioning style settings.

Using one of the object selection functions found in the options bar, mark or select the dimensions that are to be updated so as to reflect the new dimensioning style.

It is possible to update individual, group or all dimensions contained within a drawing at one time.

Confirm the selection process by pressing RETURN. The marked dimensions will then be updated to conform to the new dimensioning style. The program will display a x dimensions found notice telling the user how many dimensions have been updated.

The following dialog examples demonstrate the selection process and should clarify the use of the DIMUPD command.

> DIMUPD

Select Dimension(s): Fence

First Fence point: P1

Endpoint of the line: P2

Endpoint of the line: P3

RETURN

Select Dimension(s): All

2 Dimensions found

Or

> DIMUPD

Select Dimensions(s): All

Select Dimensions(s): RETURN

3 Dimensions found!

## Vertical Dimensioning



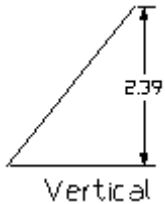
**Command: DIMVER**

**Alias DVR**

### **Located:**

- 1. Pull-down Menu: Dim – Vertical**

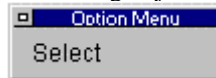
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Vertical dimensioning is linear dimensioning that measures and labels the distance between two points or the linear extension of an object parallel to the Y-axis.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the Y-axis and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMVER
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMHOR
1. Point: SELECT <CR>
select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.5000>: Return
```

## Crosshatching

Hatch...



**Command: HATCH**

**Alias H**

**Located:**

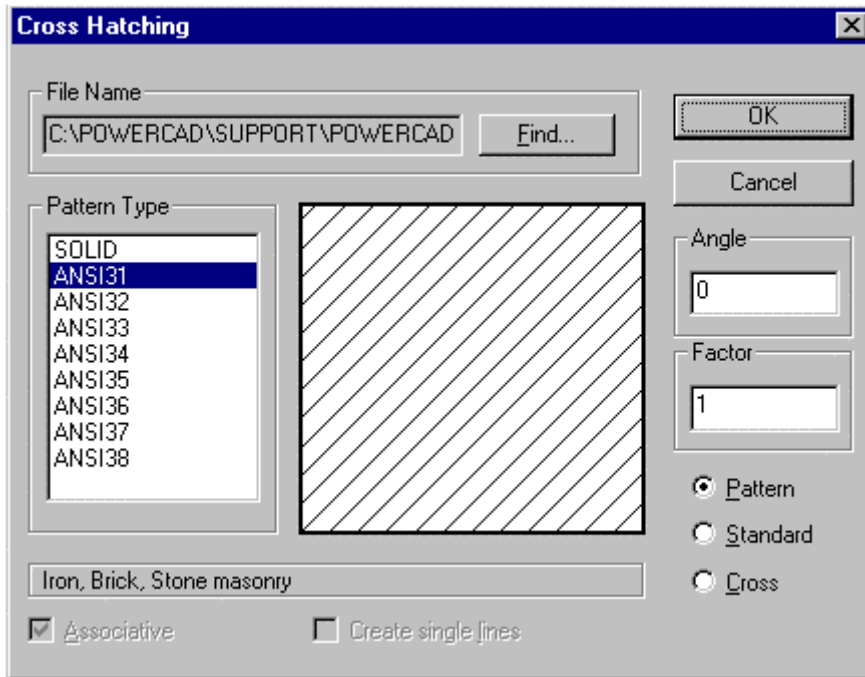
1. **Pull-down Menu: Detail – Hatch**
2. **Left Side Bar**
3. **Palette: Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

Hatching (or cross-hatching) a drawing adds meaning to the drawing, and helps to differentiate the materials, the areas, or any of a number of other unique attributes. Both the HATCH and the BHATCH commands enable you to fill in areas of your drawing with either a solid color or a distinctive pattern.

### Command Execution

To execute the command, you are first prompted for the boundary this is typically some closed entity, such as a rectangle, circle or closed polyline. **Note:** If only part of these selected entities forms the boundary, you should instead be using the BHATCH command.

Having selected the boundary, you are next presented with the dialog in which you select the pattern to apply, as well as pattern parameters:



Hatch patterns are stored together in **.pat** files. The file name input at the top of this dialog allows you to select a pattern from any of a number of .pat files that you may have available. Having selected the file, all patterns included in that file are listed in the Pattern Type window on the left. Clicking on any of these causes a representation of that pattern to appear in the central window, making selection easier.

The selections to the right on this dialog modify the pattern, in which:

- Angle – Will cause a counter-clockwise rotation of the pattern by the angle entered;
- Factor – Will cause the pattern to become more condensed or expanded, depending upon the entered factor value;
- Pattern will display a non-uniform pattern (if that is available in the particular selection);
- Standard – Will modify the pattern to a uniform spacing and
- Cross – Will create a "criss-cross" pattern effect.

The hatch will be applied immediately after the OK button is pressed.

### Hatch Color

Hatch color is, as with other drawn entities, determined by current layer color setting, or else by a set



color that over-rides bylayer color settings. Color fills are achieved by making the selection of "Solid" in the pattern name window.

Further information on hatch pattern definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

---

**See also:**

Boundary Hatching  
Modify Hatching  
Polyline Boundary

Boundary Hatching



**Command: BHATCH**  
**Alias BH**

**Located:**

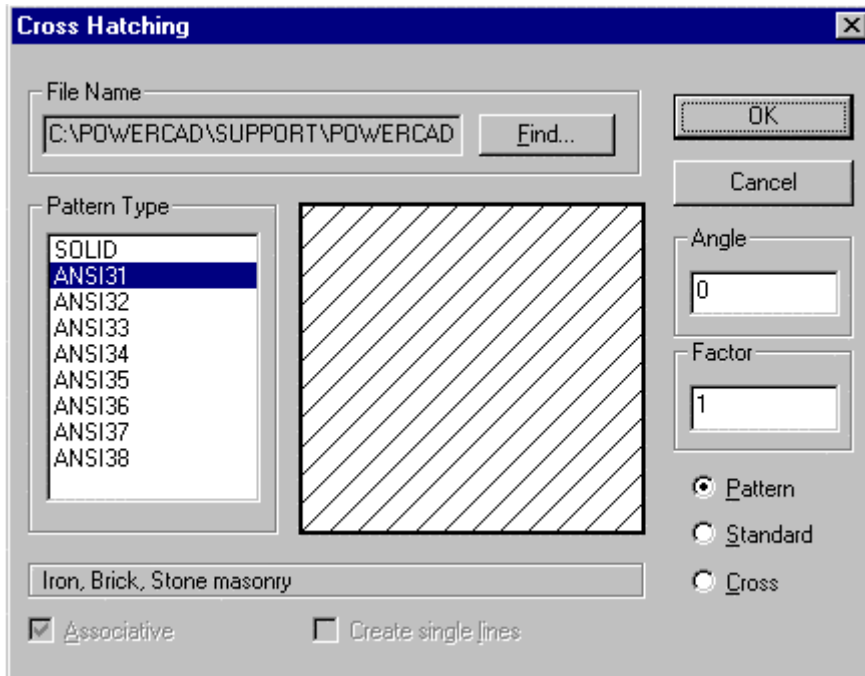
- 1. Pull-down Menu: Detail – Boundary Hatch**
- 2. Palette: Detail**
- 3. Or Enter Command or Alias Directly at the Command Line**

The command BHATCH fills a region with a hatch pattern or solid, bounded by linear entities, just by pointing to any point within the boundary.

The BHATCH command defines a boundary by creating a polyline that follows the entities that make up the boundary. Select each entity to define a boundary around the area you want to hatch.

```
> BHATCH
Select objects:
Select point inside boundary contour <+X>:
Select additional elements? <N>:
<Dialog box appears>
```

Note: to exclude internal areas from hatch application, select the internal boundaries to be ignored when prompted for "Select additional elements."



Hatch patterns are stored together in **.pat** files. The file name input at the top of this dialog allows you to select a pattern from any of a number of .pat files that you may have available. Having selected the file, all patterns included in that file are listed in the Pattern Type window on the left. Clicking on any of these causes a representation of that pattern to appear in the central window, making selection easier.

The selections to the right on this dialog modify the pattern, in which:

- Angle – Will cause a counter-clockwise rotation of the pattern by the angle entered;
- Factor – Will cause the pattern to become more condensed or expanded, depending upon the entered factor value;
- Pattern will display a non-uniform pattern (if that is available in the particular selection);
- Standard – Will modify the pattern to a uniform spacing and
- Cross – Will create a "criss-cross" pattern effect.

The hatch will be applied immediately after the OK button is pressed.

### Hatch Color

Hatch color is, as with other drawn entities, determined by current layer color setting, or else by a set color that over-rides bylayer color settings. Color fills are achieved by making the selection of "Solid" in the pattern name window.

Further information on hatch pattern definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

---

### See also:

Hatching  
 Modify Hatching  
 Polyline Boundary  
 Polyline Contour

## Create Boundary Polyline for Hatching



**Command: BPOLY**

**Alias BP**

### **Located:**

1. **Pull-down Menu: Draw – Polyline Contour**
2. **Left SideBar**
3. **Palettes: Draw and Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

The command BPOLY creates a boundary polyline from existing objects that form an enclosed area, most often for use with the HATCH or AREA commands.

The original entities can be made up from a combination of lines, arcs, circles, and polylines.

They can be in any arrangement as long as the boundary will make up an enclosed area without looping back upon itself.

Note: The BHATCH command can be used to create its own boundary polyline, while applying a hatch pattern to an enclosed area.

---

### **See also:**

Command PCONTOUR

PCONTOUR is the command line version of the BPOLY command.

### **Polyline Contour**



**Command: PCONTOUR**

**Alias PC**

### **Located:**

1. **Left Side Bar**
2. **Entered Directly at the Command Line**

This command allows creating a closed polyline on top of (overlying) objects that make up a closed boundary. Such a polyline contour can serve for hatching regions of the drawing with the HATCH command or to determine the area or circumference of a region with the AREA command (applying the option *Object*). The BOUNDARY command is identical in operation to this command and is provided for users who may be more familiar with the term.

From the menu *Draw*, choose **Polyline Contour**. At the command prompt, enter PCONTOUR.

It is recommended that you set a specific layer before invoking the command (like a hatch or contour layer), because the polyline contour will overlap entirely or partially existing drawing entities, making selection otherwise difficult.

After giving the command, you are prompted to select objects to be examined to find the contour.

**Important Note:** At this point, it is not required to select only those entities that should make up the contour. You might enter the option *All* as object selection mode, however, the supplying a smaller selection set can improve the command efficiency, as fewer objects are evaluated in boundary establishment.

After the selection set to be examined has been created, the following **Ray Casting** methods for contour

finding are available to find the next boundary object: Normally, the default works well, however there may be instances where you may wish to select a specific option:

1. **Nearest** sends radial rays from the pick point you specify (see below) to the nearest object.
2. **+X** sends a linear ray in the positive X direction from the pick point you specify to find an entity.
3. **-X** sends a linear ray in the negative X direction from the pick point you specify to find an entity.
4. **+Y** sends a linear ray in the positive Y direction from the pick point you specify to find an entity.
5. **-Y** sends a linear ray in the negative Y direction from the pick point you specify to find an entity.

Next you are prompted to pick the start point from which the program begins assessing the intended boundary..

Once a boundary object has been found in the direction of the specified pick point, the command traces the boundary in a counterclockwise direction.

After analyzing the specified selection set successfully, a closed polyline will be produced on the current layer, covering the boundaries of overlying drawing entities.

### **Modify Hatching...**



**Command: HPEDIT**

**Alias HED**

#### **Located:**

1. **Pull-down Menu: Detail – Modify Hatching**
2. **Left SideBar**
3. **Palette: Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

You can exchange an exiting associative hatch using this command.

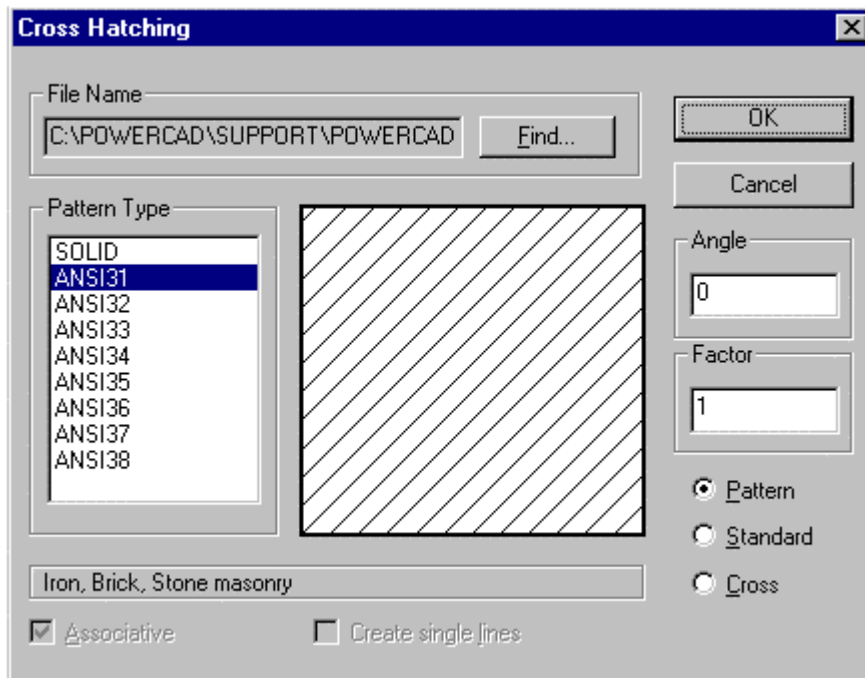
It also allows you to add additional objects to the set of entities to be hatched, as well as change the hatch pattern.

> HPEDIT

Select an associative hatch:

Select additional elements? <N>:

The routine then displays the Hatch Pattern dialog box to select the type of hatching desired.




---

**See also:**

Command QHPEDIT

QHPEDIT is the command line version of the HPEDIT command provided for utilization in Lisp routines, macros, etc.

## Model Mode / Layout Mode

### Model Mode (Model Space)

Model Space is the mode in which the program opens. Generally, this is where you will create your drawing "model", or simply, your drawing. This model usually contains most drawn entities and details.

Where Layout mode (Paper Space) is used, detail features that need to be re-oriented for use in different Layout mode viewports (notes, title blocks, etc.) are usually drawn in Layer, rather than Model Mode.

### Layout Mode (Paper Space)

One main advantage of working with CAD is that any plan and construction can be created in real world units. Nearly any construction, however, needs to be printed on paper. Usually this means plotting it to a certain scale.

For that reason, at a certain state during construction, you will want:

- To add title blocks to your drawing,
- To display different views and regions,
- To add model details,
- To lay out part lists, legends, and annotations and instructions on the drawing sheet.

To help solve these tasks in a flexible manner, the program provides two modes to handle a drawing: Model Mode and Layout Mode. In other words, there are two separate spaces to work on a drawing:

Model Space and Paper Space (or Print Space).

The drawing objects in model space constitute the geometry of the model. The objects of paper space reproduce the model in different views and paper space assists when annotating the drawing.

**Notes:**

1. Layout mode is a complex environment and not always easy to work in. Users are therefore advised to become familiar with layout mode practices and concepts through studying the documentation, before using it on mission-critical project.
2. If you create smaller, purely 2-D drawings, it is often easier to inset (on separate layers), separately scaled title blocks, part lists, and details in model space, rather than doing this work in layout space.
3. When using layout mode you should always keep in mind that layout mode is primarily intended for the arrangement of complex drawings in preparation for plotting. The true benefits of this occur when you have a drawing model that must be spanned by multiple plots, or when multiple views of your drawing model must be presented on a single drawing sheet.

**See also:**

Command TLMODE

Command VIEWPORT

## Toggle between Model Mode and Layout Mode



**Command: TLMODE**

**Alias TM**

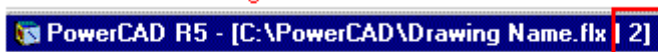
**Located:**

1. **Pull-down Menu: File – Layout / Model Mode**
2. **Left SideBar**
3. **Or Enter Command or Alias at the Command Line**

To switch from model mode to layout mode and vice versa, type in the command TLMODE (Toggle Layout Mode).

The tiled drawing windows of the current drawing in Model Mode, that display the model in different views, are replaced by a single window. The title bar of this window shows the caption **Layout:** and the drawing name. The model mode references to drawing window disappear.

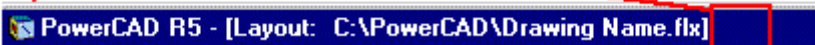
Model Mode - Drawing Window 3



Window Number (0,1,2 or 3)

Layout Mode

No Window Identifier



**Notes:**

When you switch from model space to paper space the first time, the Layout window is an empty window that represents a blank drawing sheet. This print space is not blank, however, if your template drawing already contains paper space objects (in this case paper space usually contains a company title block). In layout mode, all viewing of your drawing (which is in model mode) is through viewports that "cut

through" the layout mode drawing surface to permit viewing the drawing in model mode that lies "behind it".

The first steps when entering layout mode are usually to perform two types of operations:

1. Determination of the sheet size for printing/plotting particularly by insertion of a title block;
2. Creation of viewports that display individual view onto the model.

You can draw in paper space as used to in model space. All the draw and edit commands are available. Usually, you will want to restrict your layout mode drawing exercise to the creation of sheet-specific notes, dimensions and title block information.

As paper space is designated to lay out drawing output, a number of 3D commands are disabled in layout mode: 3DVIEW, PLANVIEW, RENDER, HIDE, Fhide, LIGHT, LIGHTEDIT, SETVIEWDIR, WOPEN, and QWOPEN. As described in the following sections, layout mode allows you to arrange individual 3D views of the model by creating and modifying viewports.

---

**See also:**

Layout Mode

VIEWPORT

### **Using Viewports, Windows and Views**

These three AllenCad concepts or features may be a bit difficult for the new user to grasp at first, so the following distinctive characteristics of each should be noted:

- **VIEWPORTS** are strictly Layout mode (paper space) drawing "pictures" of the Model mode drawing. These "pictures" can be 3-dimensional and can be grouped (as many as you like) within a surround and title drawn in Layout mode and plotted in layout mode as a single drawing.
- **Windows** (Commands WOPEN, Wclose, WTILE, WCASCADE), on the other hand, are strictly a model mode (or space) feature. The initial frame in which the drawing edit area resides when the program opens is the initial window. Other windows, offering unique working outlooks, can be opened in the same drawing. Drawing windows allow you to jump back and forth and work on separate parts of the drawing without needing to pan/zoom back and forth across your drawing. The "outlook" in each window can be 3-dimensional, as required. You can only print/plot from within a single window and you can have a maximum of four windows for any drawing at any given time.
- **Views** – (Command VIEW) These are simply - named 3 dimensional views of the drawing that are defined by vantage point and direction of view. View settings are stored in memory in this named file and can be called into drawing windows or viewports to specify the specific vantage point and drawing view as required. A single View is visible within a viewport or window at any given time.

## **Layout Viewport Control**



**Command: VIEWPORT**

**Alias VP**

### **Located:**

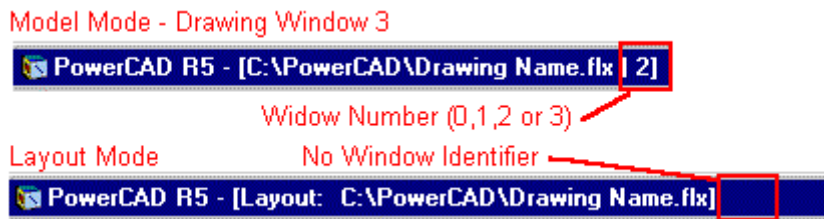
1. **Pull-down Menu: File – Layout Viewport Control**

2. **Left SideBar**
3. **Or Enter Command or Alias at the Command Line**

The command VIEWPORT command is used in layout mode to create rectangular windows that display the drawing model that resides in model space. In these viewports, you can control both the view direction and the display of a specific portion of the model. The VIEWPORT command options are: New, ON, OFF, Layer, and View. These functions are described in detail, below.

### Creating Using Drawing Windows

After switching to layout mode for the first time, usually you will create one or more viewports to show one or more views to your model.



Before creating a viewport, you must ensure that you are first in layout mode. At the top of the desktop window, you should check to see that "Layout" appears before your current drawing name, as in the above illustration. The command TLMODE toggles the display between layout and model mode and may need to be executed if you are still in model mode.

When you issue the VIEWPORT command, you are presented with the following dialog:



The option *New* of the command VIEWPORT predictably allows you to create a new viewport. New is also the default option of the VIEWPORT command. Next, you are asked to specify two opposing corners for the new viewport. If you have determined the size of the viewport that you need, you can either enter defining corner coordinates, or pick the corners of a rectangle drawn previously in layout mode, for this purpose. After specifying two corners of the viewport, the entire model is displayed in plan view and to its current extents, in the newly created viewport.

```
> VIEWPORT
First corner: 10,10
Opposite corner: @45,25
```

Then the next step will be to determine the size of the content of viewport to match to a certain scale and to maintain the accuracy of this scale between model space and paper space within these viewports.

Detailed hints and tips to apply appropriate techniques to set and keep a scaled relation between paper space viewports and model space are described later in this section after all the options of the command VIEWPORT have been discussed.

### Viewport Alteration using Modify Commands

A Viewport is a drawing entity represented by an rectangle and contains a certain view of the drawing. Theoretically, any number of viewports may be created in layout mode.



Several editing commands can be performed on viewport entities, because these are treated as drawing objects:

- You can move a viewport to another position.
- You can change the size of a viewport with the Stretch or the scale command. In these cases, the view inside the viewport is not scaled with the viewport, only the frame, itself. If you scale a viewport, however, the displayed view inside of the viewport is scaled with the same factor as the viewport frame.
- You can remove viewports from the drawing, using the Delete command and picking the frame.
- Using `_Drawing_Windows` may be copied.
- Note also that, the commands Rotate, Array, Flip, and Mirror modify the position of a viewport (respectively a copy of a viewport), but the edges of the rectangle remain parallel to the X- and Y-axis of layout mode. Also, these commands maintain the displayed view in the rotated or copied viewport.
- When editing viewports and when drawing in paper space you can snap to the end point, mid point, etc. of the viewports rectangle, and you can utilize the coordinate filter options.

### Command Options:

#### On/Off - Turning viewports off and on

The Viewport command options **On** and **Off**, control the visibility of the contents displayed in existing viewport windows. To turn the visibility of the viewport objects off, choose the option OFF and select the viewport frame or pick to an object inside the frame. To redisplay the representation of the model within a deactivated viewport, choose the option ON and pick an edge of the corresponding viewport frame.

The **visibility of the viewport frames** can also be controlled through layers. Like to any other drawing entity a viewport entity is located on a layer. It is recommended that you place viewport entities on one or two specific layers (for example LM\_VIEWPORT\_ON and LM\_VIEWPORT\_OFF). Now you can control the visibility of the frames of viewports by turning off or freezing a corresponding layer containing viewports.

#### Layer - Control of viewport contents

The VIEWPORT command option **Layer** permits manipulating the display in specific viewports by freezing and thawing drawing layers. This means that it is possible to have layers frozen and invisible in one viewport, yet unfrozen and viewable in another.

After choosing the option *Layer* the dialog box to control layer visibility in layout mode is displayed:

The list box on the left-hand side the dialog box contains all layers defined in the current drawing. The other list box is meant to determine those layers not to be displayed in the currently selected viewport ("Frozen Layers"). You can alter the visibility mode by marking layer names in one of the two lists and add or remove them from the list of frozen layers with clicks to the > button or < button. To freeze or thaw the status of entire layer groups, you can utilize the edit boxes by specifying strings with wild cards (for example "2D\_\*") and then click to the Filter button that will highlight all matching layers in the appropriate list.

#### View - Control in viewports

The VIEWPORT command option **View** toggles the layout window into a mode, where the selected viewport is displayed such that it covers the entire window. This mode serves to change the view to the model inside of the specified viewport.

```
> VIEWPORT
First corner: View
Select viewport: Specify viewport
Switch to viewport view ...
Viewport control:
```

When calling the option *View* the prompt "Viewport control:" is displayed. The following view options are at your disposal: Zoom, Pan, 3DView, View.

### **Zoom**

By intention the default prompt at the command line after choosing the Zoom function says "Zoom Scale factor: ". This will remind you that - once a scaled relation between model space and paper space has been established - you should manipulate the view within the viewport only via scaled zooming (usually by specifying whole numbers) to maintain the scale. In addition, however, the option bar contains the other zoom options (Window, Extents, Previous) as usual.

### **Pan**

The option *Pan* solely moves the visible portion within a viewport to a direction specified by two points without changing the scale of the model representation in paper space.

In *Viewport control* mode any transparent command may be used. By this, you can also utilize the pan functions from the control panel (Pan Left, Pan Right, Pan Up, Pan Down); for example:

Viewport control: 'PANLEFT'

### **3DView**

Naturally, the layout mode can be utilized also to illustrate the model from different viewpoints in space. The option *3DView* of viewport control allows you to set individual 3D views or predefined views like plan view, front view, left side view.

The option *3DView* will open a dialog box similar to the one invoked by the command 3DVIEW in model space.

**Note:** Using the option *3DView* will zoom the contents of the viewport to the drawing extents. To achieve a scaled relation between paper space and model space for isometric or other individual 3D views of the model, it is recommended that you use named views prepared in model space, as discussed next.

### **View**

The option *View* allows you to display views in specified viewports, that have been saved as named views in model space. Layout mode efficiencies can be improved by saving named views in model mode designed for reuse in the viewports of the paper space. This is especially true if maintaining a scaled relationship between model and print space is required. See the section *Techniques and Tips for Layout Mode* below, for further hints and recommendations.

The option *View* will open a dialog box similar to the one called by the command VIEW in model space.

A particularly useful help feature is provided with the Show button in the Save/Retrieve named views dialog box, because this option allows you to preview a selected named view in the drawing before confirmation.

**Note:** When calling the command VIEW in layout mode outside the command VIEWPORT, you can create named views within paper space (for example to zoom fast to portions of the drawing sheet).

### **Utilizing transparent commands**

You can utilize in *View control* mode all transparent commands. As already mentioned, you can call the functions 'PANLEFT', 'PANRIGHT', 'PANUP', 'PANDOWN' from the control panel. Another example is the transparent command 'SETVIEWDIR' to set an individual or standard isometric 3D view within a viewport. Or, the transparent command 'DIST' that allows you to measure the distance between two points.

### **Techniques and Tips for Layout Mode**

Create your construction (in model space) in real world units (at scale 1:1). Utilize specific layers for dimensions, hatching, text, etc. When managing and using layers, keep in mind that you might want to illustrate different "logical views" of your model later, when presenting or plotting your construction with

the aid of layout mode.

At a certain stage of drawing construction, you may need to print out the drawing for presentation within your company or to forward it to a manufacturer. In this situation layout mode can help in many cases with the quality of your presentation:

- a. Switch from model space to paper space using the command TLMODE. The title bar now says Layout: followed by the drawing name.
- b. Set the limits to equal the paper size in those dimensions (units) that will be used when printing (for example A-size, B-size, C-size, ISO-A0, ISO-A1, ISO-A2, ... Letter Portrait, Letter Landscape).
- c. Create a layer for the title block (unless not existent), set it current, and insert your title block in paper space at coordinate origin (0,0). Or, draw a rectangle on a temporary layer with the paper size you will use when printing.
- d. Create a layer to place the viewports on. Make this layer current before you create new viewports with the VIEWPORT command and make sure to use this specific layer only for viewports. It is recommended that you activate the snap mode with applicable units (inch, millimeter, centimeter, ...) when creating viewports.

The next step, once a viewport has been established, is to set up scaling between the model and its representation on the drawing sheet. This step sometimes causes headaches. Setting up the appropriate scale for displaying the model, can be prepared in advance, in model space in several ways:

- Positioning rectangles on a temporary layer over sections of your drawing in model space that match viewport windows in paper space at a particular scale;
  - Creating named views in model space with the command VIEW that then can be retrieved within the viewports in paper space with the option View.
- e. Once the scaled relation is set up, you should avoid uncontrolled zooms in the viewport, and confine yourself to Pan commands and to the ZOOMFAC command (usually specifying whole numbers).

To modify the height or length of a viewport, we recommend you use the command STRETCH or SCALE. The display of the model in the specified viewport will not be scaled with this command. It maintains the scaled relation between model space and paper space.

Keep in mind the "golden rule":

*Drawing entities you create in model space can only be edited in model space. Drawing objects you create or insert in paper space can only be modified in paper space.*

---

**See also:**

Layout Mode

TLMODE command

## Layout Viewport Control (Command Line Version)

**Command: QVIEWPORT**

**Alias QVP**

**Located:**

**1. Entered Directly at the Command Line**

The command QVIEWPORT allows you to create rectangular areas displaying the model while in layout mode and to control the view direction or display portion of the model in those viewports.

The command QVIEWPORT works like the command VIEWPORT but with one difference: QVIEWPORT is performed completely at the command line or used in macros or LISP files..

---

See also:

VIEWPORT command

## Copy a Drawing Viewport

**Command:** QWOPEN

**Alias:** QW

**Located:**

1. **Pull-down Menu:** Window – Copy Current Viewport 1:1
2. **Or Enter Command or Alias at the Command Line**

The option **Copy Viewport** on the *Window* menu allows you

1. to make a quick copy of the current viewport or
2. a plan view, by choosing the desired option button.

## Drawing Windows



### Windows

AllenCad is capable of opening as many as four drawings at the same time. Each of these drawings can be viewed in its own window. Further, it is possible to open as many as four viewing windows for each drawing so that the drawing may be seen from different angles or so that portions of the drawing can be examined. A maximum of sixteen drawing windows can then (theoretically) be active at any one time.

### Opening Drawing Windows

In order to open a drawing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing. Open the menu *Window* and choose the option *Open Another Viewing Window* (command WOPEN). Before the program opens another window, you will be asked to determine the type of view in this new window. Choose a standard view from the menu, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

### Changing Drawing Windows

There are two ways to change the "active" window:

- ▶ Clicking on any part of the chosen window with the cursor will activate it.
- ▶ Open the menu *Window*, and find where all open drawing windows are listed. The active window will be marked by a check in front of its name. When a new window is opened, a number in front of the specific entry marks it.

### Closing Drawing Windows

It is possible to close the viewing window, as in all Windows programs, by double clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

### Viewpoint / Angle of View

The direction from that an object is viewed influences the view of the object, and may be changed in many ways inside one drawing window. Each additional view will be portrayed in another viewing window. It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the pull-down menu VIEW provides the facility to make that definition.

### Saving and Retrieving Views

It is convenient when working with different drawing portions to save, and later, to retrieve changes in angles of view, different drawing portions and scales of view. This may be accomplished by using the command VIEW.

See also

Zooming and Panning

Parameter for View Commands

## Open New Drawing Window

**Command: WOPEN**

**Alias WO**

**Located:**

1. **Pull-down Menu: Window – Open New Viewport**
2. **Or Enter Command or Alias at the Command Line**

In order to open a viewing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing.

From the menu *Window* choose the option **Open Another Viewing Window....**

Before the program opens another window you will be asked to determine the type of view in this new window. Choose an individual viewpoint or a standard view from the dialog box, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

## Close Drawing Window

**Command: WCLOSE**

**Alias WC**

**Located:**

1. **Pull-down Menu: Window – Close Current Viewport**
2. **Or Enter Command or Alias at the Command Line**

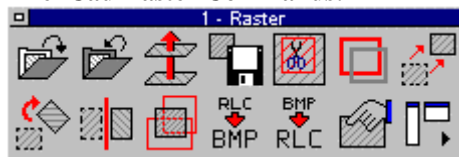
It is possible to close the current viewport, as in all Windows programs, by double-clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

# Raster Graphics

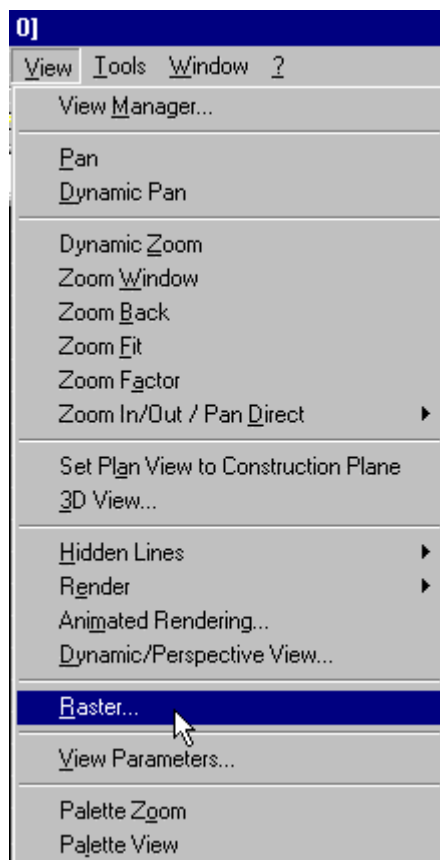
## Raster Graphics Overview

**Functional in: AllenCad Pro 4**

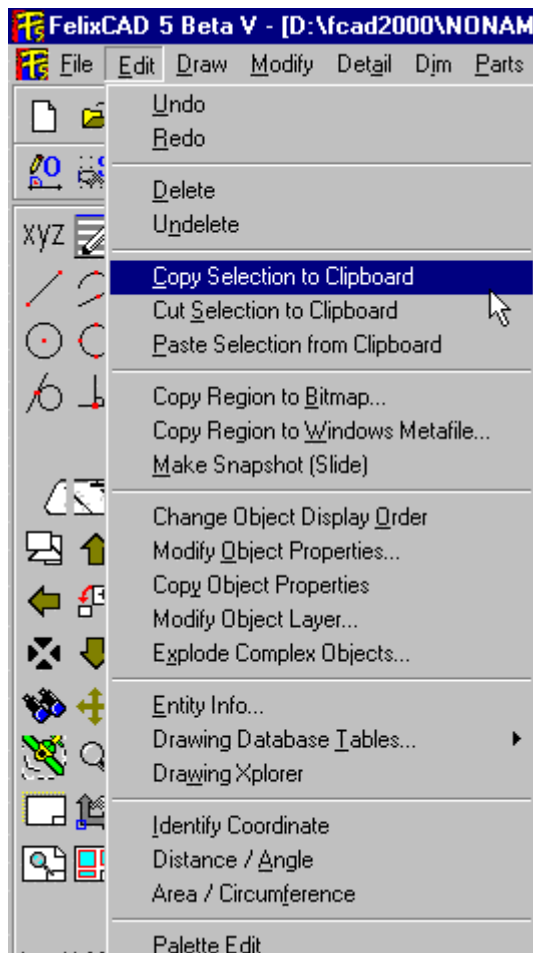
**AllenCad Raster Commands:**



Most of AllenCad's raster graphic-related commands are available on the Raster palette. This can be called onto the desktop from the View – Raster pull-down menu as indicated below. Not all graphics commands can be successfully entered at the command line, so users are advised to use the palette commands when working with Raster graphics.



Apart from these commands, a number of raster graphic export commands are located on the Edit pull-down menu (below). These include copying/cutting to the Windows clipboard, importing an image from the clipboard, and creating new graphic files in BMP or WMF format from selected drawing entities.



There are many different raster graphic formats, however AllenCad has been designed to work with some of the more common ones. The program can import .RLC, .BMP and .TIF format files, as well as any graphic that has been copied to the AllenCad Clipboard. The program can also directly export selected entities in .BMP and .WMF file formats as well as exporting to the Windows Clipboard, where the image can be copied into other graphics utilities and translated into the format needed. Where editing in AllenCad is required, the user should try to use the .RLC file formats as input, as the program will only edit raster graphics in this format. A bitmap (.BMP) to .RLC translator is provided on the raster palette to prepare files for import in this format.

Imported graphics are referred to by number (IE #0 for the first imported image and #1 for the second.) Prompts will be issued for you to identify the image to be addressed in this fashion, once you begin editing or working with these graphics in the AllenCad drawing editor.

### **Raster Graphic Overview:**

There are presently two general types of graphics files; vector and raster. Graphics files, such as produced by AllenCad are generally vector graphics, where a line (for example) is defined as a continuous straight line between two points. This same line can also be represented by a dense cluster of points along the line alignment that gives the same visual image as the defined line. The second format is referred to as a Raster graphic. Scanned photographs and screen captures are both examples of this format.

Because they are defined by a vast number of points, Raster graphic files are considerably larger in file size than vector graphics and are generally used to present a general picture or image, rather than to represent geometrically defined lines, arcs, etc.

Their use in CAD is primarily as a translation medium, where the operator uses a scanned image to trace a vector definition on top of the raster image, or also as graphical background imaging or "image painting".

## Load Raster Image File

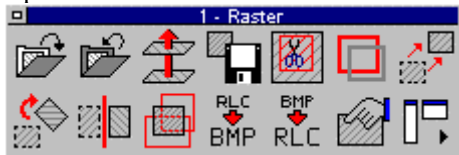


### Command RLOAD

#### Located:

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This item allows you to insert a scanned, raster image that has been stored in the RLC, TIF or BMP format, into your current drawing, thus allowing you to see the raster image and the vector information at the same time. In addition, you can import any graphic that has been placed on the AllenCad clipboard by a previous COPYSELECT or CUTSELECT command.



As with most Raster commands, you can find this command on the Raster palette that is placed on the drawing editor when you click on the Raster entry on the View pull-down menu. This command cannot be entered directly from the keyboard.

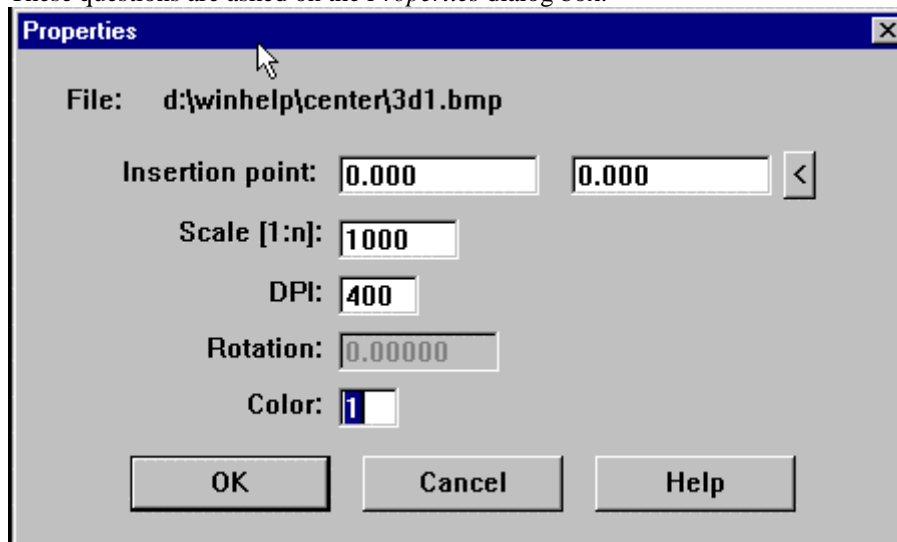
There are many different formats that a scanned image can be stored in such as BMP, GIF, CALS, TIFF, etc., and RLC.

At this time the program will only edit RLC format and import RLC, BMP and TIF files. AllenCad has a utility program that converts files from BMP to RLC formats, however, there are many other conversion programs on the market that may assist you in changing from another format to RLC.

When you start the routine, you will be shown the following dialogue box to allow you to select the raster image you wish to insert into the drawing. You must specify the format of the file that you are trying to locate.

Once you have selected the file to load, you will be asked where to insert the image, the scale to insert it at, the color of the image, the DPI (resolution in dots per inch), and the rotation factor.

These questions are asked on the *Properties* dialog box.



Once you have entered the correct information, your raster image will then appear on screen.



## Assemble / Fit Raster Files



**Command: RMONT**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command lets you designate a portion of an imported RLC file for saving in a new file after defining the area by window selection. In effect, you will be saving a "cropped" image in a different file.

You will be prompted for the First Corner and then the Second Corner.

After picking the area to copy, you will be asked to confirm the properties of this new image.

The new file will only be created after you save the property changes and you select the Save Raster button on the palette. You will also be prompted to save the file when you remove the raster, if the file has not been saved to this point.

### Paste Selection from Clipboard



**Command: PASTE**

**Alias PST**

**Located:**

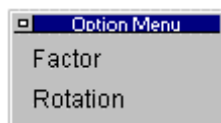
1. **Pull-down Menu: Edit – Paste Selection From Clipboard**
2. **Top Function Bar**
3. **Palette: Edit and Modify**
4. **Or Enter Command or Alias at the Command Line**

The PASTE command copies objects from the **AllenCad-specific** clipboard to point you select in the current drawing. These objects have typically placed on the clipboard by a preceding COPYSELECT or CUTSELECT command.

Use the command to copy graphic data from other drawings into your current drawing.

**To paste the objects from the clipboard into the current drawing:**

1. Choose the PASTE command.



2. You are presented with the option dialog. If you plan to scale or rotate your objects during insertion, select your option now. You will be offered the opportunity to scale and rotate as a conclusion to the command later if you do not choose at this point, however you will not be able to visualize the scaled or rotated objects in advance, if you wait until later.
3. Specify the insertion point by entering coordinates of the insertion point, OR
4. Drag the highlighted objects into position on the drawing. The objects of the clipboard contents appear highlighted (as well as scaled and/or rotated - if you selected this in the last step), and attached to the cursor during the exercise. This feature allows you to visualize object placement prior to selecting the insertion point with a mouse click.
5. Specify the scale factor for insertion – you can specify a scale factor at this point (if you have not done so already) and have the objects inserted at a larger or smaller scale than they were

originally. Note that the relative position of these objects to each other remains the same (is scaled too) during the operation).

6. Specify the rotation angle (if you have not done so already) – you can choose to rotate the object by any input angle during the exercise.

**Example:**

```
> PASTE
Insertion point:
Scale factor <1>:
Rotation angle <0>:
```

When inserting the objects from the clipboard you can also specify the scale factor and/or rotation angle before you indicate the insertion point by choosing the appropriate option from the context bar when prompted for the insertion point:

```
> PASTE
Insertion point: Factor
Scale factor <1>: .5
Insertion point: Rotation
Rotation angle <0>: 90
Insertion point:
```

**Note:**

The objects selected with COPYSELECT remain on the clipboard, ready to be placed in another drawing, until you place new objects on it with the COPYSELECT command.

To copy objects within the current drawing it is recommended that you use the COPY or ARRAY command found in the *Modify* menu.

---

**See also**

Command COPYSELECT

Command CUTSELECT

## Remove Raster Image



### Command RUNLOAD

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This option will remove the selected image from the drawing and optionally allow you to save any changes to the properties of that image.

The following prompt is displayed;

Save properties for 'C:\RLC\_Images\MyImage.rlc ?':

If you answer NO then the image is simply removed from the screen and the properties remain unchanged. If you answer YES, the following is displayed:

Properties saved for 'C:\RLC\_Images\MyImage.rlc'.

The image is then removed from the drawing.

## Edit Raster Image Properties

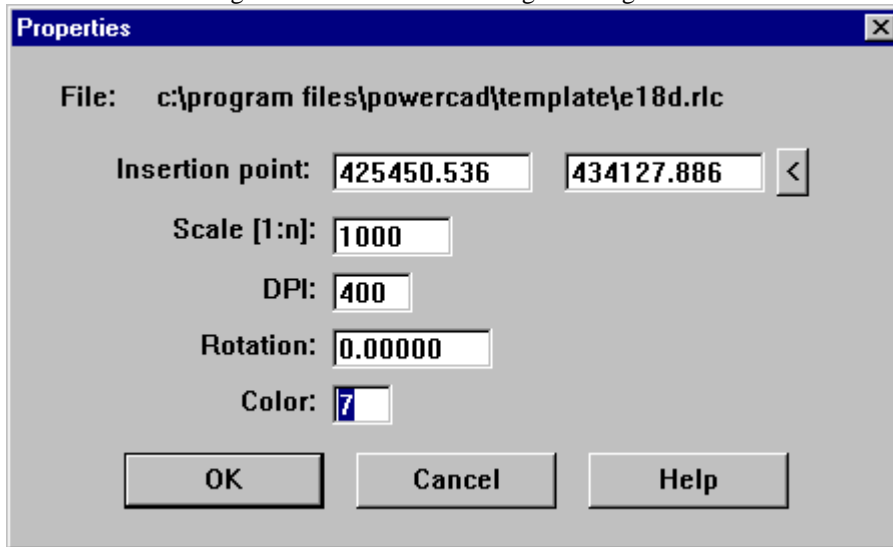


### Command: RPROP

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command brings a dialog box up that allows you to change the properties of an imported raster image. It raises a dialog in which any of the fields can be modified as needed. The changes are only done in the current drawing and will not affect the original image on the hard drive.



Insertion point:

Location to insert the image on your drawing.

Scale [1: ]:

Sets the relative scale factor to match your drawing.

Scales the raster image larger or smaller.

DPI:

Dots Per Inch. This sets the resolution of the Image.

Rotation:

Affects the orientation of the image in reference to the drawing.

Positive rotation is counterclockwise, negative rotation is clockwise.

Color:

Sets the color the image will take on when inserted into a drawing.

The whole image is the same color.

Each raster can be brought in with it's own color.

## Save Raster Image and Properties



**Command: RSAVE**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This function allows you to save all of the changes to the image, done by the edit image properties item, back to your hard drive. As edits can be made on RLC format graphics only, this command is ineffectual for other formats, unless they have been converted by the program from RLC format after editing.

The following message is displayed.

Properties saved for 'C:\RLC\_Images\MyImage.rlc'.

**Note:**

These changes are permanent so be careful and make sure you have a backup of the original raster image before saving the changes. The save function

## Raster Frame On/Off



**Command: RBORDER**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

The rectangular frame border around the images can be turned off or on by using this command. All frames are affected at the same time. When on, the frame shows the outer edge of the image and is in the same color as the image.

## Cut Raster with Polyline Boundary



**Command: RCUT**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command applies to imported or internally converted RLC format graphics only. If you have a portion of the raster that you do not want and you wish to remove it from the job you can use this command to do so.

You must first draw a closed polyline around the area to be clipped. The polyline **MUST** be a series of straight line segments, arcs will be treated as if the chord of the arc was the boundary, and you **MUST** close the polyline.

You will be prompted to pick the raster image to clip, just as in the move image and rotate image commands.

Select raster /<point>/?/Last/#n [#3]:

Once it knows the correct raster image you will be asked to pick the polyline to use for the clipping. Then the program will ask you

Save properties for 'C:\RLC\_Images\MyImage.rlc ?':

If you answer NO then the image is simply clipped and that portion falling within the polyline boundary is removed from the screen and the properties remain unchanged. If you answer YES the following is displayed;

Properties saved for 'C:\RLC\_Images\MyImage.rlc'.

and then the image is cut or clipped inside of the polyline and the properties are saved for the original raster image.

## Move Raster Image



**Command: RMOVE**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command moves an raster image on your screen. If you have more than one raster image in the drawing, you will be asked to select the one to be moved in the following prompt;

Select raster /<point>/?/Last/#n [#0]:

You can point at the raster image, choose Last to pick the last one selected, or enter in the number of the image and the ? mark displays all the numbers to choose from.

You will then be asked for a Base point and a Destination point, just like the standard move command.

## Rotate Raster Image



**Command: RROTATE**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command applies to imported or internally converted RLC format graphics only. You can rotate the selected raster image to match your vector drawing by using this command. The prompt is as follows:  
Select raster /<point>/?/Last/#n [#0]:

This allows you to select the correct raster image to rotate, similar to when you move a raster image.

Rotation angle:

This is the amount of rotation, clockwise is a negative rotation and counter-clockwise is a positive rotation.

## Mirror Raster Image



**Command: RMIRROR**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command allows you to mirror image the image of your choice.

You must first select the correct image on screen:

Select raster /<point>/?/Last/#n [#0]:

After selecting the image, you will be asked;

Mirror axis:

There will be two options showing on the button bar - Horizontal and Vertical. Pick the one you wish to use.

You now have the option to edit the image properties.

The original image is not removed and the mirrored image will be displayed where and how the image properties are set.

## Copy Region to Bitmap



**Command: BMPOUT**

**Alias BO**

**Located:**

1. **Pull-down Menu: File – Export Region To Bitmap**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

Call the command either by keyboarding BMPOUT or by selecting **Copy Region to Bitmap...** from the menu *Edit*.

The following options allow all **entities** in the region to be copied into a bitmap:

**Window**

Define a window by setting two points.

**View**

The complete current view will be exported into a bitmap.

**Extents**

The region to be copied is defined by the extent of the drawing.

Next, specify the **Width** and **Height** of the bitmap in Pixel Units (10...2000 pixels).

Finally, you may choose whether the bitmap should be written into a **File** or copied to the Windows **Clipboard**.

**Notes:**

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The bitmap remains in the clipboard until you clear it. Place new objects on it with the RENDER, BMPOUT, WMFOUT or other command, or Cut or Copy in other Windows applications.

## Copy Region to Windows Metafile



**Command: WMFOUT**

**Alias WM**

**Located:**

1. **Pull-down Menu: File – Export Region to Metafile**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

The following options allow the region to be copied into a WMF vector format file:

**Window**

Define a window by setting two points.

**View**

The complete current view will be exported into a bitmap.

**Extents**

The region to be copied is defined by the extent of the drawing.

Finally, you may choose whether the picture should be written into a **File** or copied to the Windows **Clipboard**.

**Notes:**

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The WMF vectors remain in the clipboard until you clear it.

## Transformation Vectors



**Command: RVECTORS**

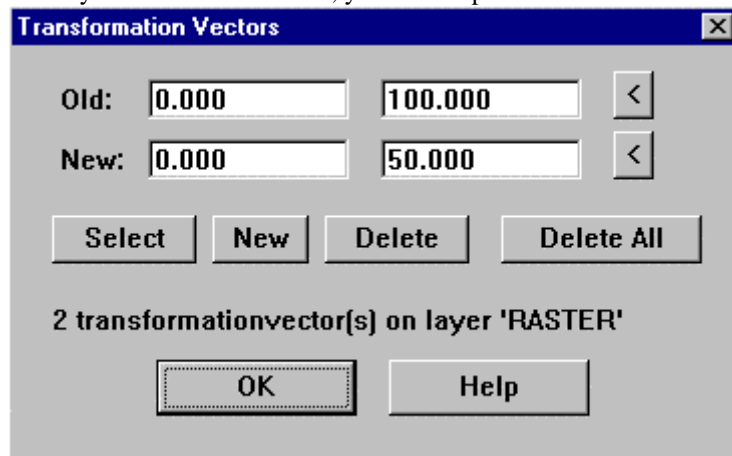
**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command allows you to stretch your imported RLC graphic to the extent and with a rotation as indicated by the relative length and alignment of two lines (vectors)

Before issuing the command, you will need to create a new layer called "Raster".

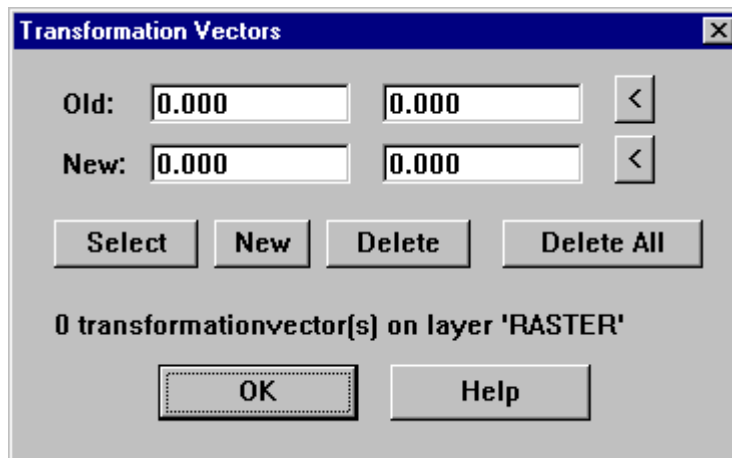
When you issue the command, you will be presented with the following dialog:



The dialog box titled "Transformation Vectors" has a close button (X) in the top right corner. It contains two rows of input fields. The first row is labeled "Old:" and has two text boxes with values "0.000" and "100.000", followed by a button with a left-pointing arrow. The second row is labeled "New:" and has two text boxes with values "0.000" and "50.000", followed by a button with a left-pointing arrow. Below these are four buttons: "Select", "New", "Delete", and "Delete All". At the bottom, it says "2 transformationvector(s) on layer 'RASTER'" and has "OK" and "Help" buttons.

This dialogue box allows you to enter and edit the transformation vectors required to adjust the raster image to match the vector drawing.

The vectors you define must be drawn on the layer RASTER, and you can actually draw the vectors before starting this command and it will detect them for you. Simply draw one line (vector) on the raster image and then a second line (vector) to match the size, location and orientation of the first line after the raster is transformed to match your drawing elements.



The dialog box titled "Transformation Vectors" has a close button (X) in the top right corner. It contains two rows of input fields. The first row is labeled "Old:" and has two text boxes with values "0.000" and "0.000", followed by a button with a left-pointing arrow. The second row is labeled "New:" and has two text boxes with values "0.000" and "0.000", followed by a button with a left-pointing arrow. Below these are four buttons: "Select", "New", "Delete", and "Delete All". At the bottom, it says "0 transformationvector(s) on layer 'RASTER'" and has "OK" and "Help" buttons.

## Old Vector Coordinate

This is displaying the current starting location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

## New Vector Coordinate

This is displaying the current ending location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

## Select Button

This button will let you select that vector you are currently editing by allowing you to pick that vector on screen.

## New Button

This button will create a new vector from the coordinates displayed in the OLD and NEW fields above. It will duplicate a vector if you pick this button while old values are in the fields above but you can still manipulate those vectors by using the two pick buttons to the right of the NEW and OLD coordinates displayed.

## Delete Button

This button will delete the currently selected vector, both from the routine and the screen.

## Delete All Button

This button will delete all the vectors on the layer RASTER.

## Raster Layer Display

This location displays the number of vectors detected on the layer RASTER.

Note: Do NOT use this layer for anything else or you will confuse the program.

## Pick Button for New Coordinate

This button allows you to pick a new location for the vector to end. This is the ending location of the transformation for this vector. You can pick any where on the screen as the original coordinate for the vector.

## Pick Button for Old Coordinate

This button allows you to pick a new location for the vector to start. This is the starting location of the transformation for this vector. In other words, the amount of transformation will be determined by the start and end of the vector. The amount of transformation is the delta X and delta Y of this vector. You can pick any where on the screen as the starting (old) coordinate for the vector.

## 4-Point-Transformation



**Command: RTRANS**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This simple transformation allows you to stretch the raster image to match your vector line work. You are required to enter 4 vector translations in the edit/define transformation vectors menu. Then when you select this option the stretching of the raster will be done. Make sure you enter exactly 4 transformation vectors as this routine will not run with more or less than four! Once the transformation is completed, you will be prompted for the properties to confirm the insertion point color, etc.

## Drawing Output



**Command: PRINT**

**Alias PT**

**Located:**

1. **Pull-down Menu: File – Print**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

Start the output of a drawing on the printer/ plotter or plotter by selecting the option *Print ...* or by entering the command PRINT.



Select one of the options available in the options bar or menu (depending upon which you have selected for use in Desk Manager under Options. These options allow the user to determine the size and position of the print zone or the area of the drawing to be output. If you have Alternately, you may choose to use the first letter of the option to indicate your selection.

### **View (V)**

The currently visible window of the drawing will be copied on the print.

### **Border (B)**

This option determines the print zone so that all objects included in the drawing are covered.

Note that before using this option the drawing must be regenerated (Command REGEN).

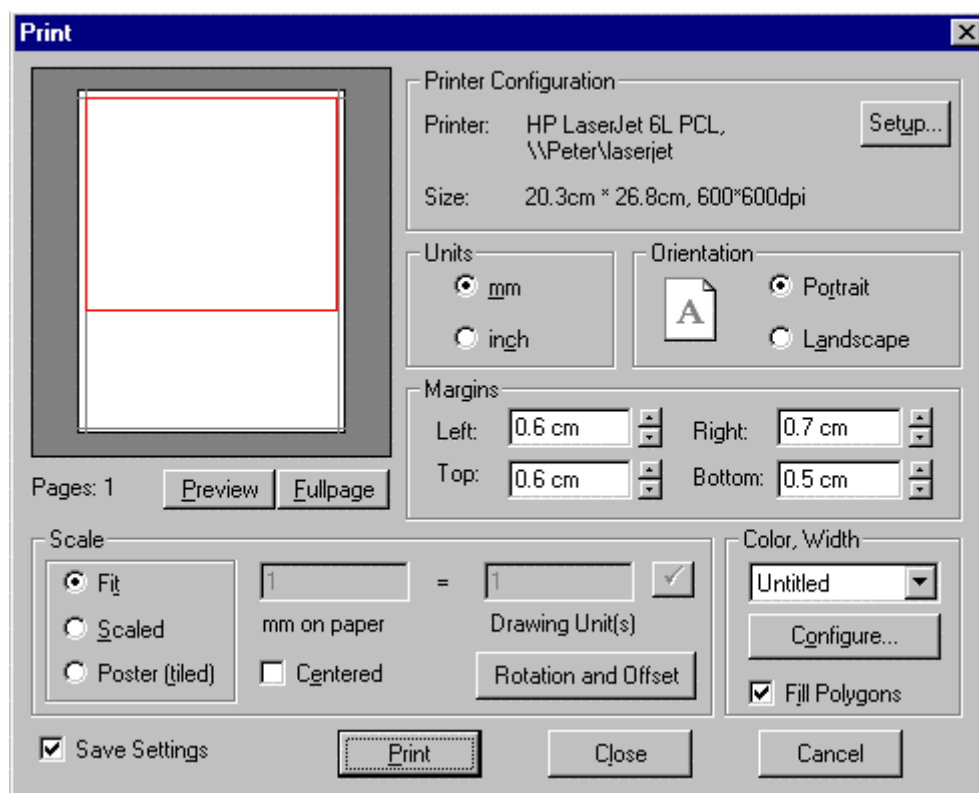
### **Window (W)**

Window enables the user to determine the print zone by selecting a window in the drawing. Indicate or enter the coordinates of two diagonal opposite corner points inside the drawing.

The command PRINT controls output of drawings to printers or plotters configured on your system. It uses the Windows print drivers and by this requires that the appropriate drivers have been set up under Windows.

## **The Print dialog box**

After defining the size of the print, the Dialog Box *Print* will open, and additional print parameters required for the orientation of the print in relation to the edges of the paper format and to the picture size must be set.



The Dialog Box is divided into the areas Configuration, Layout, Color/ Width, Pages and Scaling. It also contains a preview window and selections for *OK*, *Escape* and *Setup*.

The *preview window* will assist the user in controlling the effects of setting changes in Layout and Scaling.

## **Printer Configuration Area**

The Printer/ Plotter configuration area contains the options for the current Printer/ Plotter, including the desired paper size and print resolution (in dpi).

### **Setup**

Selection of the *Setup* switch allows the user to influence such parameters as the paper feed (printer/ plotter drawer), paper size, format position, resolutions and the number of the copies to be run.

Note: Only use settings supported by the selected device in the SETUP Dialog Box.

### **Layout Area**

The parameters of the Layout Area control the size and position of the print on the available format.

#### **Margins**

Controls the area that is not to be printed (in mm) at the sides of the page. Enter the desired values in the proper window or select with the Cursor at the right edge of the fields and set the values for margin width.

#### **Fit**

Selecting the option *Fit* will result in a format-filling picture of the object in the print zone.

This option is activated by default during any print or plot output. To de-activate, click the control field. A cross in the control field displays the active status of the option.

Note that when the option *Fit* is active, it is not possible to set the options for scaling.

#### **Fill Area**

*Fill Area* will cause filled polygon areas to be displayed with fill in the print output. If the option not activated, only the contours of filled areas will be output.

The default position of this option is *Inactive*. To activate select the control field. A cross in the control field displays the *Active* status of the option.

#### **Center**

*Center* has a very close relation to the option *Fit*. Because the relation between the pages of the print zone remains unchanged during *Fit*, a format-filling picture may only be reached either to the length *or* width of the picture, but not with both.

*Center* results in a centered placement of the print zone on the paper in relation to the side that could *not have been fitted* to the margins of the page.

Note that *Center* is only available when using the *Fit* option.

#### **Area Scaling**

*Area Scaling* allows the user to define the exact scale of the drawing.

Note that *Area Scaling* is only available when the *Fit* option has been *de-activated*.

The definition of the picture scale is made in two steps:

1. Define the size of a drawing unit in mm. Place the cursor in the input field for the drawing unit - mm and enter the desired value.
2. In the input field Scale enter the scale for the print output or select a value by means of the arrow keys at the right side next to the input field.

The user can view the effects of the inputs in the preview window.

#### **Area Pages**

The *Area Pages* contain a Dialog. Enter the number of pages to be used in the drawing.

Multiple pages will result when the user has selected a picture scale that results in a print output larger than the selected page size (minus the defined margins).

#### **Area Color / Width**

By choosing the *Configure* button, the Color/Width Configuration Dialog will appear.

*Using saved stroke width-attachments*

Using saved stroke widths facilitates outputting drawings and assures constant settings.

In the selecting window of the area *Color/ Width* in the Dialog Window *Print* the current stroke width will be displayed. Selecting the arrow at the right side of the window will display a list of all saved configurations. Select the desired configuration.

## Command BMP2RLC

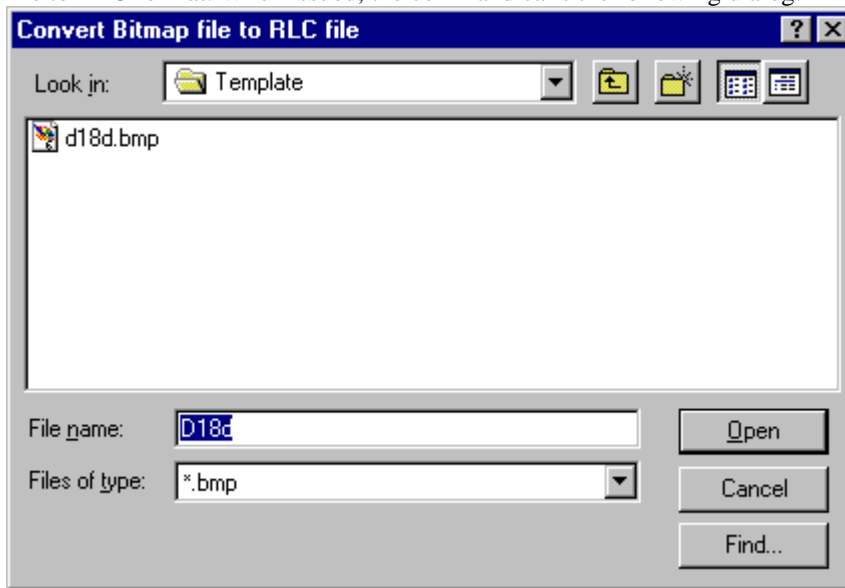


### Command BMP2RLC

#### Located:

1. **Palette: Raster**
2. **Or Enter Command Directly at the Command Line**

Like its counterpart (RLC2BMP), BMP2RLC simply converts an external monochrome bitmap (BMP) file to RLC format. When issued, the command calls the following dialog:



To complete command execution, simply navigate to your BMP file through the "Look in:" drop down window and select the file. The file is then immediately converted with the new file remaining in the same location as the original and having the same filename but with .rlc extension.

## Command\_RLC2BMP

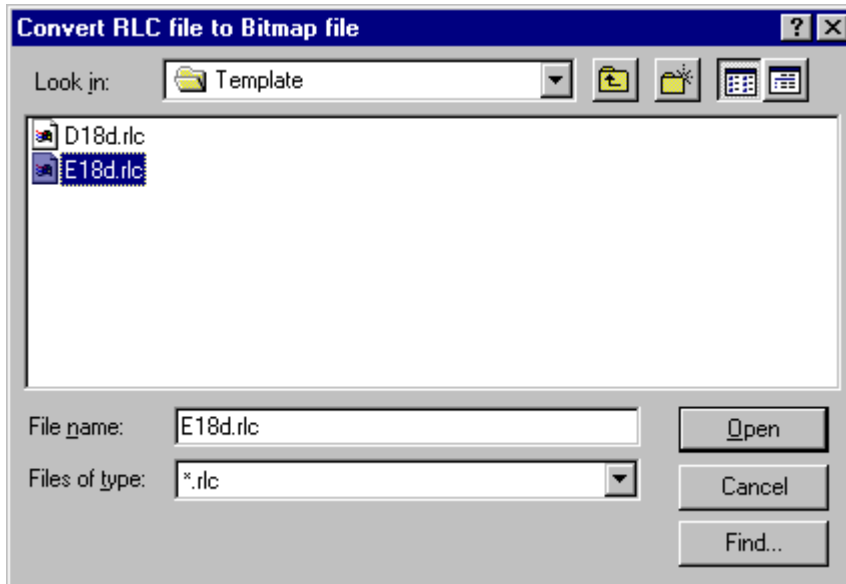


### Command\_RLC2BMP

#### Located:

1. **Palette: Raster**
2. **Or Enter Command Directly at the Command Line**

Like its counterpart (BMP2RLC), RLC2BMP converts an external RLC graphic file into bitmap (BMP) format. When the command is issued, the following dialog appears:



To complete command execution, simply navigate to your RLC file through the "Look in:" drop down window and select the file. The file is then immediately converted with the new file remaining in the same location as the original and having the same filename but with .bmp extension.

## Rendering

### Render



**Command: RENDER**  
**Alias REN**

#### Located:

1. **Pull-down Menu: View – Render – Render**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**

Rendering allows you to create an image of your 3D drawing that illustrates very realistically the model or plan. To increase the realism, the rendering may perform smooth shading on the objects that meet at a specific angle and evaluate the lights set in the drawing.

Using rendering, you visualize and present your 3D geometry as it would appear, after construction.

To render the current viewport of a drawing, call the command RENDER. The command is immediately performed after you have specified where to display or output the rendered view:

| Screen | File | Clipboard |

The rendering is based on the preferences set with the RCONFIG command and evaluates the light sources found in the model. The command LIGHT allows you to place lights in the drawing and define their type and settings. The command LIGHTEDIT is used to modify the settings of light sources.

---

#### See also:

RCONFIG

## Render Settings



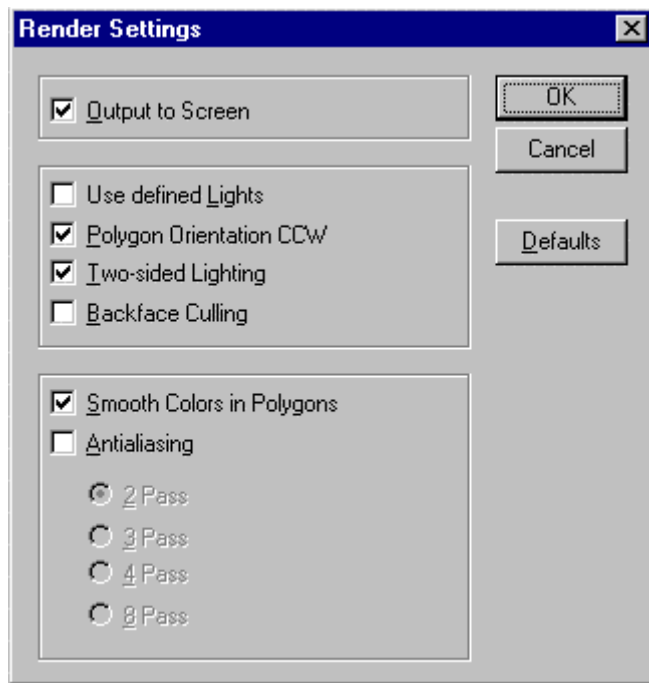
Command: RCONFIG

Alias RC

### Located:

1. Pull-down Menu: View – Render – Render Parameters
2. View Palette
3. Or Enter Command or Alias at the Command Line

The command RCONFIG displays a dialog box to view and set general preferences for subsequent rendering.



The button **Defaults** resets the preferences to default values (shown in the illustration above). This is convenient, because a number of settings can influence the performance of render computations dramatically.

The option **Output to Screen** controls whether the rendered polygons are output on the screen (to the front buffer) during computations or are written into a display buffer before presenting the rendered image. If this check mark is not set, the rendered image is not displayed before the entire rendered image is computed.

The option **Use Defined Lights** allows you to activate or deactivate the use of light sources. If disabled, the program generates - in computer graphic terms - a simple **shading** of the 3D model, otherwise the inserted lights are utilized during the rendering.

Three other options allow users who are familiar with computer graphics and advanced rendering techniques to save time when careful drawing methods have been used during the creation of a 3D model: **Polygon Orientation CCW**, **Two-sided Lighting**, and **Backface Culling**. It is recommended to use the default values if you are not familiar with these terms.

The option **Smooth Colors in Polygons** specifies whether upon each individual polygon surfaces smoothing colors are computed or not. Applying an adequate light environment allows smooth coloring on individual surfaces.

The option **Anti-Aliasing** allows you to control the process of smoothing jagged edges caused by aliasing. Aliasing is a term in computer graphics. It describes certain display effects like the staircase effect seen on diagonal lines displayed in a drawing. Anti-Aliasing can reduce those artifacts.

Turn anti-aliasing on for high-quality rendered images. To improve the rendering significantly you can choose between **2, 3, 4 or 8 passes** with the anti-aliasing algorithm. Because multiple passes will increase the rendering time dramatically, anti-aliasing should be applied only for high-quality image output (usually when rendering to a file).

By default the anti-aliasing option is disabled. It produces a faster output for viewing a rendered image on the graphics display, and you do not normally notice the elimination of artifacts on the computer screen.

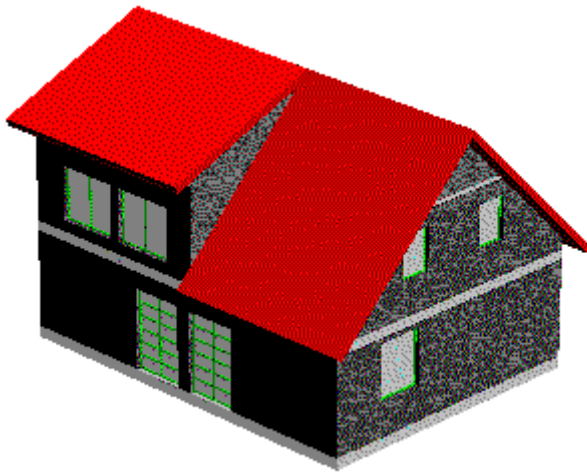
---

**See also:**

RENDER

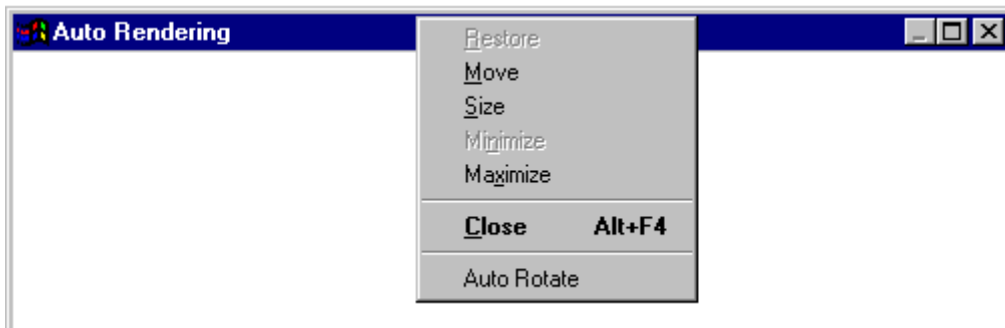
## AutoRender

The Arender command (View > AutoRender) displays a rendering of the current drawing in a separate pop-up window.



You can rotate the rendering by situating your pointer inside the Arender window, and holding down the left mouse button. While the mouse button is depressed, the icon will change to a small hand, allowing you to rotate the image about the screen.

Zooming and panning for Arender is accomplished by holding down the right mouse button. While depressed, you can slide the pointer up or down to zoom and pan the image.



To auto-rotate (spin) the rendering image, right click on the blue command line area and toggle the auto-rotate setting.

For the control of lines and hatches with ARENDER, see ARLINEDISP.

## Print and Data Exchange

### Printer Setup

**Command:** PSETUP

**Alias** PS

**Located:**

1. **Pull-down Menu:** File – Printer Setup
2. **Or Enter Command or Alias at the Command Line**

The command PSETUP (Printer/ plotter Setup) or the selection of the option **Printer Setup...** from the *File* menu will start the selection and installation of the desired printer or plotter.

This procedure for installing printers calls up the standard Windows printer/plotter installation sequences.

The user will find detailed instructions regarding the operating mode of the specific options for the printer installation and for installing and connecting a printer or plotter in the Windows Manual or in the output device manual. Only the most important steps for printer/plotter installation are reviewed in this text.

The command will open a dialog box.

### Dialog Control Elements

#### Printer

In the field Printer/ Plotter the user will specify the printer/ plotter or plotter to be used in the output of the drawing. Select either one of the standard devices defined in the Dialog Box control, or select another printer or plotter that is available under Windows. Available output devices are displayed in a list in the selection area under the heading *Special Printer*. To see the complete list, select the arrow at the right edge of the window. Select from the list of the installed output devices the desired printer.

#### Orientation

Select the format **Portrait** or **Landscape** for the desired alignment of the paper.

#### Paper

The size of the paper as well as the feed from a particular printer or plotter is controlled from the

field **Paper**. Select the proper size and feed.

### Options

The **Options** switch (at the right edge of the Dialog Box) opens another Dialog Box that contains parameters that are mainly a function of the capability of the device to be connected.

## Drawing Output



**Command: PRINT**

**Alias PT**

### Located:

1. **Pull-down Menu: File – Print**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

Start the output of a drawing on the printer/ plotter or plotter by selecting the option *Print ...* or by entering the command PRINT.

Select one of the options available in the options bar or menu (depending upon which you have selected for use in Desk Manager under Options. These options allow the user to determine the size and position of the print zone or the area of the drawing to be output. If you have Alternately, you may choose to use the first letter of the option to indicate your selection.

### View (V)

The currently visible window of the drawing will be copied on the print.

### Border (B)

This option determines the print zone so that all objects included in the drawing are covered.

Note that before using this option the drawing must be regenerated (Command REGEN).

### Window (W)

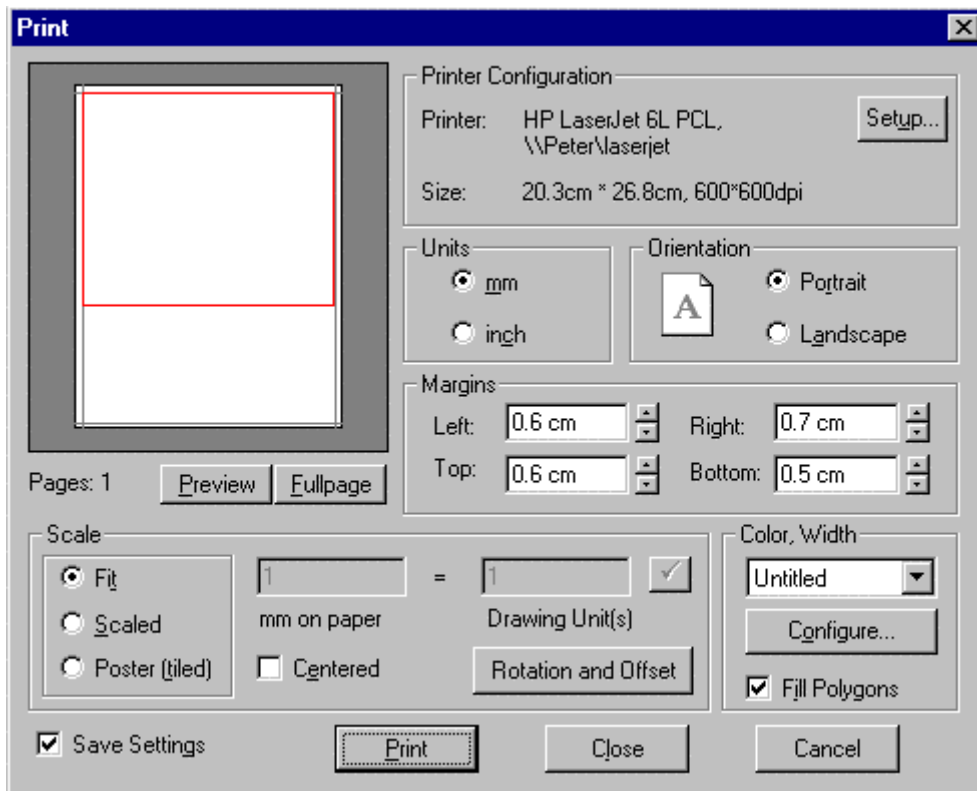
Window enables the user to determine the print zone by selecting a window in the drawing. Indicate or enter the coordinates of two diagonal opposite corner points inside the drawing.

The command PRINT controls output of drawings to printers or plotters configured on your system. It uses the Windows print drivers and by this requires that the appropriate drivers have been set up under Windows.

### The Print dialog box

After defining the size of the print, the Dialog Box *Print* will open, and additional print parameters required for the orientation of the print in relation to the edges of the paper format and to the picture size must be set.





The Dialog Box is divided into the areas Configuration, Layout, Color/ Width, Pages and Scaling. It also contains a preview window and selections for *OK*, *Escape* and *Setup*.

The *preview window* will assist the user in controlling the effects of setting changes in Layout and Scaling.

### Printer Configuration Area

The Printer/ Plotter configuration area contains the options for the current Printer/ Plotter, including the desired paper size and print resolution (in dpi).

### Setup

Selection of the *Setup* switch allows the user to influence such parameters as the paper feed (printer/ plotter drawer), paper size, format position, resolutions and the number of the copies to be run.

Note: Only use settings supported by the selected device in the SETUP Dialog Box.

### Layout Area

The parameters of the Layout Area control the size and position of the print on the available format.

#### Margins

Controls the area that is not to be printed (in mm) at the sides of the page. Enter the desired values in the proper window or select with the Cursor at the right edge of the fields and set the values for margin width.

#### Fit

Selecting the option *Fit* will result in a format-filling picture of the object in the print zone.

This option is activated by default during any print or plot output. To de-activate, click the control field. A cross in the control field displays the active status of the option.

Note that when the option *Fit* is active, it is not possible to set the options for scaling.

#### Fill Area

*Fill Area* will cause filled polygon areas to be displayed with fill in the print output. If the option not

activated, only the contours of filled areas will be output.

The default position of this option is *Inactive*. To activate select the control field. A cross in the control field displays the *Active* status of the option.

### **Center**

*Center* has a very close relation to the option *Fit*. Because the relation between the pages of the print zone remains unchanged during *Fit*, a format-filling picture may only be reached either to the length *or* width of the picture, but not with both.

*Center* results in a centered placement of the print zone on the paper in relation to the side that could *not have been fitted* to the margins of the page.

Note that *Center* is only available when using the *Fit* option.

### **Area Scaling**

*Area Scaling* allows the user to define the exact scale of the drawing.

Note that *Area Scaling* is only available when the *Fit* option has been *de-activated*.

The definition of the picture scale is made in two steps:

1. Define the size of a drawing unit in mm. Place the cursor in the input field for the drawing unit - mm and enter the desired value.
2. In the input field Scale enter the scale for the print output or select a value by means of the arrow keys at the right side next to the input field.

The user can view the effects of the inputs in the preview window.

### **Area Pages**

The *Area Pages* contain a Dialog. Enter the number of pages to be used in the drawing.

Multiple pages will result when the user has selected a picture scale that results in a print output larger than the selected page size (minus the defined margins).

### **Area Color / Width**

By choosing the *Configure* button, the Color/Width Configuration Dialog will appear.

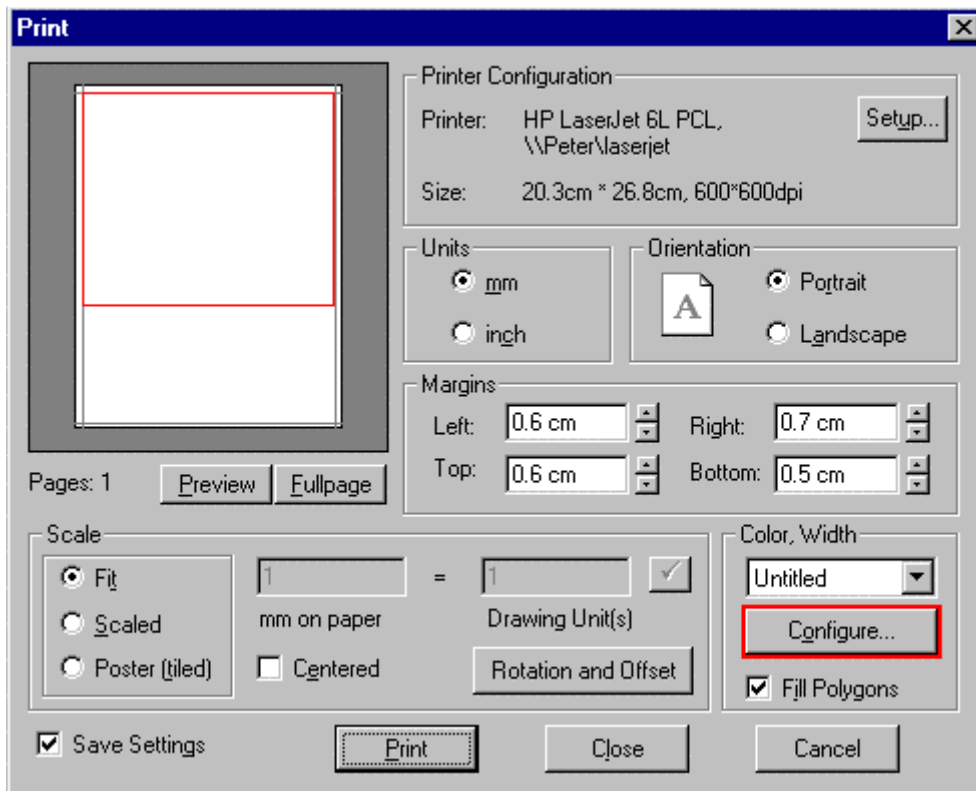
*Using saved stroke width-attachments*

Using saved stroke widths facilitates outputting drawings and assures constant settings.

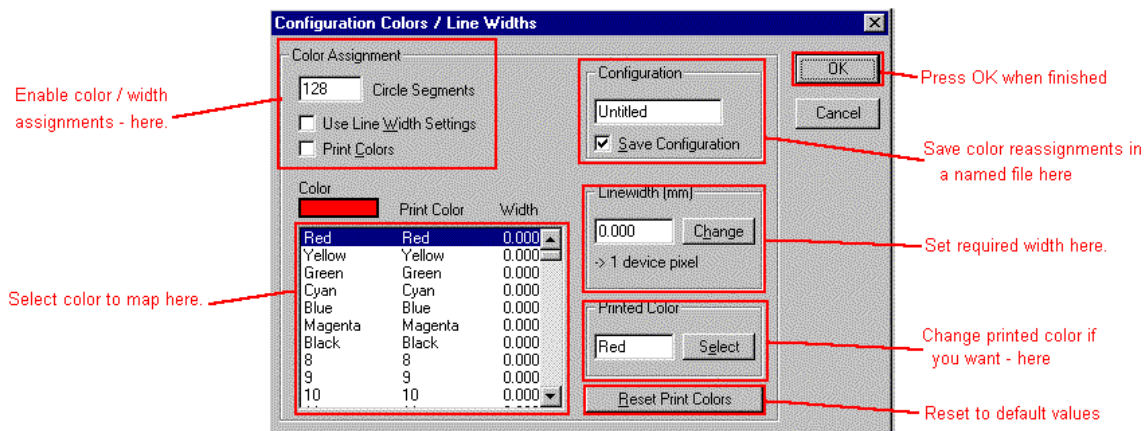
In the selecting window of the area *Color/ Width* in the Dialog Window *Print* the current stroke width will be displayed. Selecting the arrow at the right side of the window will display a list of all saved configurations. Select the desired configuration.

## **Print: Color / Width Configuration Dialog**

*Area Color / Width* defines the widths of the stroke for the output of the printer/ plotter or plotter. This choice of line widths permits the structuring of a drawing and improving its legibility. You set the color - line width associations by clicking on the Configure button on the PRINT command dialog.



You will then be presented with the following dialog:



The definition of stroke width is a combination of width and color (or color identification number). The current setting of stroke widths can be saved under a name and then, like a parameter, used again for future printing operations.

Every object has been assigned a color that is defined by a color identification number (also called color index) already. For detailed instructions on the definition of colors for layers or for objects, see the Help section "Layer and Object Properties".

During the preparation of the print output, any defined color can be given a stroke width defined between 0 and 50 mm.

Select the area *Color / Width* of the Dialog Box *Print* and then select *Config*. The dialog box containing *Color/Width Configurations* will appear.

## Switching Line Width Selection On/ Off

Independent of the correlation of color identification numbers and stroke widths, the option to output different line widths can be switched on or off. Select the control field *Use Thickness Settings*. A cross in the control field displays the active status of the option.

There is a second control field with the name *Print Colors*. This field will switch the color printing On or Off. Of course, the output device must have color capability to print in color.

To activate or de-activate the option *Print Colors* select the control field. A cross in the control field displays the active status of the option.

## Attach stroke widths

Use the selection area for coordinating color and width.

Select the desired color or its color index (color identification number). In the selecting window the color identification number and the present stroke width is displayed. In the preview window *Color Index* the user can preview the selected color.

Enter the desired value in the input field *Stroke Width*. A range from 0 to 50 mm is possible. Attach the entered stroke width to the previously selected color by pressing the key *Change*.

## Save Assignments

The program allows the user to save the correlation between line widths and colors.

To do this, enter a name in the area *Configuration* for the current setting of the stroke widths in the input field there. Switch on the function *Save Configuration* by selecting that control field. A cross in the control field displays the active status of the option.

Confirm the settings for the stroke widths by OK.

## Copy Region to Bitmap



Command: **BMPOUT**

Alias **BO**

Located:

1. **Pull-down Menu: File – Export Region To Bitmap**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

Call the command either by keyboarding **BMPOUT** or by selecting **Copy Region to Bitmap...** from the menu *Edit*.

The following options allow all **entities** in the region to be copied into a bitmap:

### Window

Define a window by setting two points.

### View

The complete current view will be exported into a bitmap.

### Extents

The region to be copied is defined by the extent of the drawing.

Next, specify the **Width** and **Height** of the bitmap in Pixel Units (10...2000 pixels).

Finally, you may choose whether the bitmap should be written into a **File** or copied to the Windows **Clipboard**.

### Notes:

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The bitmap remains in the clipboard until you clear it. Place new objects on it with the **RENDER**, **BMPOUT**, **WMFOUT** or other command, or Cut or Copy in other Windows

applications.

## Copy Region to Windows Metafile



**Command: WMFOUT**

**Alias WM**

### **Located:**

1. **Pull-down Menu: File – Export Region to Metafile**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

The following options allow the region to be copied into a WMF vector format file:

### **Window**

Define a window by setting two points.

### **View**

The complete current view will be exported into a bitmap.

### **Extents**

The region to be copied is defined by the extent of the drawing.

Finally, you may choose whether the picture should be written into a **File** or copied to the Windows **Clipboard**.

### **Notes:**

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The WMF vectors remain in the clipboard until you clear it.

## Command COPYSELECT



**Command: COPYSELECT**

**Alias CST**

### **Located:**

1. **Pull-down Menu: Edit – Copy Selection to Clipboard**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

The command COPYSELECT copies selected objects from the drawing, placing them onto a **AllenCad-specific** clipboard. The selected objects remain in the drawing.

The command COPYSELECT is useful especially to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

### **To copy objects onto the clipboard:**

1. First specify the base point for later insertion of the objects.
2. Then select the objects to be copied to the clipboard.

**Command line example:**

```
> COPYSELECT  
Insertion base point:  
Select objects:
```

**Note:**

Do not confuse this command with the COPY command found in the *Modify* menu that makes copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another drawing. They will remain on the clipboard until you place new objects on it with the COPYSELECT command.

---

**See also:**

Command PASTE  
Command CUTSELECT

## Cut Selection to Clipboard

**Command: CUTSELECT****Alias CT****Located:**

1. **Pull-down Menu: Edit – Cut Selection to Clipboard**
2. **Top Function Bar**
3. **Palette Edit**
4. **Or Enter Command or Alias at the Command Line**

With the command CUTSELECT selected objects are copied to a **AllenCad-specific** clipboard (as with the command COPYSELECT)

However, unlike COPYSELECT, with CUTSELECT, the originally selected objects are deleted from the drawing.

The command PASTE will insert the objects from the PCAD clipboard into any open drawing. PASTE will request the insertion point.

---

**See also:**

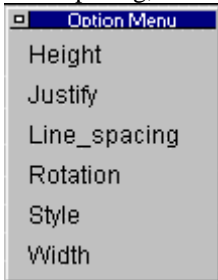
Command COPYSELECT  
Command PASTE  
Command DELETE

**Command MTEXT****Command MTEXT****Alias MT****Located:**

1. **Pull-down Menu: Detail – Multiline Text**
2. **Palette: Detail**

### 3. Or Enter Command or Alias at the Command Line

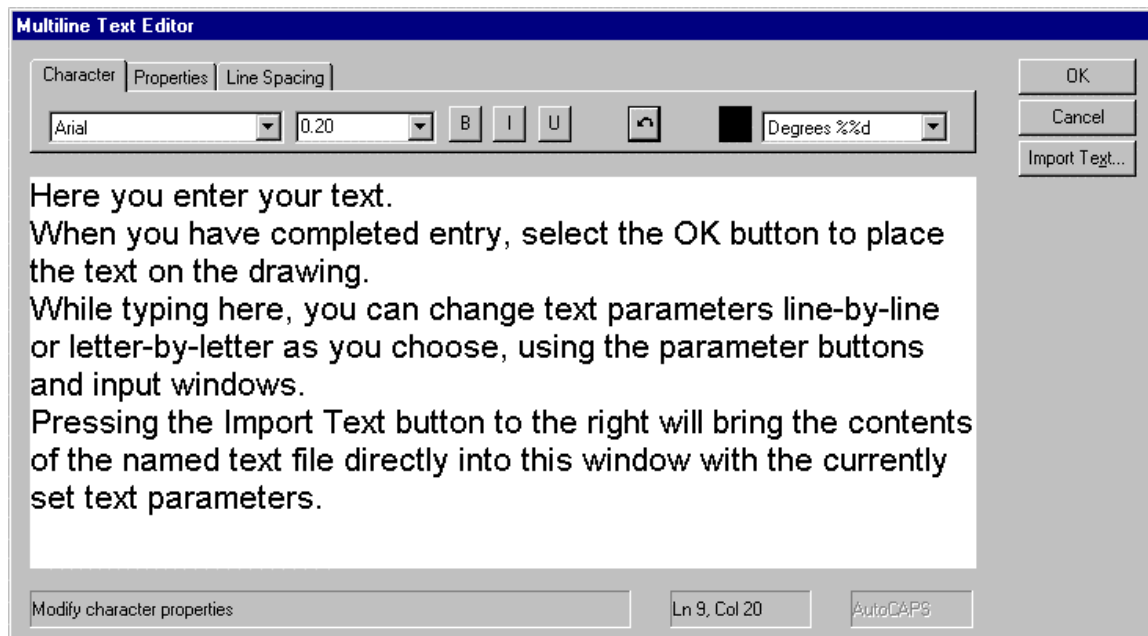
The MTEXT command provides AllenCad with full multiline text functionality. After issuing the command you are asked to specify the location of the new text by picking an upper left and lower right corner on the drawing. This window roughly locates the placement for the text but is essentially used to determine the line length. Text will be wrapped into the next line if the available width is insufficient. While specifying the two window points, you are presented with options to change Height, Justification, Line Spacing, Rotation, Style and Width.



You may choose to set these values here, however these options are also available in the dialog that follows.

Next, you are presented with a windowed dialog box in which you enter your text. Three tabs allow you to exercise control over all text parameters and a **Text File Import function** enables you to import an external text file into the window. These duplicate and add to the earlier options that were presented and you can set your preferences in either manner.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's MTEXT command dialog window, either through picking on the window and using Windows' "Ctrl V" function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry



#### MTEXT Dialog showing Character Options.

**Note:** to allow the text to fill the indicated space in your drawing and to line wrap as needed within that designated width, be sure to continue the line entry beyond the visual extents of the entry window. Pressing the Enter button as we have done in the above example will allow you to see all the text without

scrolling left-right, however it may not give the desired effect when this text is transferred to the drawing.

Most of these windows and buttons you are probably fairly familiar with if you do much word processing. Viewed left to right, these windows and buttons:

1. Specifies the required font
2. Toggles between bold and regular type
3. Italicizes text
4. Underlines Text
5. Removes the latest entry or reverses the latest action
6. Indicates the color to be applied to text. Click on this to change the setting
7. The last button allows you to call up special symbols (diameter, degree, plus/minus)



#### **MTEXT Dialog showing Property Options.**

Here, you can specify a named text Style, set Justification, override text Width originally set using the picked window points, and finally, Rotation, where you can specify an angle or bearing at which to place the entered text.



MTEXT Dialog showing Line Spacing options.

Here you can specify line spacing distance -either a minimum (At Least) or (Exactly)

See Also MTEXTEDIT , QMTEXT

## **Creating and Viewing Slides**

### **Commands: MSLIDE**

#### **MSLIDE is Located:**

1. Enter Command or Alias at the Command Line

### **VSLIDE (Alias VS),**

#### **VSLIDE is Located:**

2. Pull-down Menu: View – View Slide
3. Or Enter Command or Alias at the Command Line

### **SLDOUT (Alias SO)**

#### **SLDOUT is Located:**

1. Pull-down Menu: View – Take Snapshot (Slide)
2. Or Enter Command or Alias at the Command Line

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.



You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

#### **To create a slide file**

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.
3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

#### **To view a slide file**

Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

## **Batch Processing**

### **BATPROC**

#### **Command BATPROC (Batch Processing)**

**Alias BT**

##### **Located:**

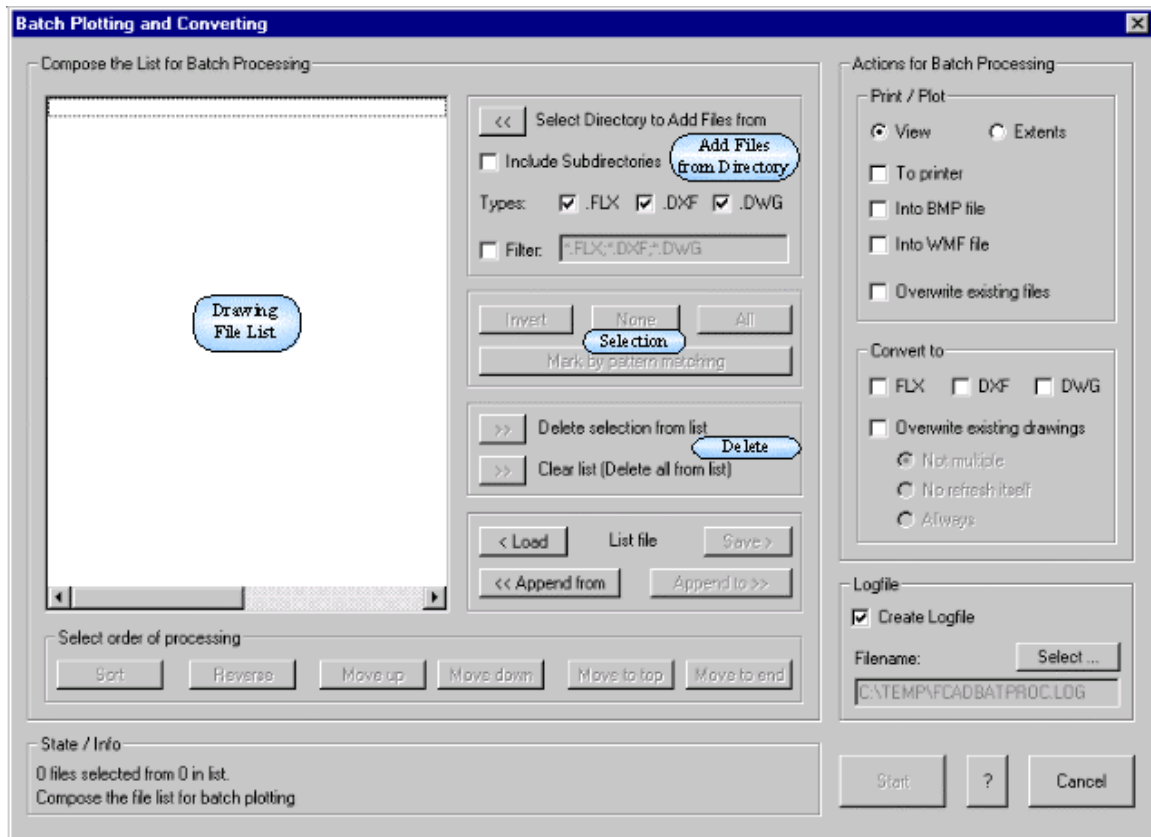
1. **Pull-down Menu: File – Batch Processing**
2. **Or Enter Command or Alias at the Command Line**

#### **Introduction**

The command BATPROC makes it possible for you to arrange a list of drawing files and to do something with each drawing on this list. There are many built in capabilities to build the list and manipulate it. After arranging the list you can plot each file on the list not only to the printer but also to a Bitmap file (.BMP) file or into a Windows Metafile (.WMF). Moreover you can batch-convert drawings from one format (of the set DXF, FLX or DWG) to another of this set.

#### **Preferences and Start of the Command**

You can start the command simply by typing "BATPROC" or by selecting the menu point BATCH PROCESSING inside of the menu TOOLS. Now the following dialog box appears at the screen:

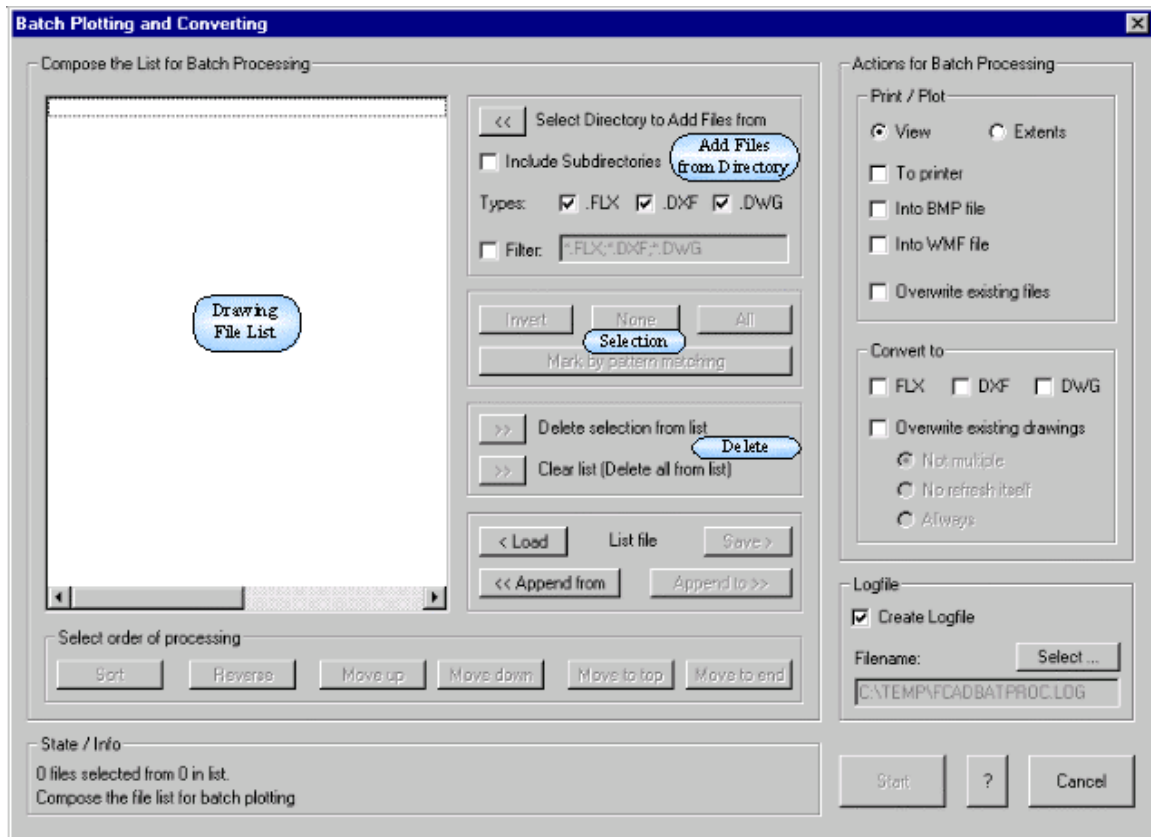


This dialog is divided in two main parts, on the left side you find the functions to compose and manipulate the list, while on the right part the actions for the batch process can be defined. Following the meaning of the fields will be described.

### **BATPROC- Actions for Batch Processing**

## **Defining the Actions for the Batch Processing**

After composing the list, the actions for all items of the list have to be specified. The related fields are placed on the right side of the dialog box.



### 1) The Plotting/Printing Field

Field provides the following actions:

Plotting to the printer.- Before you start the batch printing, it's necessary to select the printer and the options for printing. It's highly recommended that you open one drawing of the files that should be plotted and print this using the set options. These options apply to each drawing of the list.

Plotting into an WMF-file (Windows Metafile)

Plotting into a BMP-file (Bitmap)

With the radio buttons "View / Extends" you can decide if you want to plot the drawing like it's stored, or if it is zoomed to extents before plotting.

The switch Overwrite Existing files determines, if BMP and WMF files will be overwritten or not.

### 2) The Converting Field

This allows selection of the target format that can be FLEXIBLE DRAWING EXCHANGE (FLX), DXF or DWG. The AutoCAD formats will be written for release 12. It's possible to realize multiple conversions, i.e. each drawing can be converted to each other format at once. But, in this case it's highly recommended to switch off the overwrite option, because enabling overwriting of existing files can destroy the original drawings. If you switch on the overwriting option, the batch processing tool should be only applied to files, a backup exits from.

The options for overwriting existing files have the following meaning:

The setting "Not multiple" prevents overwriting the file, if it have already written or if it is inside of the drawing file list, that will follow. As an example assume you have the composed a list, that contains both, a file conv.flx and a file conv.dwg. These files should be converted into both the DXF and the DWG format. If the unlimited overwriting would be set, the first file conv.flx would be open a saved as conv.dxf and as conv.dwg. So it would be overwrite the existing file conv.dwg too. Because this file is also in the list, it would be opened again.. To prevent such cases it's highly recommended to use this option, if you activate overwriting of drawing files. The technical background of this option is, that for each file that

would be overwrite another one is checked, if this file have already be written by the batch process or if it is part of the remaining list. In this case the file won't be overwritten.

The option "Not refresh itself" only prevents overwriting.

The option "Always" allows you to overwrite each file - itself too.

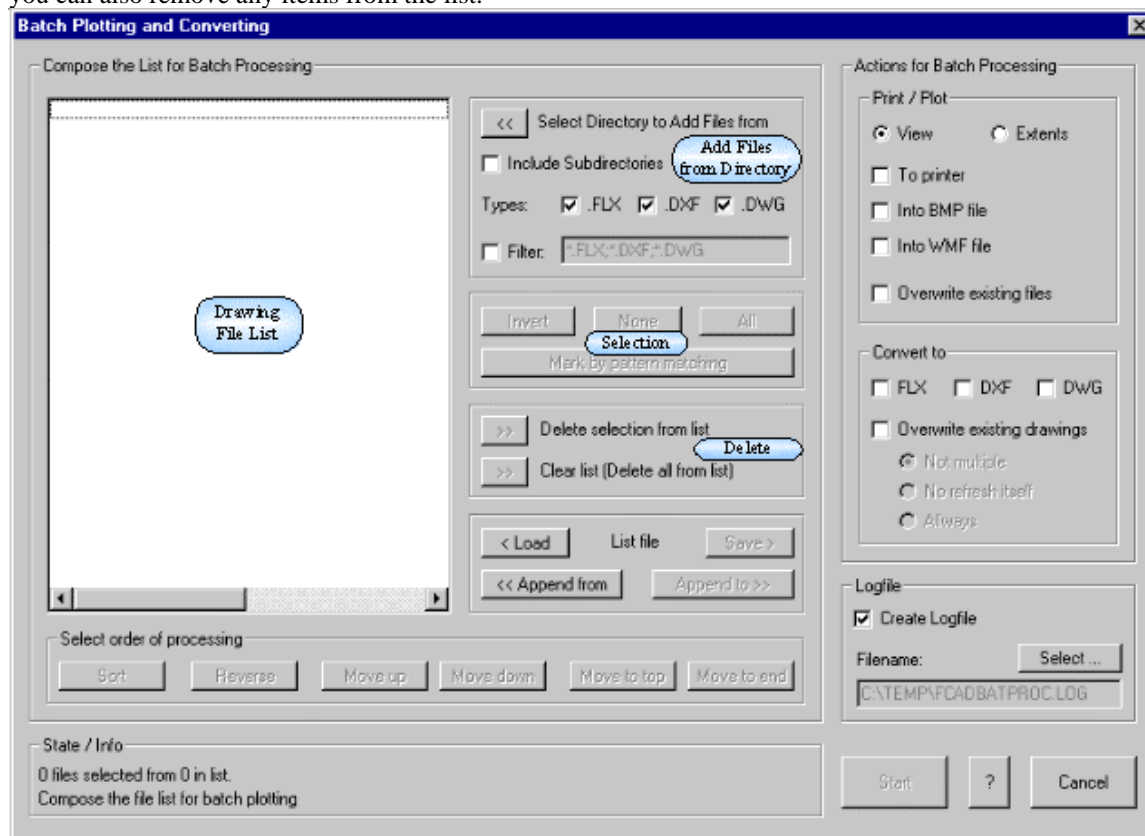
### 3) The Log File Field

This allows you to define, if a log-file should be written during the batch processing. Moreover, you can select a filename for it. If this already exists, it will be overwritten. It's useful to leave this option activated, in other case after finishing the batch processing there is no information about any error available.

## BATPROC - Composing and Arranging the List

### Functions for Composing and Arranging the List

The first step consists of composing a list of drawing files. Therefore, the tool provides you with the ability to add files from a any directory. It's also possible to load or append files from a list. Of course, you can also remove any items from the list.



## Adding Files from Directory

This field contains the ability to select a directory and add all files inside of the directory. Before pressing the button "Select Directory to Add Files from" you should setup the desired options. For that you can switch on the recursive search inside of subdirectories. Moreover you can select a filter for the files, you want to add to the list. Two different alternates can be selected here: The first and simple one by defining the desired set of extensions, while the second one provides the ability to apply a normal file filter that is conform to those of the system. Here you can also put in more then one file pattern separated by a space or a ';'.  
".

## List Box

When the list of drawing files has been filled by adding files (see above) or by loading / appending an

existing file list, you can select some files by marking them inside of the box. You can use the mouse in combination with the SHIFT- or CONTROL- Key to get a multiple selection. The selected files will be highlighted. Note that if you start the processing, all files inside of the list but not only the highlighted files will be used.

## Selections

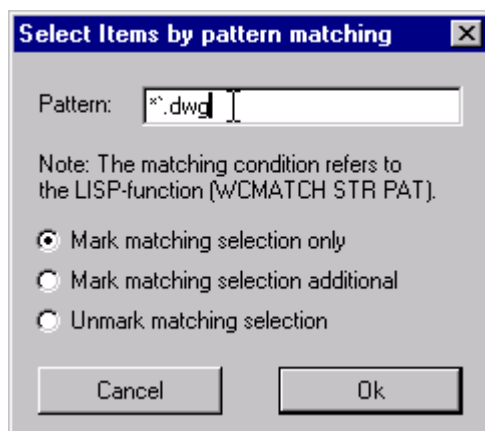
After selecting or deselecting items of the list box you can change the selections in four ways:

"All" will select all items inside of the list.

"None" will deselect all selected items of the list. As a result, no item of the list will be selected.

"Invert" will invert the selection. That means all selected items will be deselected, while all other will be selected.

"Mark by Pattern Matching" will open a new dialog, there you can select or deselect items by applying one or more patterns to the list of drawing files. The following picture shows the dialog:



The meaning of the pattern matching is not the same as for DOS/ WIN - file names, but the same as the for wcmatch - function. See the "Customization Document" for more information about this.

Here is a short summary of the special characters:

*	=	any sequence of characters;
?	=	any single character;
@	=	any single letter;
#	=	any single number;
.	=	any non-alphabetical character;
,	=	separates patterns;
~	=	not the following match (works only at the first position)
[..]	=	any character inside
.-.	=	character in the range
[~ ]	=	any character, that is not inside of the brackets
'	=	suppresses the special meaning of a character above

### Delete List Items

Using these buttons, you can clear the list (fully clean up) or you can remove the selected items from the list. These functions do not delete any file from your file system.

## List File

This field contains four buttons that serve to store or reload a list of drawing files.

After pressing the "Load" button, you can select a file, that should contain a list of drawing files. If you select one, the old list will be substituted by the list inside of the file.

"Append from" will do almost the same like "Load", but the files in the list file will be added to the current list.

Using the "Save" button you can store the current drawing file list to a list file.

"Append to" appends the current list to a file.

**Note:** You can simply generate a drawing file list using the old DOS-Command dir:

```
> dir *.flx /S /B > C:/TEMP/draw.lst
```

## List Order

Five buttons will help you, to define the order of the processing:

1. "Sort" will sort the list in alphabetical order. If you apply this function, duplicate items will be removed too.
2. "Reverse" turns around the order of the items inside of the list box.
3. "Move up" will move all selected items by one item.
4. "Move down" moves all selected items down by one step.

5. "Move to top" moves all selected items up to the top of the list.
6. "Move to end" moves all selected items to the end of the list.

Note that all of these functions for changing the list order leave the selection unchanged.

## Info/State Field

This field informs you about the process and the results of all list actions. At the first line you can see the current state – I.E. how many files are in the list and how many are selected. The second line shows you the progress of the current action, and if ready the result of this action.

## Recover Damaged Drawing Files

### Command FLXRECOVER

**Command:** FLXRECOVER

**Alias** REC

#### Located:

1. **Pull-down Menu: File - Recover**
2. **Or Enter Command or Alias at the Command Line**

Beginning with the version 4.01 (Service Pack2) release, the FLX drawing format modifications permit the partial recovery of damaged drawing files. This utility, called by the FLXRECOVER command, is therefore available to FLX files saved in this (or later) release format. For damaged drawing files saved in AutoCAD DWG or DXF format, applicable AutoCAD recovery utilities should be used.

Usually, the need to use this utility becomes apparent when you experience difficulties opening an existing drawing file. File damage may apply to only a portion of the drawing database and the recovery utility will help you to salvage undamaged drawing elements from the file.

To use the recovery routine, simply issue the FLXRECOVER command from the pull-down menu selection or at the command prompt and select the damaged filename in the dialog presented.

## Chapter 3 Command Reference

### Command Reference Chart

AllenCad Command	Alias	Command Description
2DFACE	2F or SO	Creates solid-filled polygons
3DBOX	3B	Creates a three-dimensional box
<u>3DCONE</u>	3DC	Creates a three-dimensional solid cylinder
3DELLIPS	3L	Creates a 3d ellipsoid
3DFACE	3F	Creates a three-dimensional face
3DMESH	3M	Creates a free-form polygon mesh
3DMIR (Application)	M3 OR 3MR	Creates a mirror image of objects about a plane
3DOBJECTS (Application)	3O	Creates three-dimensional polygon mesh objects
3DROT (Application)	3R or R3	Moves objects about a three-dimensional axis
3DSPHER (Application)	3S or 3DS	Creates three-dimensional solid sphere
3DTORUS	3T or TOR	Creates a donut-shaped solid
3DVIEW	3V	Sets the three-dimensional viewing direction
3DWEDGE (Application)	3W or WE	Creates 3D solid with a sloped face tapering along the X axis
'ABOUT	AB	Displays information about the Program
' <u>APPLOAD</u> (Application)	AP	Loads and unloads applications and defines which applications to load at startup
ARC	A	Creates an arc
AREA	AA	Calculates the area and perimeter of objects or of defined areas
ARENDER	AE	Automated Rendering
ARRAY	AR or RA	Creates multiple copies of objects in a pattern
ASSISTANT	AST	Your one-stop drawing setup wizard
ATTDEDIT	ATD	Modify existing attribute definition
<u>ATTDEF</u>	AD	Creates an attribute definition
ATTEDIT	ATT	Modify part attributes, one by one
'ATTEXP (Application)	ATX	Extracts attribute data
' <u>ATTMOVE</u> (Application)	ATM	Move attribute text
ATTROT (Application)	ATR	Rotate attribute text
ATTVALUE	AV	Edit attribute text value
ATTXEDIT	AX	Edits the variable attributes of a block
BATPROC (Application)	BT	Batch processing - Plots and conversion
BHATCH	BH	Fills an enclosed area with a hatch pattern

BMAKE	BK	Defines a block using a dialog box
BMPEDIT (Application)	BE	Calls external BMP into Windows Paintbrush program for editing
BMPOUT	BPO	Saves selected objects to a file in device-independent bitmap format
BOUNDARY	BD or BO	Creates a boundary polyline (command line version of BPOLY)
BPOLY	BP	Creates a region or a polyline from an enclosed area
BREAK	BR	Erases parts of objects or splits an object in two
CALLOUT	CA	Inserts boxed text into the drawing with a directional arrow.
CHAIN	CN	Draws single entity continuous lines
<u>CHAMFER</u>	CHA	Bevels the edges of objects
CHANGE	CV	Changes the properties of existing objects
CHPROP	CHP	Changes the color, layer, linetype, linetype scale factor, and thickness of an object
CIRCLE	CI	Creates a circle
CLOSE	CL	Exit the drawing but not close the program
CONFIG	CFG	Configure program operating parameters
COPY	C or CO	Duplicates objects
COPYPROP (Application)	CP or MA	Change object properties to that of selected entity
<u>COPYSELECT</u>	CST	Copies objects to the Clipboard
CUTSELECT	CT	Copies objects to the Clipboard and erases the objects from the drawing
DELETE and ERASE	DEL	Removes objects from a drawing
DELFLK	DLK	Deletes FelixCAD lock files (*.flk)
DELPARTIAL	DPL	Delete part of an entity
DESKCONF	DSK	Configures the desktop options
DIGICONFIG	DC	Digitizer configuration
DIGIOFF	DGF	Switch digitizer mode off
DIGION	DGN	Switch Digitizer mode on
<u>DIMA3P</u>	D3P	Creates an angular dimension
DIMA4P	D4P	Creates an angular dimension out of four definition points
DIMALI	DAL	Creates an aligned linear dimension
DIMBAS	DBA	Continues a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension
DIMCENT	CM or DCE	Creates the center mark or the center lines of circles and arcs
DIMCON	DCN or DCO	Continues a linear, angular, or ordinate dimension from the second extension line of the previous or a selected dimension
DIMDIA	DDA or DDI	Creates diameter dimensions for circles and arcs
DIMDRAW (Application)	DDR	Creates and modifies dimension styles
DIMEDIT (Application)	DED	Edits dimensions
DIMHOR	DH	Horizontal dimensioning
DIMLIN	DL or DLI	Creates linear dimensions



DIMOBL	DOB	Modifies the orientation of existing linear dimensioning objects.
DIMORD	DOD or DOR	Creates ordinate point dimensions
DIMRAD	DRA	Creates radial dimensions for circles and arcs
DIMREST	DRE	Reinstate a stored style file so that it can be reinstated as the current style
DIMROT	DRO	Rotate existing dimensions
DIMSAVE	DS	Allows the user to combine the individual parameters and store them in a named dimensioning style
DIMSER	DSR	Adds multiple additional dimension labels in the same orientation
DIMTHOME	DTH	Undoes the movement of dimension text
DIMTMOVE	DTM	Moves dimension text
DIMTNEW	DN or DOV	Changes dimension text value
DIMTROT	DTR	Rotate dimension text
DIMTYPE	DTP	Creates and modifies dimension styles on the command line
DIMUPD	DU	Updates dimension
DIMVER	DVR	Labels dimensions along the Y (vertical dimension) axis
DINSERT	DIN	Loads but doesn't insert the part in the drawing
'DIST	D or DI	Measures the distance and angle between two points
DIVIDE (Application)	DIV	Places evenly spaced point objects or blocks along the length or perimeter of an object
DLGEDIT	DM	Dialog and Menu editor
DOFFSET	DOF or O	Creates concentric circles, parallel lines, and parallel curves
DPAN	DP	Pan dynamically
DRAWMODE	DMD	Specifies the display mode and size of point objects
DRAWORDER	DR	Changes the display order of images and other objects
DTEXT	DT	Displays text on screen as it is entered
DVIEW	DV	Defines parallel projection or perspective views
DXFIN	DXN	Imports a drawing interchange file
DXFOUT	DXO	Creates a drawing interchange file of the current drawing
DZOOM	DZ	Dynamic zooming
EDGE (Application)	EG	Changes the visibility of three-dimensional face edges
EDITPAR	EP	Dialog to change edit parameters
EINFO (Application)	LI or LS	Displays database information for selected objects
'ELEVATION	ELE	Sets elevation properties of new objects
ELLIPSE	EL	Creates an ellipse or an elliptical arc
ERASE	E	Removes objects from a drawing
EXIT	Q	Exit the program
EXPAND	EX, or EXP	Extends an object to meet another object
EXPLODE	X	Breaks a compound object into its component objects
EXTEND	ET	Extends an object to meet another object
FCOMPILE	FC	Compile a shape file to a binary font file
FHIDE	FHD	Creates a view in the current window.
FILLET	F	Rounds and fillets the edges of objects
FLIP	FL	Mirror and delete original entity

FLXRECOVER	REC	Repairs a damaged drawing
FONT	FT	Loads text styles from the specified font file and customize it to the individual requirement.
FREEHAND	FH	Cursor tracing function that places a 3-D polyline along the cursor path. Useful for freehand line and text drawing.
`GRAPHSCR	GS	Switches from the text window to the drawing area
GROUP	G	Creates a named selection set of objects
HATCH	H	Fills a specified boundary with a pattern
HELP (F1)	?	Displays online help
HIDE	HI or HD	Creates a second drawing file with a hidden line view
HPEDIT	HE or HED	Modifies an existing hatch object
HPUPDATE	HPU	Updates hatch pattern parameters to new settings
ICONMENU	IM	Creates menu selection with graphics using .mni files
'ID	ID	Displays the coordinate values of a location
INSERT	I	Places a named block or drawing into the current drawing
INSERTQ	IQ	Inserts an internal part into the drawing
INTERSECT	IN	Selects a location at the theoretical intersection of two linear entities
LAYCONF	LC	Save and restore layer settings
LAYER	LA	Manages layers and layer properties
LAYREF	LRF	Change object layer properties to match reference object
LENGTHEN	LEN	Lengthens an object
LIBRARY	LB	Manages collections of symbols and parts
LIGHT	LIT	Manages lights and lighting effects
LIGHTEDIT	LE	Modify light source settings
'LIMITS (Application)	LM	Sets and controls the drawing boundaries
LINE	L	Creates straight line segments
LINETYPE	LT or LTYPE	Creates, loads, and sets linetypes
LISTFILE	LF	View contents of an external file
LOADLISP	LL	Loads a LISP file for use
LOADLTYPE	LLT	Creates, loads, and sets linetypes
LOGFILEOFF	LOF	Closes the log file opened by LOGFILEON
LOGFILEON	LON	Writes the text window contents to a file
'MACRO	MAC or SCR	Executes a sequence of commands from a script
MEASURE (Application)	ME or MR	Places point objects or blocks at measured intervals on an object
MENU	MU	Loads a menu file
MERGE	MG	Import an external .flx drawing file
MIRROR	MI	Creates a mirror image copy of objects
MOVE	M	Displaces objects a specified distance in a specified direction
MSLIDE	SLD	Save current view of drawing as a slide
MTEXT	T or MT	Creates multi-line text
MTEXTEDIT	MTE	Edit multiline text / Import text files
NEW	N	Creates a new drawing file
NGON	NG or POL	Creates an equilateral closed polyline

OFFSET	O	Offset linear objects in parallel orientation
OOPS	OOPS	Restores erased objects
OPEN	OPN	Opens an existing drawing file
OSNAP	OS	Calls for manual object snap assignment
PALCLOSE	PLC	Removes a palette from the desktop
PALETTE	PA, or PLT	Places a palette on the desktop
PALMAN	PM	Opens the palette manager
PALOPEN	PLO	Opens a new palette
'PAN	P	Moves the drawing display in the current viewport
'PANDOWN	PD	Pans down
'PANLEFT	LFT	Pans left
'PANRIGHT	RGT	Pans right
'PANUP	PUP	Pans up
PARTDEF	B or PDF	Define a part (block)
PARTEXP	PX or W	Writes objects to a new drawing file
PARTLIB	LIB	Calls the part library manager
PASTE	PST	Inserts data from the Clipboard
PCONTOUR	PC	Creates a boundary polyline on top of other entities
PFACE	PF	Creates a three-dimensional polyface mesh vertex by vertex
PLAN	PLN	Displays the plan view of a user coordinate system
PLANVIEW	PV	Returns a 3d view to being planar to current UCS
PLBSETUP	PLB	Sets up or modifies existing part (block) libraries
PLINE	PL	Creates two-dimensional polylines
POINT	PO or POI	Creates a point object
POLY3D	3P	Creates a polyline of straight line segments in three-dimensional space
POLYEDIT	PE	Edits polylines and three-dimensional polygon meshes
POLYLINE	PL	Creates two-dimensional polylines
'PRECPAR	OS or PCP	Customizes the program precision settings
PRINT	PT	Plots a drawing to a plotter, printer, or file
PROEDIT	PED	Changes the color, layer, linetype, and thickness of an object
PROPLAYER (application)	PP	Moves objects to selected layer
PSETUP	PSTP	Sets up print devices
PULLDOWN	MU or PN	Loads a menu file
PURGE	PU or PRG	Removes unused named objects, such as blocks or layers, from the database
QARRAY	QR	Command line version of ARRAY command
QATTDEF	QAD	Command line version of ATTDEF command
QATTEDIT	ATE or QAE	Changes attribute information independent of its block definition (Command line version)
QBMPOUT	QB	Command line version of BMPOUT command
QFONT	QF	Command line version of FONT command
QHATCH	QH	Command line version of HATCH command
QHPEDIT	QHE	Command line version of HPEDIT command
QINSERT	QI	Command line version of INSERT command
QLAYER	QL	Command line version of LAYER command

QMTEXT	QM	Command line version of MTEXT
QOPEN	QO	Command line version of OPEN command
QPRINT	QP	Command line version of PRINT command
QPROPEdit	QPR	Command line version of PROPEdit command
QSAVE	QS	Saves the current drawing
QSLDOUT	QSL	Command line version of SLDOUT command
QTEXT	QT	Command line version of TEXT command
QVIEW	QV	Command line version of VIEW command
QVIEWPORT	QVP	Command line version of VIEWPORT command
QWMFOUT	QWO	Command line version of WMFOUT command
QWOPEN	QW	Command line version of WOPEN command
QXLINK	QX	Command line version of XLINK command
RBORDER	RB	Controls whether the image frame is displayed on the screen or hidden from view
RCONFIG	RPR or RC	Sets rendering preferences
RECTANGLE, RECTANG	RT	Draws a rectangular polyline
REDEFINE (Application)	RED	Restores AutoCAD internal commands overridden by UNDEFINE
REDO	RD	Reverses the effects of the previous UNDO or U command
REDRAW	R	Refreshes the display of the current viewport
REDRAWALL	RDA	Refreshes the display in all viewports
REGEN	RE or RG	Regenerates the drawing and refreshes the current viewport
REGENALL	REA or RGA	Regenerates the drawing and refreshes all viewports
REJOIN (application)	RJ	Creates single entity from co-linear objects (lines and arcs)
RENAME	REN or RNM	Changes the names of objects
RENDER	SHA or RR	Displays a flat-shaded image of the drawing in the current viewport
REVSURF	RS	Creates a rotated surface about a selected axis
RING	DO or RI	Draws filled circles and rings
RLOAD	RL	Inserts images in many formats into an AutoCAD drawing file
ROTATE	RO	Moves objects about a base point
RTEXT (Application)	RX	Writes new text with same properties as a selected text entity
RULESURF	RSF	Creates a ruled surface between two curves
RUNMACRO	RMAC	Calls and executes a macro
SAVE	SAV	Saves the drawing under the current file name or a specified name
SAVEALL	SAL	Saves all open drawings
SAVEAS	SA	Saves an unnamed drawing with a file name or renames the current drawing
SCALE	SC	Enlarges or reduces selected objects equally in the X, Y, and Z directions
_SCRIPT	MAC or SCR	Executes a sequence of commands from a script
SELECT	SL	Places selected objects in the Previous selection set

SENDALL	SDA	Sends all drawings via Microsoft Exchange
SENDFLX	SD	Sends the current drawing via Microsoft Exchange
'SETCOLOR	STC	Sets the color for new objects
'SETLAYER	STL	Change current layer to selection
SETLIB (Application)	SLB	Displays current part library dialog
SETLINETYPE	SLT	Sets linetypes
SETTINGS	OP, RM, PR or ST	Customizes the program settings
SETUP (Application)	US	Sets up drawing units
'SETVAR	SET or SV	Lists or changes the values of system variables
SETVIEWDIR or 3DVIEW	STV	Sets the viewing direction for a three-dimensional visualization of the drawing
SKETCH	SK	Traces cursor movement to generate polylines.
SLDOUT	SLD	Creates a slide file of the current viewport
SOLID	SO	Creates solid-filled polygons
SPLINE (Application)	SP or SPL	Creates quadratic or cubic spline (NURBS) curve
SPLINE;D	SPD or SPE	Edits a splined polyline
STRETCH	S or SR	Moves or stretches objects
_STYLE	STY	Loads text styles from the specified font file and customize it to the individual requirement.
TABCONFIG	TC	Configure tablet menu areas
TABLES (Application)	TBL	Documents and controls database information and drawing parameters
TABLET	TA	Calibrates, configures, and turns on and off an attached digitizing tablet
TABSECTION	SEC	Replaces a section of the tablet menu
TABSERVICE	TB	Changes the names of named objects
TABSURF	TF	Creates a tabulated surface from a path curve and a direction vector
TCORRECT	TCOR	Corrects text entities and attribute definitions
TEMPLATE	TP	Defines drawing file to be used as example in initial drawing setup
TEXT	TX	Creates a single line of text
`TEXTSCR	TN	Opens the text window
TEXTEDIT	ED or TE	Edits text and attribute definitions
'TGRID	TG	Toggles grid display on/off
TICON	TI	Controls the visibility and placement of the UCS icon
'TLMODE	MS, PS or TM	Switches from a layout tab to the Model tab and makes it current
'TORTHO	TORT	Constrains cursor movement
TRAPEZOID (Application)	TZ	Creates a 3D trapezoid
TRIM	TR	Trims objects at a cutting edge defined by other objects
'TSCREEN	TN	Opens the text window
'TSCREENFULL	TSC	Toggles command line window on/off (full screen)
TSNAP	SN or TS	Restricts cursor movement to specified intervals. Toggles snap modes on/off
UCS	UC	Manages coordinate systems

UCS_C	UCC	Manages defined user coordinate systems
UNDEFINE (Application)	UF	Allows an application-defined command to override an internal AutoCAD command
UNDELETE	UD	Restores erased objects
UNDO	U	Reverses the effect of commands
UNITSETUP	UN or US	Sets coordinate and angle display formats and precision
VIEW	V	Creates and restores views
VIEWPAR	VR	Selects view parameters
VIEWPORT	MV or VP	Creates floating viewports and turns on existing floating viewports
<u>VIEWPORT</u> option LAYER	VPL	Sets layer visibility within viewports
VNOTE	VN	Add an audio or text comment into your drawing
VSLIDE	VS	Displays a raster image slide file in the current viewport
VTXEDIT	VTX or VX	Edits polyline vertices
WCASCADE	WD	Arrange windows in cascade format
WCLOSE	WC	Closes the current viewport
WICONARR	WI	Arranges all drawing windows
WMFOUT	WM	Saves objects to a Windows metafile
WOPEN	WO	Opens new viewport
WTILE	WT	Arranges Windows in a tiled format
WTILEHOR	WH	Arranges Windows in a horizontally tiled format
WTILEVER	WV	Arranges Windows in a vertically tiled format
XINSERT	XI	Attaches an external reference (Xref)
XLINK	XL	Controls external references to drawing files
XPLODE (Application)	X	Breaks up complex objects into component entities
XPLORER	XP	Accesses and allows alteration of the current drawings properties
'ZOOM	Z	Increase or decreases the apparent size of objects in the current viewport
'ZOOMBACK	ZX	Zoom to previous view
'ZOOMFAC	ZF	Zoom by a factor
'ZOOMFIT	ZE	Zoom to extents
'ZOOMIN	ZN	Zoom into the drawing
'ZOOMOUT	ZT	Zoom out from the drawing
ZOOMPAGE	ZP	Zoom to drawing page limits
'ZOOMWIN	ZW	Zoom to a window

## Alias Commands

### Alias Names for Commands

As in AutoCAD, you can define aliases for command names. In most cases, these are one, two, or three characters, such as L for Line and C for Copy. A full set of command aliases is available for use and can be simply modified to suit your own needs and preferences. In addition, all supported AutoCAD 2000

commands have been mapped to their corresponding AllenCad commands. This enables you to enter familiar AutoCAD commands at the command line when running AllenCad. Please refer to the Command Chart to see the listing of alias commands that come pre-configured in the program.

Alias commands, as well as hot key assignments are all stored in a .key file by the Config – Alias Commands and Hotkeys options. Different .KEY commands can be specified, as required, after designating them in the CONFIG - Directories option.

In AutoCAD, aliases are limited to executing commands; they cannot handle options. In contrast, AllenCad allows aliases to execute commands and options, such as ZE for Zoom Extents.

Another use for command aliasing is to assign familiar AutoCAD command names to corresponding AllenCad commands. For example, you could assign PLine to the Polyline command, and Solid for the 2DFace command.

The CONFIG command displays Configuration dialog box; the Alias Commands tab lets you create, modify, and delete aliases for command names.

There is a difference in where and how AutoCAD and AllenCad store the alias names of commands. Users familiar with AutoCAD will be aware of the ACAD.PGP file that contains aliases; AllenCad stores its aliases in a .key file of the user's individual configuration directory, typically the Cfg folder. The .key file contains a section named [ALIAS COMMANDS] where the alias names are defined using the following format:

<Alias>=<Command name or command macro>

For example, here are the several lines from the PCAD5.Key file:

[ALIAS COMMANDS]

3V=3DVIEW

A=ARC

BH =BHATCH

Etc.

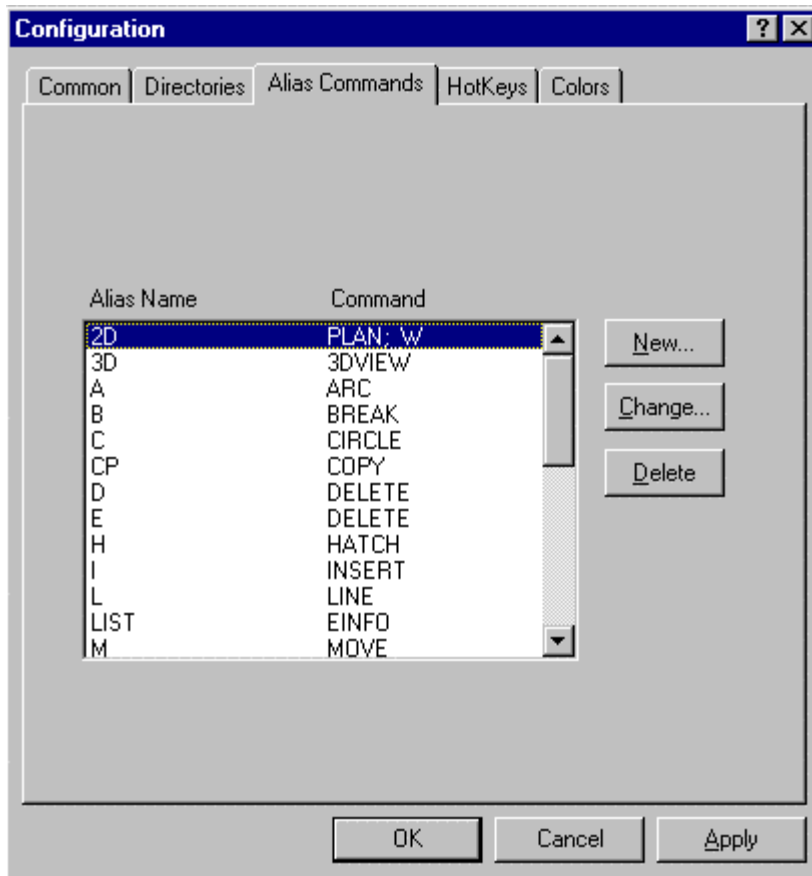


Figure 2.2: The dialog box for creating and editing command aliases.

Alias-commands can be defined freely so that  
`LINE 0,0 210,0 210,297 0,297 _C`

could be replaced with the alias command A4. Using this method it would no longer be necessary to input the parameters to draw an oblong to represent an A4 sheet of paper, the same result could be achieved by simply typing A4.

This method of operation not only increases the speed and efficiency with that the command is carried out, but also greatly reduces the risk of input errors.

### Notes to Application Developers

AllenCad uses alias names in commands defined by LISP and FDT.

Unlike AutoCAD, a AllenCad alias can consist of a macro, such as `Zoom;E`.

Accelerator key definitions may contain the transparent command prefix ( `'` ) and the Esc characters ( `^^` ); command aliases, however, ignore these special characters.

## Configuration: Shortcuts and Alias Commands

The function key allocation can be used as a very simple and flexible method of personalizing the program to suit individual requirements and work styles.

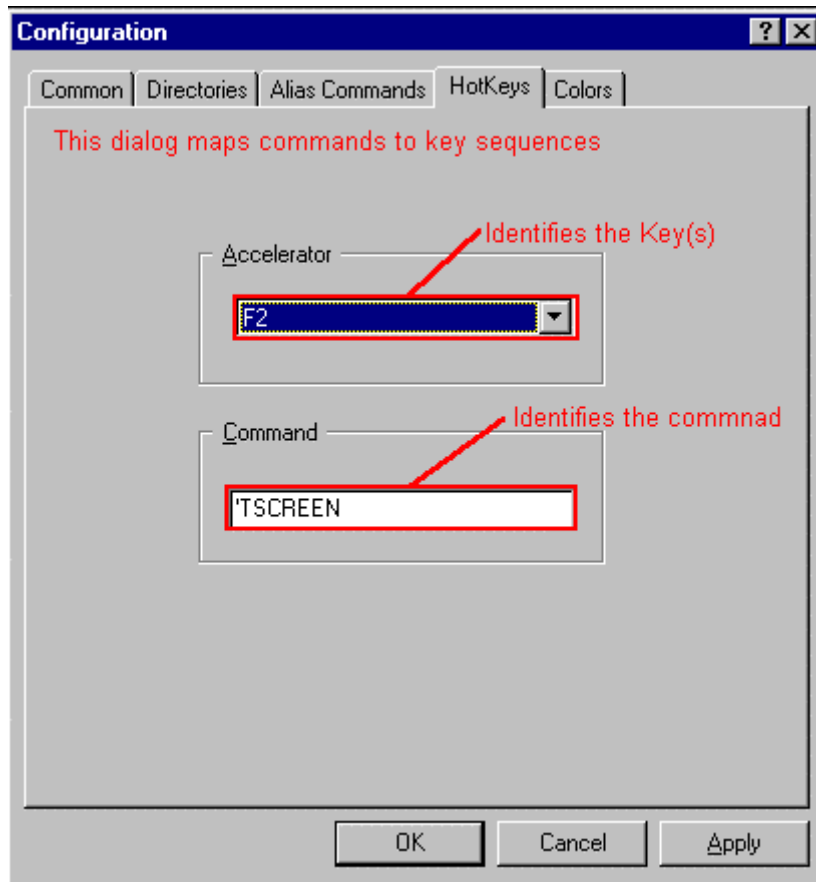
Using either the individual F1 ... F 12 function keys, or combining these with the CTRL and ALT keys, it is possible to automate up to 48 functions and commands that then become available at the "touch of a button". Most of the 48 alternatives can be freely defined.



The F1 key is allocated to the help function and reserved.

The CTRL+F4 and ALT+F4 key combinations will close the program.

By allocating regularly used commands, such as drawing or editor instructions, object capture, or object selection functions to these keys it is not only possible to simplify your construction work, but also to dramatically increase speed and efficiency.



From the CONFIG command's *Hot Keys* dialog, select the function key, or key combination to which you want to allocate a command from the drop-down list. If this key has already been allocated, the corresponding command will appear in the command display box. If the display remains empty, then no command has yet been allocated to this key combination.

You allocate a new command to the selected key by moving the cursor into the Input field and type in your desired command or function. Should you wish to allocate a function or command to a key that already contains an entry, just overwrite the entry with the desired command.

Please note that the parameter changes only become effective once the OK button has been selected. No changes will be made if you close the dialog by either clicking to the Cancel button or pressing the ESC key.

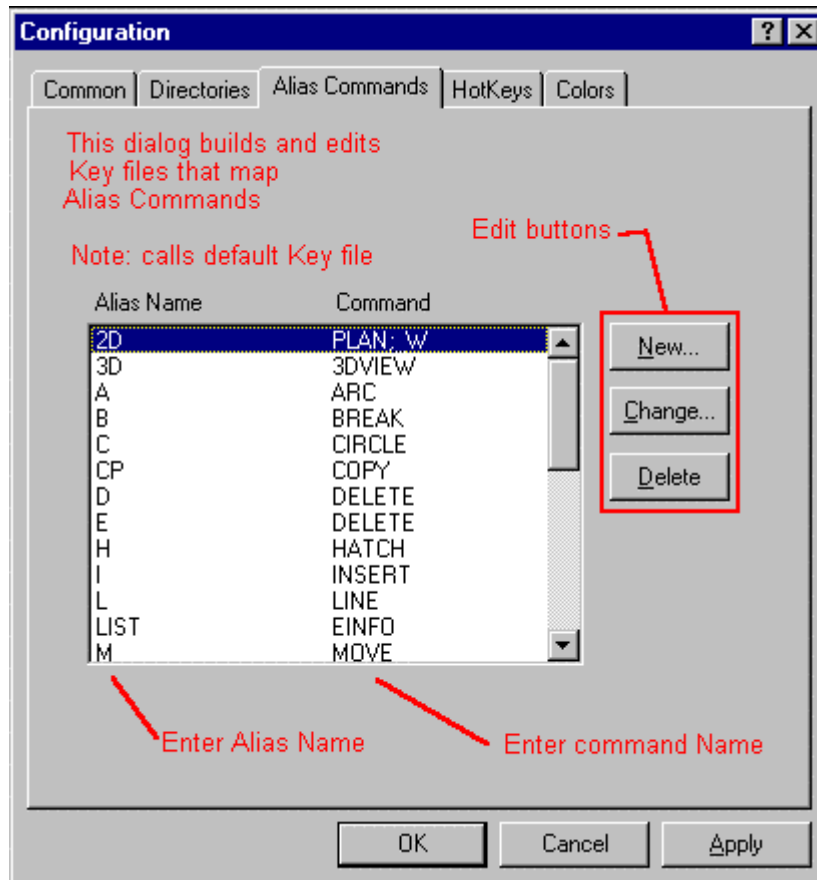
### Alias Commands

As is the case with function keys, Alias Commands also offer a very simple and effective way of customizing the program to meet the individual needs of the user. Alias-commands can contain command entries that are not only made up of a command itself, but also corresponding command arguments, such as parameters. Alias-commands can be defined freely so that

```
LINE 0,0 210,0 210,276 0,276_C
```

could be replaced with the alias command LET. Using this method it would no longer be necessary to input the parameters to draw a rectangle to represent an 8 ½" x 11" sheet of paper, the same result could be achieved by simply typing the alias LET.

This method of operation not only increases the speed and efficiency with that the command is carried out, but also greatly reduces the risk of input errors.



Alias-commands can be defined by selecting the Alias-command option from the configuration window. Further information on Alias Commands and Hotkey assignments can be found in Chapter 3 of the Customization and Programmer's Guide.

Dialog Box Control Elements:

Alias Commands List

New

Change

Delete

## Define a new Alias Command

By selecting *New...* it is possible to define a new Alias-Command. Enter the name and command line, including parameter arguments if necessary.

## Alias Commands List

All previously defined Alias-commands are displayed in the selection field. This includes a default selection of "two-finger" command aliases, as well as mappings to directly supported AutoCAD command names.

## Change Alias Command

In order to change an already defined alias command, select an item from the selection field and click the *Change...* button. In the dialog box *Change Alias Command* enter a new alias name and the built-in command, including parameter arguments if necessary, into the input window.

## Delete an Alias Command

In order to *Delete* an Alias-command select it from the list box and click the delete button. It is important to note that the command will be deleted immediately you will not be requested to re-confirm the deletion.

## Transparent Alias Commands

While a command is active, transparent alias commands can be issued by immediately prefacing the alias with an apostrophe, in a similar fashion to non-transparent alias commands.

See the example of the transparent alias command 'Z below:

Definition in the PCAD5.key file:

Z=ZOOM

> LINE

From point:

To point:

To point: 'z

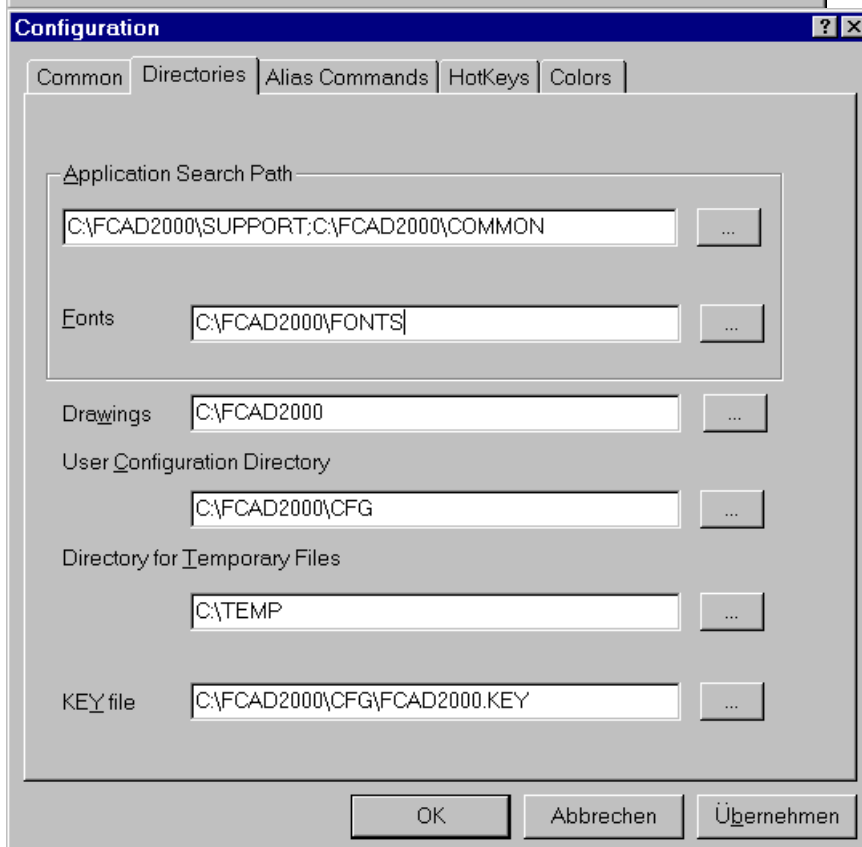
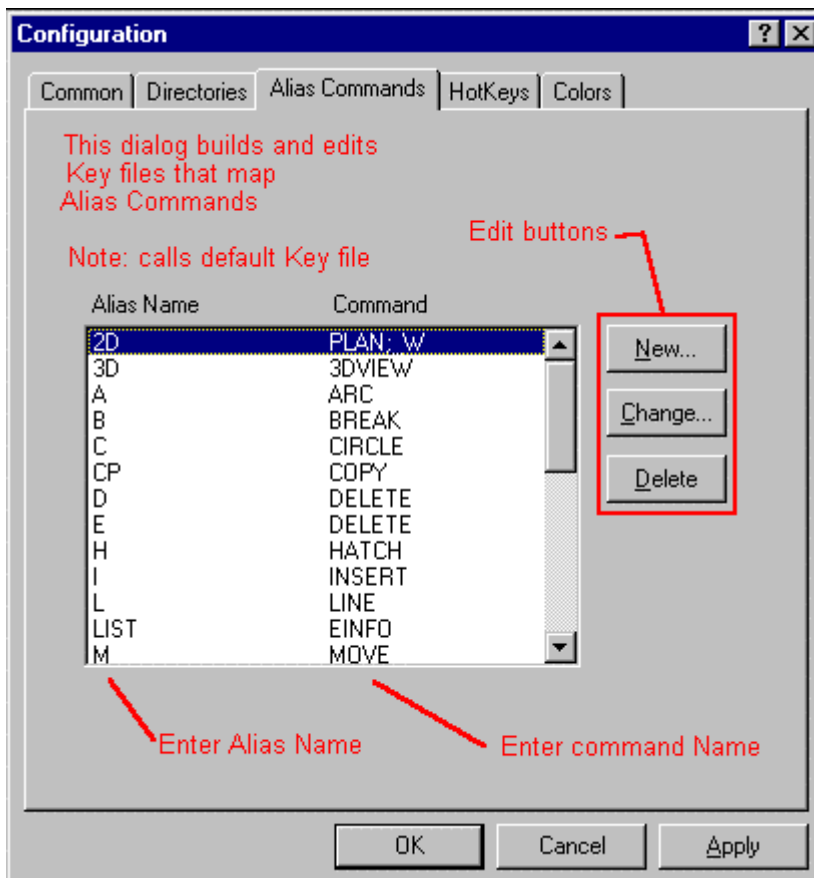
Zoom scale factor: Window

First corner:

Second corner:

To point:

Note: Alias Commands are defined under the Alias Commands tab on the CONFIG dialog.



# "2"

## Filled Faces



**Command: 2DFACE**

**Alias 2F**

**Located:**

1. **Pull-down Menu: Draw – Filled Faces**
2. **Or Enter Command or Command Alias Directly at the Command Line**

**2D Faces** make it possible to draw solid filled planes that have straight-line borders. This is done by creating and connecting any number of three or four-cornered planes. It may be done continuously or using segments. It is also possible to connect 2D planes to already existing 2D faces.

2D Faces are always drawn by determining the corner points (three or four). The option **Close** is available to draw triangular planes so that the third entered point is connected with the starting point of the plane.

### Indication

Remember, when defining the end points of the edges that the order in which these points are entered influences the resulting shape.

The user should always specify the end points of the edges in the same direction. If the end points of the edges of a plane are determined clockwise or counterclockwise, a "butterfly" effect results.

### Options

#### Continuous Filled Faces

"Continuous" is the standard option for drawing filled planes; it will be used automatically if no other option is chosen.

When drawing rectangular planes, determine the end points of the edges. Remember that the first and third and the second and fourth point are connected. The edges will make up the plane.

After determining the end points of the first two edges, the user may enter additional points, and they will be used as third and fourth edge (and so on) points for another plane immediately connected to the first one. To terminate the command, press RETURN.

#### Close

To draw a triangular plane, enter three end points for two of the edges successively, and choose the option **Close** from the options panel or enter the command **Close** manually.

#### Filled Face Segments

The option **Segments** is used to draw a single rectangular or triangular filled plane. In contrast to the option **Continuous**, this option will end after the entry of the points required for the plane.

#### Appending Filled Faces

This option makes it possible to append newly created planes to already existing ones. It is only possible to append a plane to the edge last drawn.

After entering the option **Append**, choose a side from an existing plane to which the new plane is to be appended. Determine the corner points of appending planes.

> 2DFACE

Choose a side: **Choice**

Third point: *P5*

Fourth point: *P6*

...

Terminate the command by pressing RETURN.

**"3"**

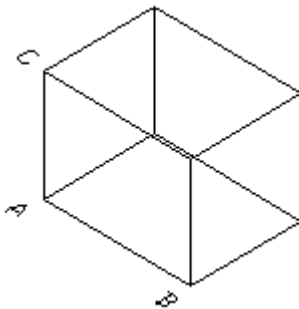
## Command 3DBOX

**Command:** 3DBOX

**Alias** 3B

**Located:**

1. **Pull-down Menu: Draw – 3D – Box**
2. **Or Enter Command or Alias at the Command Line**



The command 3DBOX allows simple creation of a 3 dimensional box. The command can also be accessed from the Draw - 3D pull-down menu.

Upon executing the command, you are asked to input the first corner ("A" in the diagram). As with the input of all location definition, you may enter coordinate values in the form x,y,z or you may pick an existing reference point in your drawing. Unless the point you want has a z or height/depth value of zero (default value for 2D objects), be sure that the point you pick has the correct z value.

Once the first corner has been established you will be presented with an option dialog offering you the opportunity to input a length to establish the second corner. The z value (height) of the second point is the same as the first and the first side is parallel to the x-axis (Y values of A and B are equal).

You will be prompted for width if you chose the length option and presented with a dialog offering Cube and Square options.

Selecting **cube** indicates that the x, y and z dimensions are equal to A-B and the cube is completed. Choosing **square** indicates that the x and y values (only) are equal to A-B and the square is drawn with a following prompt for height input, that completes the box.

The second point (picked point "C") would be regarded as the diagonally opposite corner of a rectangle and be drawn, if you were to ignore the Length option and input a point with a different y value, after setting the first corner. A prompt for height would follow.

For height, an absolute value can be input. Picking a point "D", however, will set height as being the 3 dimensional distance between points "A" and "D".

## Command 3DCONE

**Command:** 3DCONE

**Alias** 3DC

### **Located:**

1. **Pull-down Menu: Draw – 3D – Cylinder / Cone**
2. **Or Enter Command or Alias at the Command Line**



The command 3DCONE allows creation of a 3D cone, cylinder, prism, or pyramid. This command can also be executed from the Draw - 3D pull-down menu. Object creation is determined by user input. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

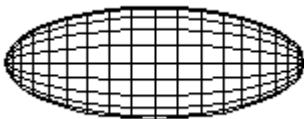
## Command 3DELLIPS

**Command:** 3DELLIPS

**Alias** 3L

### **Located:**

1. **Pull-down Menu: Draw – 3D – Ellipsoid**
2. **Or Enter Command or Alias at the Command Line**



The command 3DELLIPS (Draw>3D>Ellipsoid) allows creation of a 3D ellipsoid object. This command may also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

## 3D Faces



## Command: 3DFACE

Alias 3F

### Located:

1. **Pull-down Menu: Draw – 3D – 3D Faces**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

The **3D Faces** drawing function creates three or four sided surfaces "in space", or, in other words, in an X, Y, Z coordinate system. The difference between 2D and 3D surfaces is that the 3D surfaces are not shown as solid areas, but are drawn as wire frame models.

The 3D Faces definition points are fixed in either a clockwise or counter-clockwise direction so that one edge of a 3D surface is used as the base line for the next 3D surface.

The user may also draw "invisible" lines, a function that may increase the clarity of complicated 3D structures.

### Options

#### Continuous

The option **Continuous** is the default option when drawing 3D surfaces.

To draw a continuous 3D surface, define the corner points in a clockwise or counter-clockwise direction.

For a triangular surface, define all three corner coordinates and then end the drawing with the **Close** option located. The last defined coordinate will then be joined to the first coordinate location, forming a triangle, and the drawing will be completed.

```
> 3DFACE
First point: P1
Next point: P2
Next point: P3
Next point: Close
```

For a rectangular 3D surface, define the four corner coordinates. After having completing the fourth entry, the last coordinate will be joined to the origin, thus creating a four-sided object.

This process is then continued. The user will be asked to define further points or coordinates. The next 3D surface to be defined will use the last drawn line as the starting point or base line for the next 3D surface.

End the 3D surface drawing process by pressing the RETURN key.

```
> 3DFACE
First point: P1
Second point: P2
Third point: P3
Fourth point: P4
Third point: P5
Fourth point: P6
Third point: P7
Fourth point: P8
... and so on.
```

#### 3D-Face Segments

To draw a single 3D Face, use the **Segment** option. The method is almost identical to that just described.



> 3DFACE  
First point: P1  
Second point: P2  
Third point: P3  
Fourth point: P4

In contrast to the **Continuous** option, the drawing process will end with the definition of the fourth point and the consequent closing of the contour. No further surfaces will be "connected".

This option also allows the use of the **Close** function, thus enabling triangular shaped 3D surfaces to be created.

Should the user wish to add further 3D surfaces to an existing one, this can be achieved at any time by utilizing the **Append** option.

### Append

This option allows the user to connect new 3D surfaces to existing objects. First select the edge that is to act as the base line for the connection, then proceed by defining the required corner points for the new 3D surface.

The **Append** option will automatically change the mode to **Continuous**. This means that after having connected one 3D surface the user can define additional pairs of coordinates to create more surfaces.

### Invisible Edges

The **Invisible** option is used to draw hidden lines in a 3D-Face. This feature is particularly useful when dealing with complex drawings containing a large number of 3D surfaces. The use of hidden lines will make the drawing more understandable.

This function is available for all modes used in 3D surface drawings. The function must be activated prior to the definition of the starting coordinate of the invisible line.

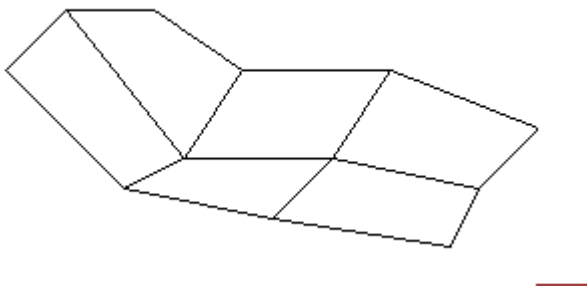
## 3D Mesh

**Command:** 3DMESH

**Alias** 3M

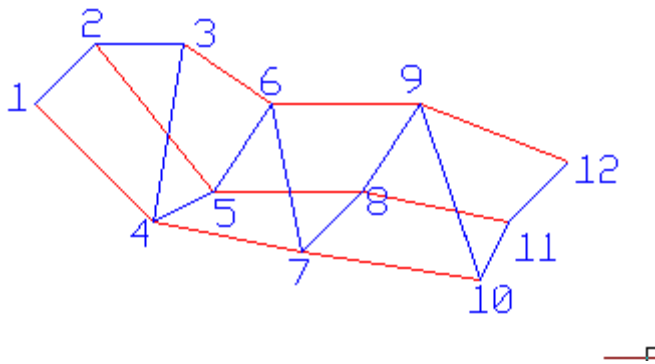
**Located:**

1. **Pull-down Menu: Draw – 3D – 3D Mesh**
2. **Or Enter Command or Alias Directly at the Command Line**



The 3DMESH command creates polygon meshes. Usually it is used to generate irregular surfaces, vertex by vertex. For example, it is used in surveying and GIS applications to create topological surfaces.

A 3D mesh is a wireframe rectilinear blanket composed of  $M$  column lines by  $N$  row lines passing through a matrix of  $M \times N$  three-dimensional points in space.  $M$  and  $N$  are indices specifying the number of rows and columns that make up the mesh, and determine the number of vertices required in the mesh.



### To Create a Polygon Mesh

First you are prompted to specify the mesh M size and the Mesh N size. This is simply the number of wire intersections wide and long.

Then the program continues to ask for corners or vertices, until you have defined all of the vertices in the one (M) direction and the other (N) direction. Remember that to complete the mesh, you will need to enter M x N vertices. You are prompted for each vertex in order, starting with vertex (0,0), continued with the second vertex in the first column (0,1), and so on. Once all vertices of a column are specified, the first vertex of the next column is requested. The blue line traces the input order in the example further below.

1. During vertex input, the mesh does not appear onscreen but is instead being constructed in memory. Terminating the command places the mesh onscreen
2. Meshes can be open or closed depending on whether the mesh joins in either the *M* or *N* direction, or both. A donut mesh would be an example of a mesh closed in both directions.

### Example

```
> 3DMESH
M-value of the mesh (between 2 and 256): 4
N-value of the mesh (between 2 and 256): 3
Vertex (0,0): 110,170,4
Vertex (0,1): 130,190,4
Vertex (0,2): 160,190,6
Vertex (1,0): 150,130,0
Vertex (1,1): 170,140,0
Vertex (1,2): 190,170,2
Vertex (2,0): 200,120,0
Vertex (2,1): 220,140,-2
Vertex (2,2): 240,170,0
Vertex (3,0): 260,110,0
Vertex (3,1): 270,130,0
Vertex (3,2): 290,150,0
```

## Command 3DMIR

**Command: 3DMIR**

**Alias 3MR**

**Located:**

1. **Pull-down Menu: Modify – 3D Mirror**
2. **Or Enter Command or Alias at the Command Line**

This command creates a mirror image in a 3-dimensional plane around a defined 3 dimensional axis. Essentially, what is originally visible in this 3-D Plane is copied in a reverse image on the other side of the axis at the end of the command, much as with the 2D MIRROR command.

The input sequence for this command is as follows:

1. First, you are asked to select the objects for mirroring
  2. You are then asked to define the mirror plane by picking three points or entering their coordinates
  3. Next, you define the mirror axis by picking two points or entering their coordinates
  4. With this information, the command completes its operation.
- IE.  
> 3DMIR  
Select objects:  
1 selected.  
Select objects:  
\*\*\* 1 selected. \*\*\*  
First point of the mirror plane <0,0,0>:0,0,0  
Second point of the mirror plane <0,0,0>:0,5,10  
Third point of the mirror plane <0,0,0>:0,20,10  
Select objects: Echoed only  
Select objects: Echoed only  
\*\*\* 1 selected. \*\*\*  
First point of mirror axis: No prompt - bypassed  
Second point: No prompt - bypassed

## Command 3DROT

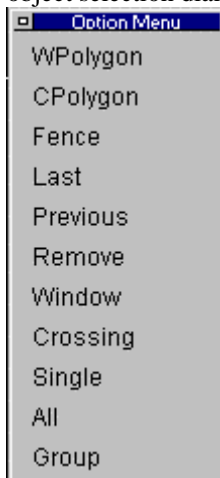
**Command: 3DROT**

**Alias 3R**

**Located:**

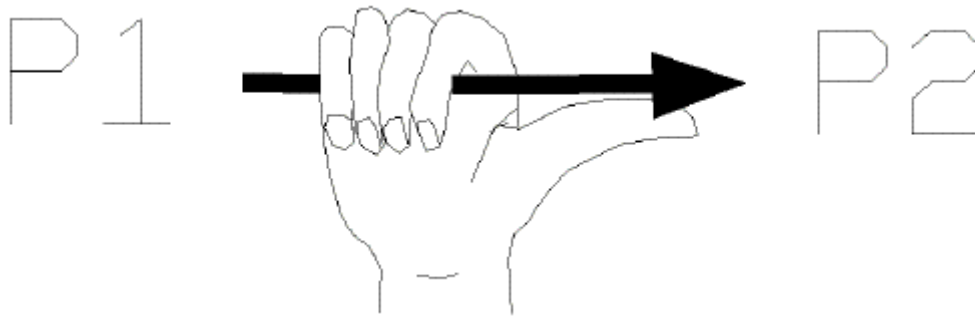
1. **Pull-down Menu: Modify – 3D Rotate**
2. **Or Enter Command or Alias at the Command Line**

When you issue the command, you are asked to select the objects for rotation and presented with the object selection dialog.



Next, you are asked to define the axis of rotation, at which point you may enter the 3 dimensional point coordinates for P1 (x1,y1,z1) and P2 (x2,y2,z2) or pick points on the drawing editor. You are then asked

to define an angle of rotation, where a positive rotation is a rotation counter-clockwise around the axis and a negative angle is clockwise. If you find this confusing, don't feel too badly. Fortunately for us all there is a "**Right-Hand Rule**" that helps to explain things.



In this rule, the axis of rotation is shown as the arrow gripped by the right hand, where the direction thumb points from P1 to to P2 in the direction of the thumb. Rotation, from the start or existing point, is indicated by the fingers on the hand (counter-clockwise when viewed along the axis from the P2 position).

The input sequence is as follows:

> 3DROT

Select objects: {PICK}

\*\*\* 6 selected. \*\*\* [ a crossing selection was used in this instance]

First point of rotational axis <0,0,0>: x1,y1,z1 [IE P1 coordinates]

Second point of the rotational axis <0,0,1>:x2,y2,z2 [IE P2 coordinates]

Rotation angle: 45 [A positive angle of 45o is selected]

The view then changes according to the input parameters.

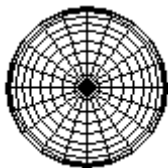
## Command 3DSPHER

**Command: 3DSPHER**

**Alias 3DS**

**Located:**

1. **Pull-down Menu: Draw – 3D – Sphere**
2. **Or Enter Command or Alias at the Command Line**



The command 3DSPHER creates a 3D sphere (ball). This command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

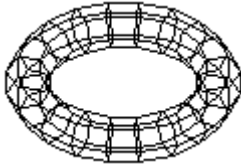
## Command 3DTORUS

**Command: 3DTORUS**

**Alias 3T**

### **Located:**

1. **Pull-down Menu: Draw – 3D – Torus**
2. **Or Enter Command or Alias at the Command Line**



The command 3DTORUS allows creation of a 3D torus. The command can also be entered from the Draw - 3D pull-down menu. Smoothing, or "roundness", is entered as a unit number. A lower number will create less 3D object faces, resulting in less smoothing, and a quickly generated result. A higher number (e.g. 200) will create more 3D faces and better smoothing. Using a higher number takes longer to compute.

### **3D View Control...**



**Command: 3DVIEW**

**Alias 3V**

### **Located:**

1. **Pull-down Menu: View – 3D View**
2. **Left SideBar**
3. **Palette: View**
4. **Or Enter Command or Alias at the Command Line**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define. The command 3DVIEW from the View menu allows you to define the view parameters. .



The program uses as its standard viewing angle, the top view (also called Plan View). At this angle the x-y-plane is viewed from the direction of the positive z-axis, and the drawing or construction plane is parallel to the screen.

Additionally, FRONT, BACK, LEFT, RIGHT, and BOTTOM are possible standard views.

Along with the ability to view the drawing from any vantage point, it is also possible to rotate the view in the X – Y plane. You may want to do this, for instance, when orienting your drawing for printing.

---

**See also:**

SETVIEWDIR command

View Manager: Saving / Retrieving Views

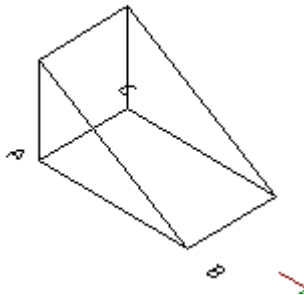
Command 3DWEDGE

**Command: 3DWEDGE**

**Alias 3W**

**Located:**

1. **Pull-down Menu: Draw – 3D – Wedge**
2. **Or Enter Command or Alias at the Command Line**



The command 3DWEDGE allows creation of a 3 dimensional wedge. This command can also be entered from the Draw - 3D pull-down menu.

## "A"

### About

**Command: About**

**Alias AB**

**Located:**

1. **Pull-down Menu: Help - About**
2. **Or Enter Command or Alias at the Command Line**

The ABOUT box reports

1. Copyright
2. Program Version
3. User Name
4. Serial Number
5. Physical Memory
6. Disk Space

### Command APPLOAD

**Command: APPLOAD**

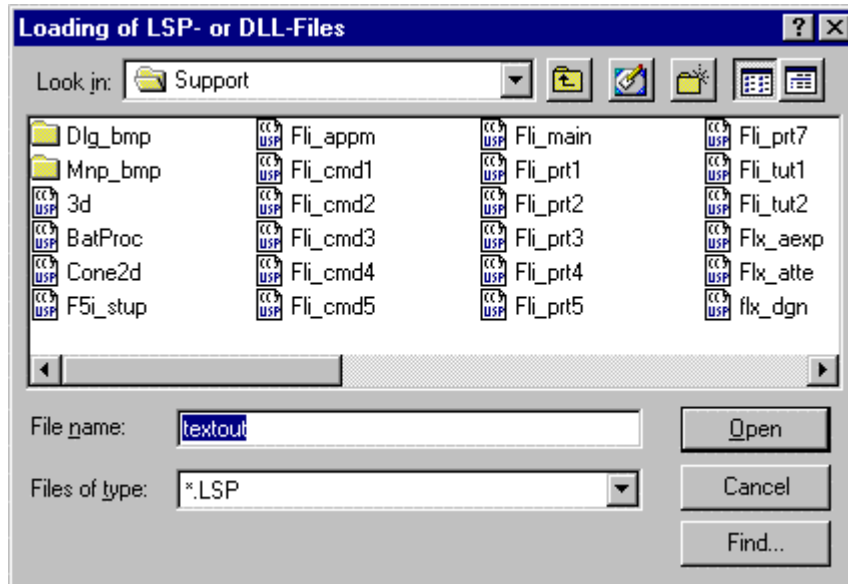
## Alias AP

### Located:

1. Pull-down Menu: Tools – Resource Manager – Load Application
2. Or Enter Command or Alias at the Command Line

The **APPLOAD** command is used to locate and load either LISP programs or .DLL (Dynamic Link Library) files for use in the program. The command is issued from either the "Resource Manager - Load Application" selection on the Tools pull-down menu, or else it can be entered at the command line.

The following dialog lets you choose between LISP and DLL file selection and offers an opportunity to browse and select your file from the dialog display.



This command performs very much the same function as the LOADLISP command, except that it allows you to load LISP and DLL Files, as you may require.

**Note:** Be sure to check the command line as the LISP file is loaded. There you will see the command name to enter whenever you want to run the routine during the current edit session. The example below indicates the command name to use when the "Textout.lsp" file is loaded.

```
Loading D:\FCAD2000\SUPPORT\TEXTOUT.LSP...Command TEXTOUT loaded.  
Done.nil  
> |
```

A DLL when loaded, however, only gives a confirming message that the load was successful:

```
> APPLOAD  
Loading D:\fcad2000\SUPPORT\Rejoin.dll .....OK.  
>
```

## Arc



## Command: ARC

Alias A

### Located:

1. **Pull-down Menu: Draw - Arc**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

There are several ways, geometrically, to describe an arc, just as with a circle. The program allows the user to choose from among these methods, and draw the arc in the most convenient manner.

The following options for drawing an arc are available:

1. Three Points (starting point, second point, end point);
2. Two points and radius;
3. Two points and angle of opening;
4. Two points and center point;
5. Two points and tangents through starting point;
6. Center point, starting point and angle; or,
7. Center point and two angles.

Variations of the options of this command may combine some of the basic geometry inherent to each method.

**All arcs will be measured to the left, starting from the positive x-axis.** In other words, you pick points on the arc in an anti-clockwise direction. It is important to recognize this directional convention, as failing to do so will produce an arc that is complementary to the one intended (the one that if added to the intended arc, would form a circle). To "pick" points for the construction of arcs, the user should always use OSNAP options.

### Options:

The options are displayed in the *Options Panel* when the command is active.

#### 3Point

Since the method most used for drawing arcs is to select three points on the arc, that is the option that has been chosen as the "standard", or default, option. It will be used if no other option is chosen.

The arc to be drawn is defined by three points. At the prompt, enter the starting point, a second point on the arc's circumference and the end point of the arc. Remember the directional convention; the arc is described anti-clockwise to the left starting with the positive X-axis.

Starting point: P1  
Second point: P2  
End point: P3

This option is also available in other options of the drawing command ARC.

#### SEnd (Starting point - End point - Radius/Angle)

Using the option **SEnd**, the user may create arcs with only the starting point and end point defined. The "arc" of the element is defined by entering or "picking" a third point on the circumference, or by specifying an angle.

The basic variation of this option simply follows the prompts, entering the starting point and end point and the radius of the arc by "picking" or by entering through the keyboard.



Starting point: P1  
End point: P2  
Radius: Value

When the user is "picking" the radius of the arc, it must be remembered that the distance of the center point to the **x-axis** of the end point of the arc is important, not its distance to the y-axis.

It is also possible to enter a radius, an angle or a direction from the starting point instead of the starting point and the end point. To do that, choose the option **Angle** or **Starting point**.

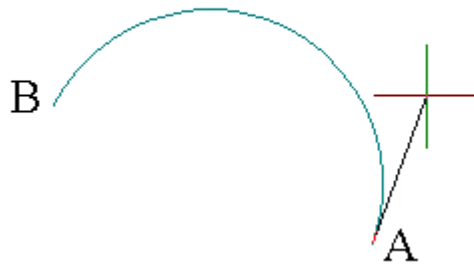
### Option Angle

With this option the arc is defined by the included angle of the arc (angle between the starting point, the center point of the arc and its end point), the starting point, and the end point. The prompts will request a value with the words **Included angle**:

Enter that value either through the keyboard, or by "picking" a value. When picking a value, remember that the x-axis is the point of reference, not the last point entered.

### Direction

The arc is defined by a tangent that touches the arc at its starting point, next to its starting point and at its end point.



In this example, we have defined the start and end points of the arc (A and B) and have selected the direction option. This results in a line being represented from the arc's start point to the current cursor position. The arc radius varies with the cursor location such that the arc remains tangential to that line representation at the start point.

The program will ask the user to pick or enter the coordinates of a point that fixes the cursor location and thereby defines the arc radius.

### SCenter (Starting point - End point - Center point)

This is a variation of the option **SEnd**, in which the arc is drawn by determining its starting point, end point and center point.

Choose the option **Center point** from the options panel after having entered the starting point. Enter the coordinates of the center point manually or "pick" them with the cursor. The arc is defined when the end point is entered.

It is also possible to end the arc by entering an angle manually or "picking" it instead of the end point. The program expects a closed angle and defines it as the angle between the starting point, the center point and the end point of the arc.

### CStart

The option **CStart** draws the arc starting from the center point.

After choosing the option **CStart** from the options panel, define the center point. Then enter the starting

point and the end point of the arc (it is possible to enter a closed angle instead of the end point).

Center point: *P1*  
Starting point: *P2*  
End point: *Angle*

### **CRadius**

**CRadius** also starts an arc by defining the center point. After that, determine the radius as well as the starting angle and the ending angle of the arc to be drawn. A rubberband line that starts in the center of the arc helps the user to determine the points to enter for the angle.

Center point: *P1*  
Radius: **r**  
Starting angle: *P2*  
Ending angle: *P3*

### **Append**

It is possible to append, or connect, arcs to lines or other arcs with the option **Append**. The arc is determined by two points, the starting point being the connection point, and the end point of the new arc.

First choose a line or an arc to which the new arc should be appended. Then determine the end point of the arc to be drawn.

Choose the line or arc: *P1*  
End point: *P2*

**Note:** The "smoothness" with which the arc is represented onscreen is controlled by the variable **CIRCLERES**. By issuing the **SETTINGS** command, users can go to the Draw Modes tab and enter a new value (between 8 and 1024) in the Arc Segments box that controls smoothness. A low value will result on a rough arc or circle, while a higher value will produce a smooth arc or circle.

## Area / Circumference



**Command: AREA**

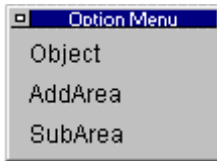
**Alias AR**

### **Located:**

1. **Pull-down Menu: Edit – Area / Circumference**
2. **Left SideBar**
3. **Palette: Edit**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

This function defines the circumference and the area of one or more drawing objects and displays the values. For objects that do not have width, the length instead of the circumference will be displayed. The areas of single objects can be also added or subtracted.

As soon as the command is issued, you are presented with the following option dialog:



These options are described below:

### Using Area / Circumference

To determine the circumference and the area of one or more objects proceed as follows:

Define the drawing object to be examined by tracing its outline or using the **Object** option. The preferred method of selection is by using the cursor and Object Snap functions, but inputting the coordinates of the points is also okay. Pressing RETURN at the request **Next Point:** will finish the selection. It is not necessary to select the first point again to close the boundary, as the program assumes a closure between first and last points. It also does not matter which direction you trace the outline in, so long as you complete the outline definition in either a continuous clockwise or anti-clockwise direction.

### Object: Regular Areas

"Regular" geometric areas (of circles, arcs, closed polylines) can be evaluated by choosing the **Object** option.

### Add or Subtract Areas

Select the first partial area or determine the points for describing the area geometry, as above. The determined values will be displayed in the text window.

Select the option *AddArea* to add a partial area, or *SubArea* to subtract a partial area, from the total area.

The values (circumference /length and area) for the total area will be displayed in the text window, after additions or subtractions are made.

When partial areas are added or subtracted from an area, the total area after the addition or subtraction is regarded as the new (initial) parameter value for subsequent additions or subtractions unless the user explicitly selects a contrary option from the Options Bar.

The Options menu will display only that option that is not active at the moment.

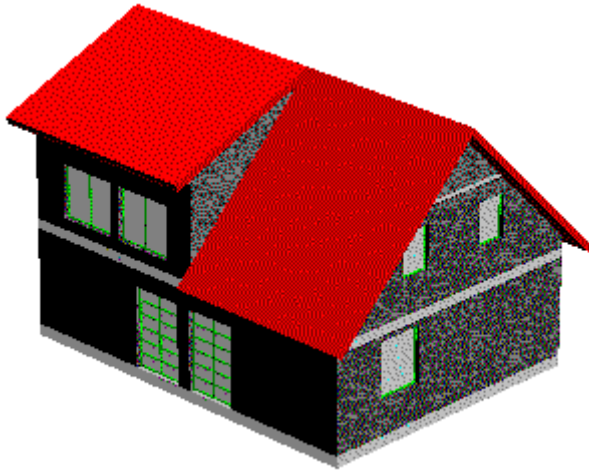
---

### See also:

System variable AREA

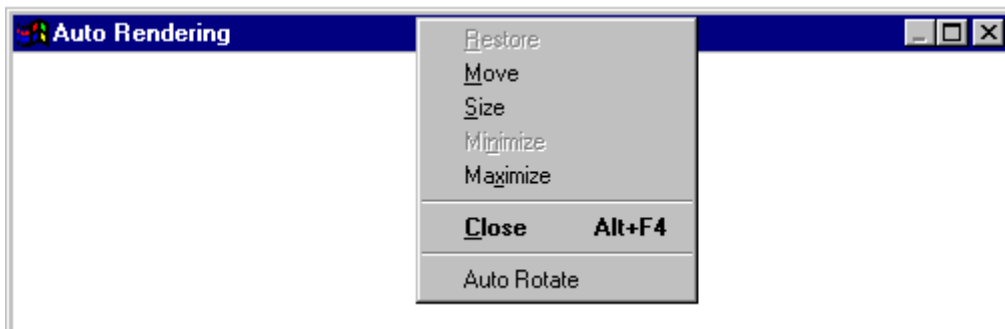
## AutoRender

The Arender command (View > AutoRender) displays a rendering of the current drawing in a separate pop-up window.



You can rotate the rendering by situating your pointer inside the Arender window, and holding down the left mouse button. While the mouse button is depressed, the icon will change to a small hand, allowing you to rotate the image about the screen.

Zooming and panning for Arender is accomplished by holding down the right mouse button. While depressed, you can slide the pointer up or down to zoom and pan the image.



To auto-rotate (spin) the rendering image, right click on the blue command line area and toggle the auto-rotate setting.

For the control of lines and hatches with ARENDER, see ARLINEDISP.

## Array

**Command: ARRAY**

**Alias RA**

### Located:

1. **Pull-down Menu: Modify - Array**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Array** represents a variant of Copy, except that Array will create multiple copies, and they can be arranged at random, or in rows, or in vertical order, or on a circle, or on an arc.

The entities created by the command are exact copies of the selected entities in which all entity attributes such as layers, line types and color are copied.

After entering the command, select an option from the Option Bar:

Individual  
Rectangular  
Circ-NRot  
Circ-Rot

### Options



#### **Individual Array**

This mode allows the user to copy the Array as many times as is required during one command sequence and place them at random on the drawing area.

The use of this option is very similar to the command Copy.

After choosing the option, select the entities to be copied using the selection modes described in the Option Bar. Always confirm object selection by pressing RETURN.

Determine the base point for the copying. A particularly discreet point in the geometry or in one of the selected objects (edges, corners, intersecting points etc.) will be useful. Next, either input the coordinates of the target point(s) or "pick" the points for placement of the Array. The program will create a copy of the Array after each definition of a target point.

As long as the command remains active the user can produce and place as many copies as are required. Terminate the function by pressing ESC or RETURN.



#### **Rectangular Array**

This option will arrange the Array copies in rows or columns. The use of this function is the same as was the use of the **Individual** mode.

Determine the number of the columns and rows in which the Array copies should be arranged. Define the distance between the columns and the rows.

Note that, when planning the arrangement, the rows and columns must be in the positive X or Y-axis. This infers that the original Array is situated in the lower left-hand corner of the quadrant plus-XY. If it is not located there, it may be useful to move the original Array before attempting to begin defining the copying positions. Use the command Move to do this.

### **Circular Arrays**

The Array can also be reproduced in a circular path. The circle (arc) itself is not displayed on the screen. The reproduction of the Array is described as follows:

1. The center of the circle (arc)
2. The number of times the Array is to be reproduced;
3. The included angle of the arc (360 if the path is to be a full circle) to be filled;

There are two ways to reproduce the Array on a circular path:



#### **Placing the Arrays Without Rotation**

This Option, will arrange the reproductions on the circle (arc) in the same alignment to the principal axes as the original Array.

Define the center of the circular path. Then determine **either**;

The number of the entities to be arrayed (including the original); and  
the included angle of the circle (arc),

or

The included angle of the circle (arc); and  
the included angle between the Arrays to be reproduced (measured **between** the reference points  
of the objects).

Note that, during the planning of the reproduction of the Array on a circle that the circle (arc) runs **counter-clockwise** from the starting point. For a **clockwise rotation** enter negative values for the angle inputs.



### Placing the Arrays With Rotation

This option will rotate the Arrays when placing them on the circular path. The Arrays will be rotated in the same direction as the circle arc that describes the path of reproduction. The rotation angle will correspond to the angle between the Arrays (measured between the reference points of the Arrays).

After selecting the command, choosing the objects in the Array, and choosing the option **Ci.Rot** from the Option Bar, define the center of the circular path and then determine **either**;

the number of the entities to be arrayed (including the original); and  
the included angle of the circular arc,

or

the included angle of the circular arc; and  
the included angle between the entities to be arrayed (measured between the reference points of  
the objects).

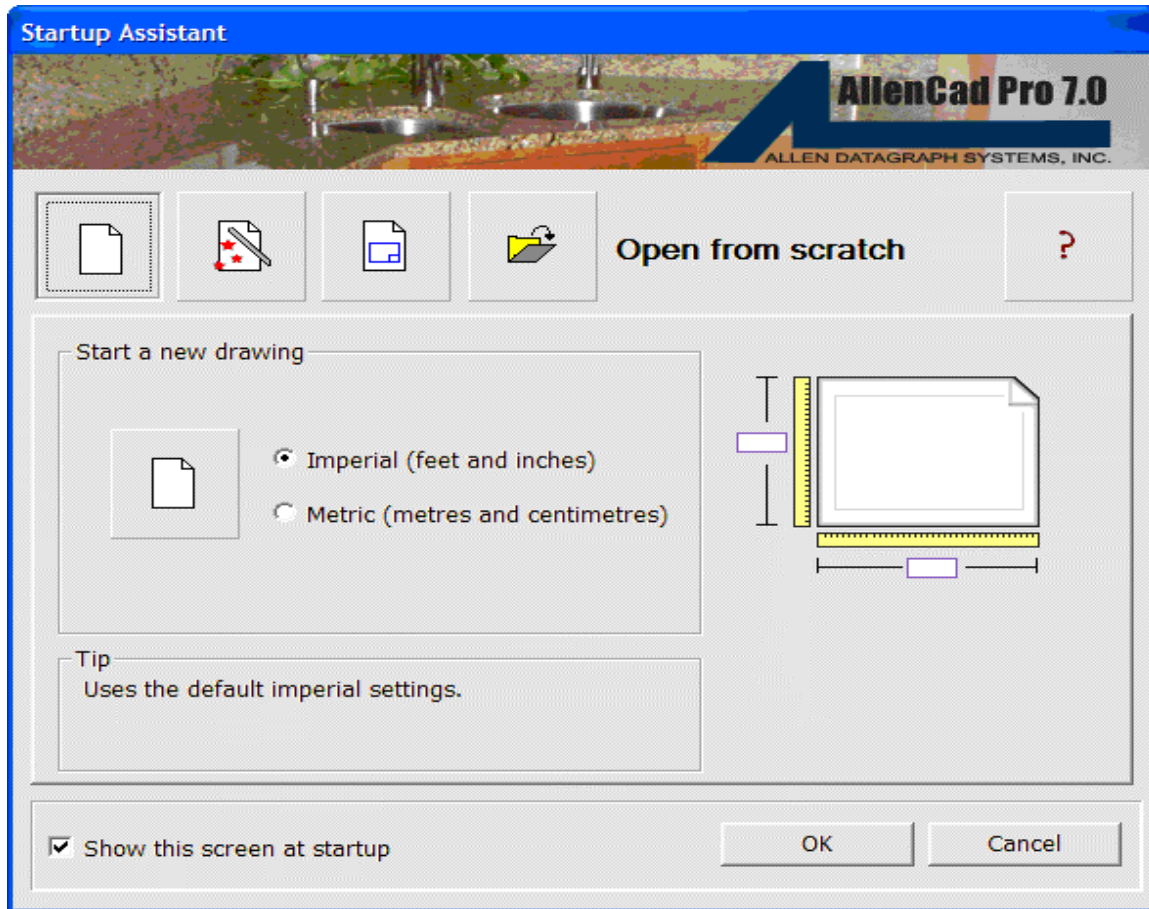
As parameter value for the Arc angle there is always used the standard value 360 degree, i.e. the copies are arrayed on a full circle, if this value has been confirmed.

Please note during the planning of the arraying on a circle of multiple copies that the circle (arc) runs from the starting point that is the original object anti-clockwise. For a clockwise rotation please enter negative values for the angle inputs.

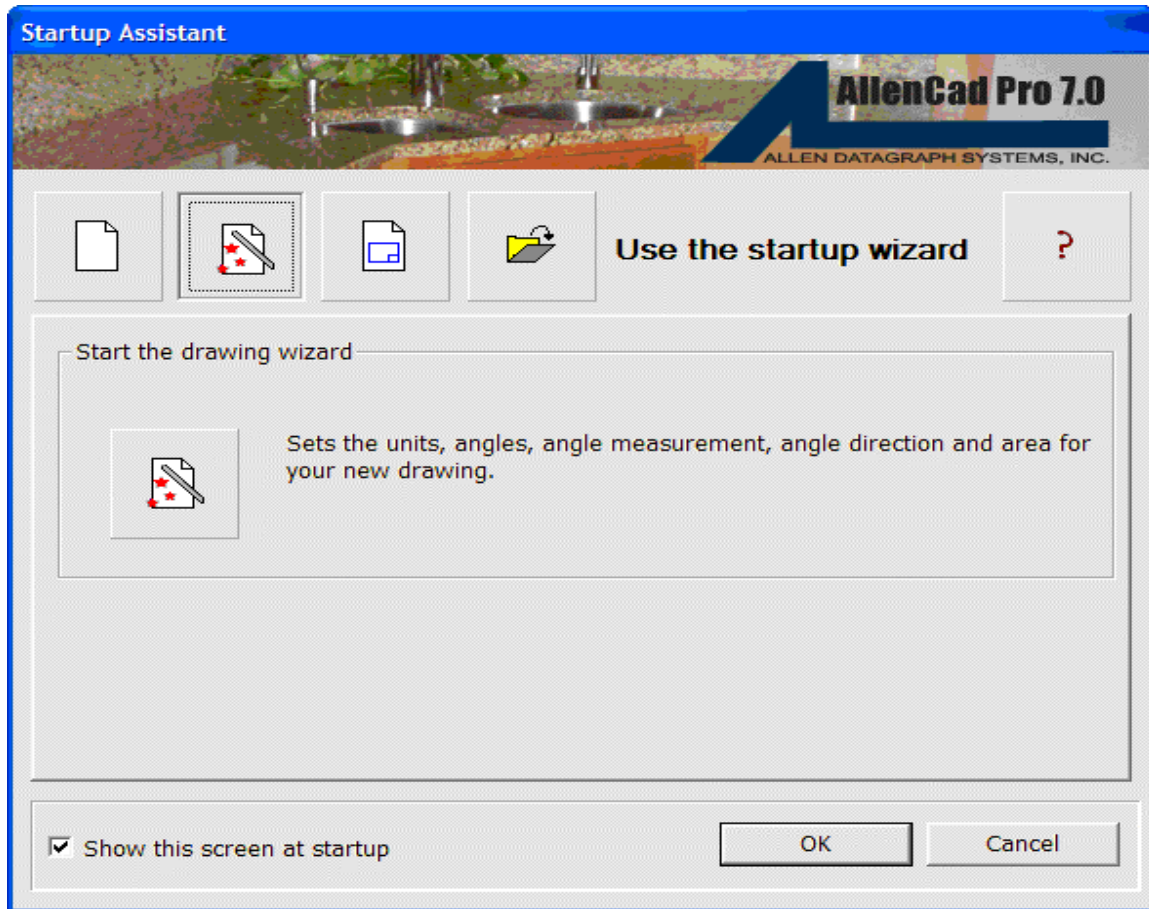
## Command ASSISTANT

The ASSISTANT command is AllenCad's one-stop drawing setup wizard. It incorporates the functions of several other configuration and program commands, including SETUP, LIMITS, TEMPLATE and OPEN. This enables you to make all of your drawing setup definitions in one location. The Assistant opens automatically during program installation, ensuring that you get up and running right away, with the settings that you need. Each time you start the program afterward, AllenCad either opens the Assistant or reminds you that the Assistant is there to help you, depending upon your preference. The command is issued by typing the command ASSISTANT (or the command alias AST) at the command prompt or selecting the desktop icon.

The startup Assistant screen is pictured below. On it, you will see four buttons, each described in the window below in the dialog, as you select each button. The first button enables you to select between the default Metric and Architectural unit settings.

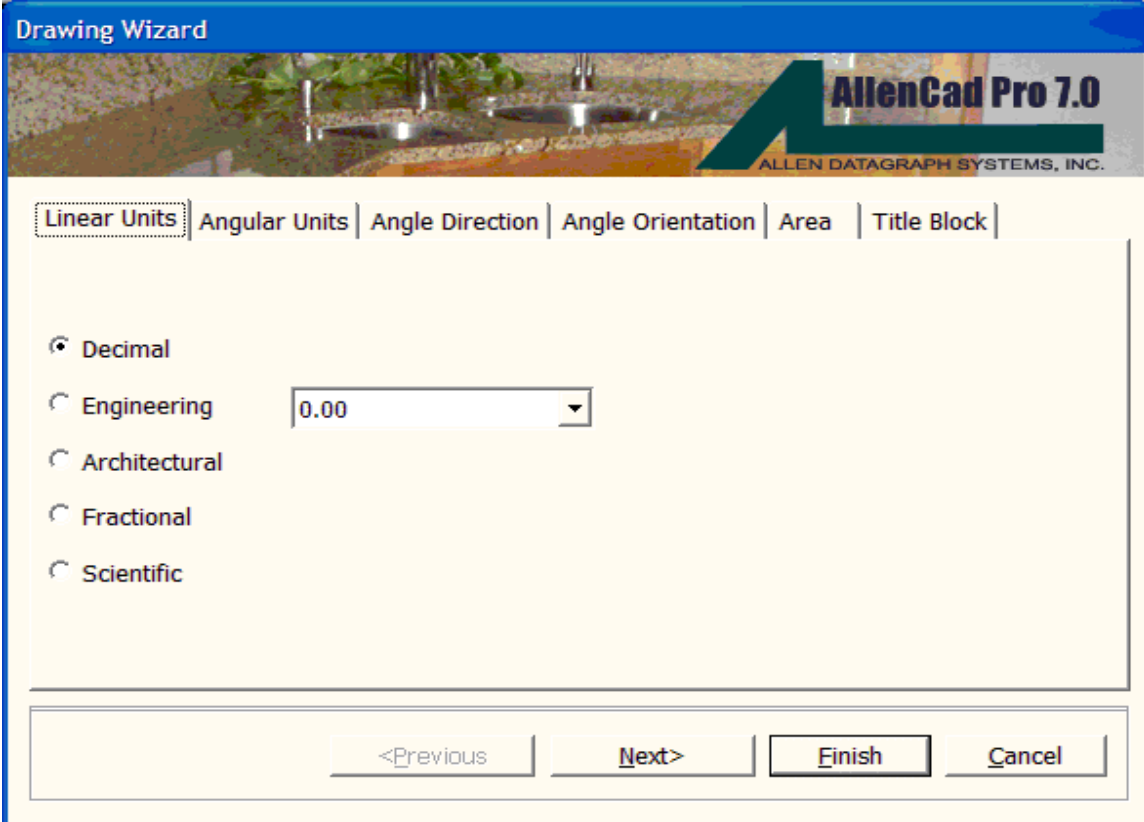


The second button is your Drawing Setup Wizard. It leads you to dialogs in which you set such things as the unit formats and accuracies that you will require, the coordinate boundaries of the drawing that you will draw, as well as the insertion of a pre-built drawing surround and title block (if you wish).



When you select the Drawing Wizard, you are presented with a dialog containing six labeled tabs. The first of them, Linear Units, is pictured below. If you will be using feet and tenths (of feet) or metric, you would select the decimal option. If your units are feet and inches, you will select Architectural or Engineering units and if you wish to measure drawing objects strictly in inches and fractions, you would select Fractional units. Scientific units are also available, if you should need to use them. Before you start though, you should ensure that you will be using only one measurement system as you draw, or else you need to have a conversion approach in mind that will let you merge data of different formats. Once you have selected the units, you must specify the precision that the program will report dimensions in. Note that this is only the reporting or labeling accuracy and that the internal, calculations are based upon a much higher level of accuracy, altogether. The illustration to the right on the dialog, shows an example of the currently selected accuracy as it will be reported.





The image shows a 'Drawing Wizard' dialog box from AllenCad Pro 7.0. The 'Linear Units' tab is selected, showing radio buttons for Decimal, Engineering, Architectural, Fractional, and Scientific. The 'Engineering' option is selected, and a dropdown menu next to it shows '0.00'. The dialog has a blue title bar, a header with a landscape image and the software logo, and a footer with navigation buttons: '<Previous', 'Next>', 'Finish', and 'Cancel'.

**Drawing Wizard**

**AllenCad Pro 7.0**  
ALLEN DATAGRAPH SYSTEMS, INC.

**Linear Units** | Angular Units | Angle Direction | Angle Orientation | Area | Title Block

☒ Decimal  
☐ Engineering 0.00  
☐ Architectural  
☐ Fractional  
☐ Scientific

<Previous   Next>   Finish   Cancel

The Angular Units tab lets you select between the different angular unit formats, in much the same manner as with the linear measurement units. As with linear measurement settings, you can also specify reporting precision and see an example of the currently selected accuracy.

**Drawing Wizard**

**AllenCad Pro 7.0**  
ALLEN DATAGRAPH SYSTEMS, INC.

Linear Units | **Angular Units** | Angle Direction | Angle Orientation | Area | Title Block

Specify the angle of measurement and the precision for angles:

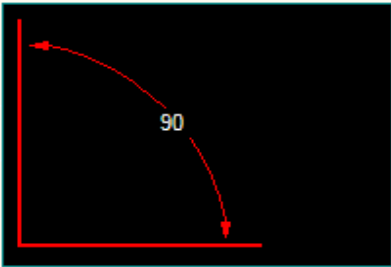
☒ Decimal      Precision:

☐ Deg/Min/Sec

☐ Grads

☐ Radians

☐ Surveyor



<Previous      Next>      Finish      Cancel

Angle direction simply sets the angle (or bearing) of "zero degrees" in your drawing. All bearings or reported drawing angles are derived from this setting. East orientation is normally selected, however North may be preferred in instances where azimuth bearings are used.

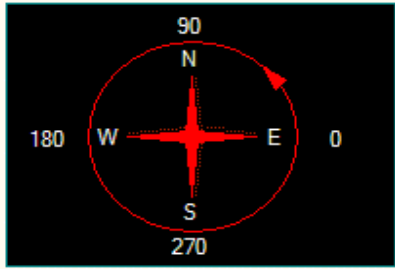
**Drawing Wizard**

AllenCad Pro 7.0  
ALLEN DATAGRAPH SYSTEMS, INC.

Linear Units | Angular Units | Angle Direction | **Angle Orientation** | Area | Title Block

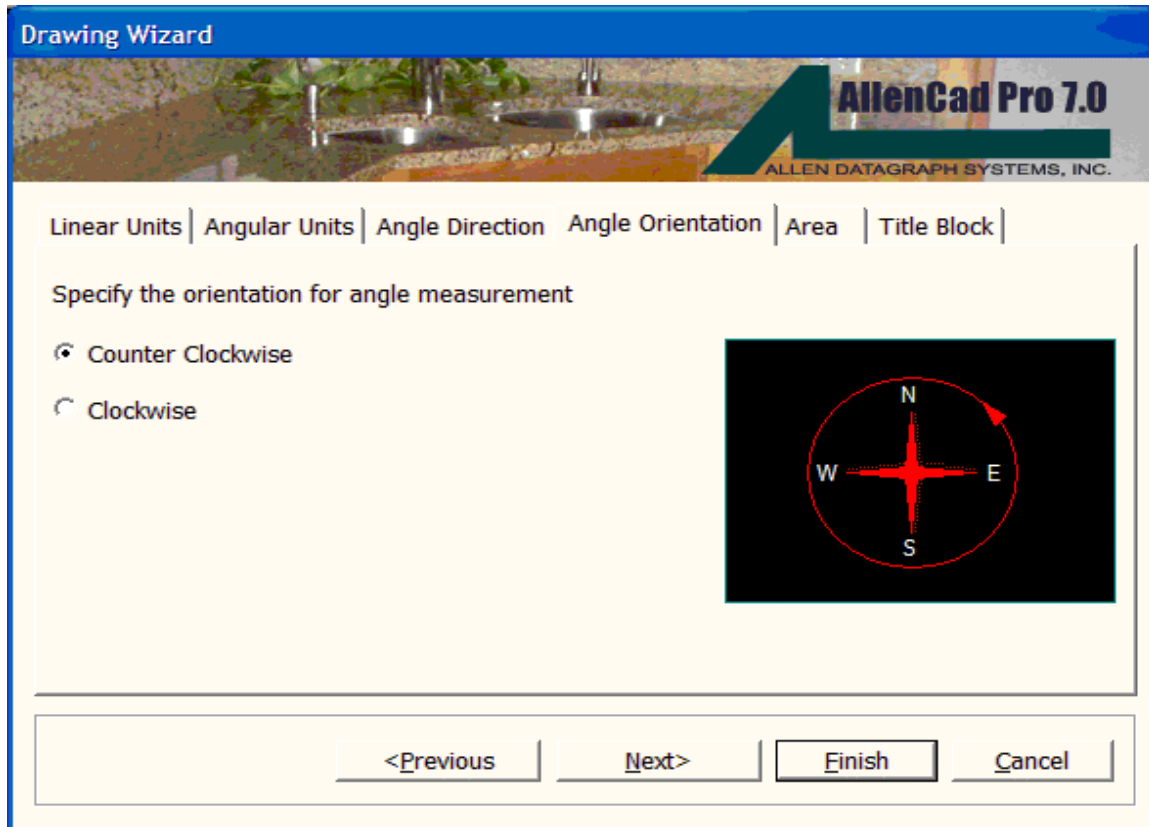
Specify the direction for angle measurement

☒ East  
☐ North  
☐ West  
☐ South  
☐ Other Angle:



<Previous   Next>   **Finish**   Cancel

Angle orientation simply specifies whether angles are measured in a clockwise or anti-clockwise direction from the specified zero alignment. By convention, anti-clockwise is most commonly used, however, clockwise measurement may be required in some instances, such as when azimuth bearings are used.




Area settings simply define the size of your drawing area. These values can be left blank if you are unsure, or if you do not wish to limit the drawing area. These values specify the "X and "Y dimensions of your drawing that correspond to the length and width of your drawing when you are looking at it in plan view (from above). There is no limit setting for elevation (the "Z coordinate values).

In the example below, these settings would be quite adequate, if the object (say a house floor plan) were to be slightly less than 35'-0x 24'-9in overall size.

Whenever you give the ZOOMPAGE command while drawing, your view of your drawing will zoom out (or in) to these limits.

Drawing Wizard



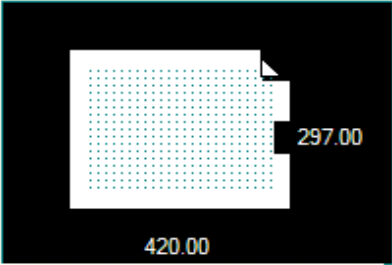
AllenCad Pro 7.0  
ALLEN DATAGRAPH SYSTEMS, INC.

Linear Units | Angular Units | Angle Direction | Angle Orientation | Area | Title Block

Specify the area you want to represent using full scale units:

Width: 420.00

Height: 297.00



Example:  
To draw in area of 420x210 mm, enter 420 under width and 210 under height.

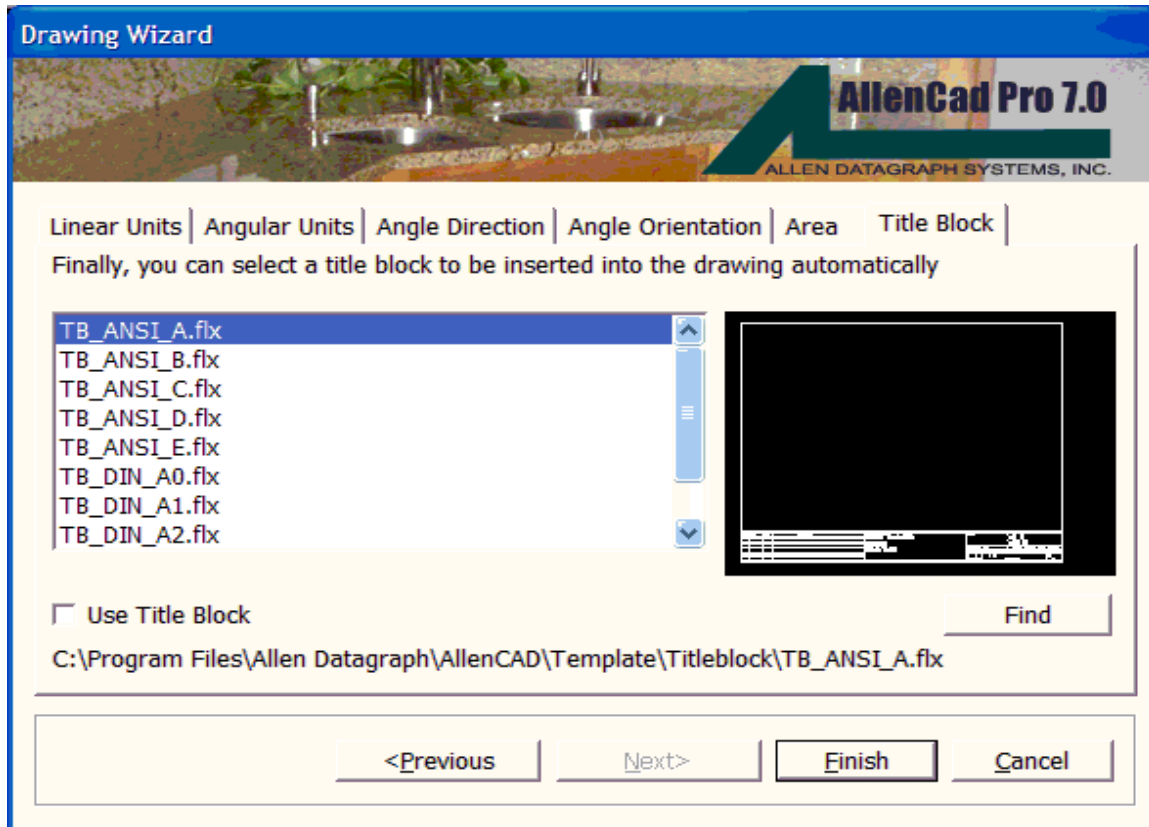
<Previous

Next>

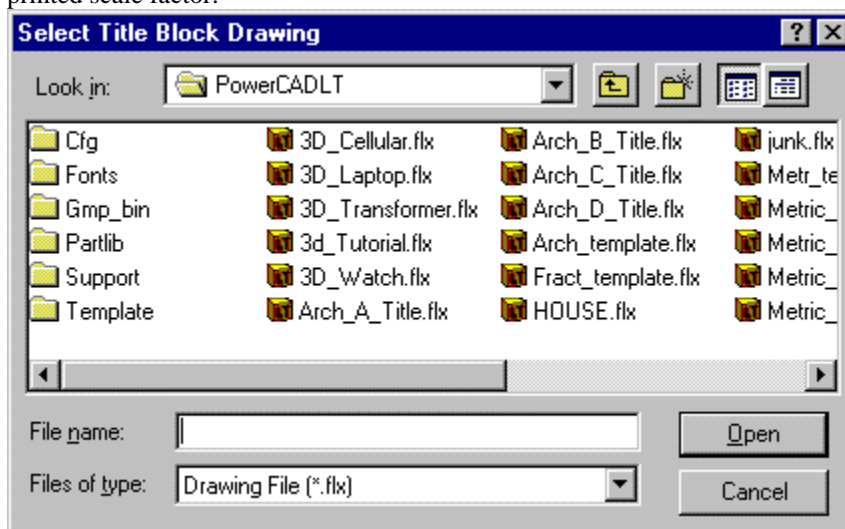
Finish

Cancel

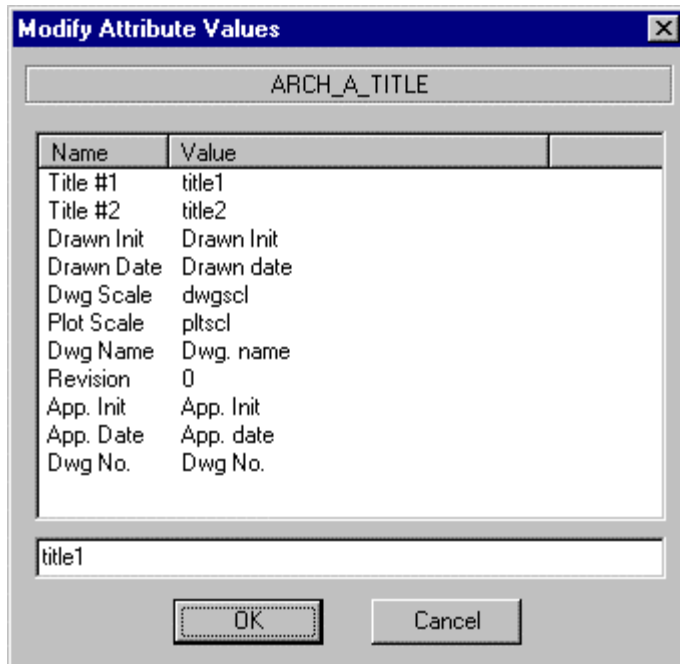
The Title Block tab helps you to insert a pre-configured title block into your drawing. This can be either one of the basic title block drawing files provided with AllenCad, or one of your own, that you have set up for this purpose.



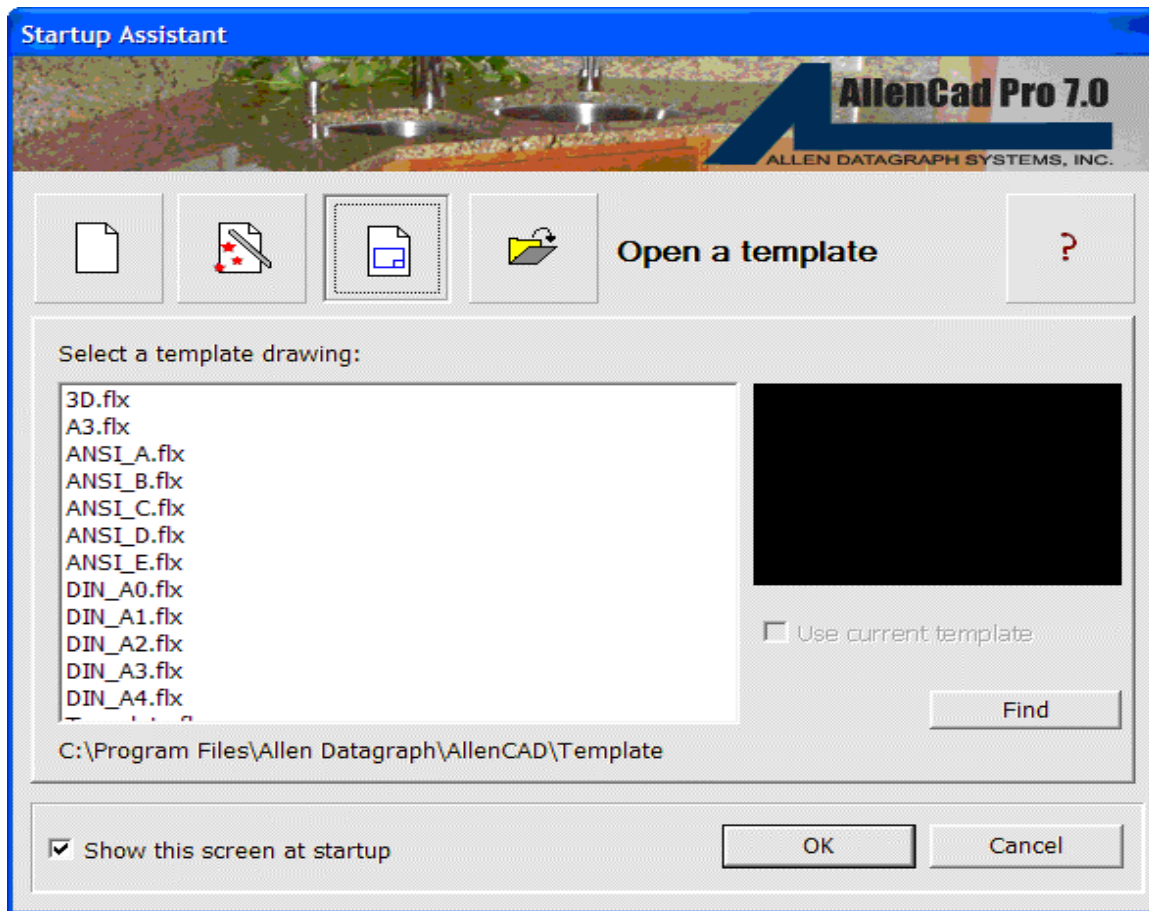
To insert one of the Architectural or Metric title blocks that are provided with your program, simply press the Find... button and navigate to the program directory. There, you will find title blocks for drawing sheet sizes A, B, C, and D as well as metric ISO size A1. Simply select the appropriate size for your drawing. This size will take into account, both the size of your drawing “model as well as the printed scale factor.



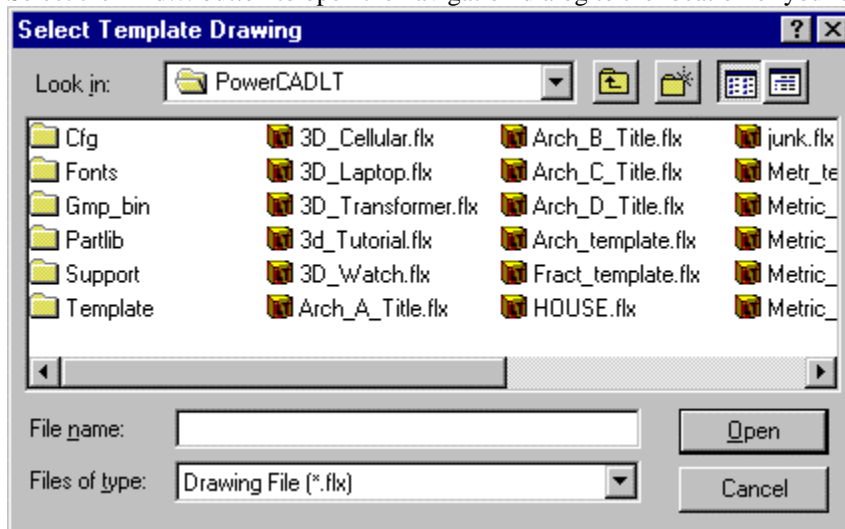
As the title block is inserted, you are presented a dialog such as pictured below that allows you to input text values that will be instead into your title block as part of the installation process. You can modify these entries afterwards, using the ATTDEDIT command, should this be necessary.



The third button on Assistant's startup dialog is the Template button. Although basic template drawings containing standard drawing setup units and parameters have been provided, you may prefer to use the Assistant to help you create new template drawing files, that contain all your own standard drawing setup information, layers, text styles, title blocks, etc. Thereafter, you can use either this Assistant utility, or set the template option to "ON" on the NEW command dialog, to automatically call your template drawing file settings into any new drawing that you create. Using drawing templates, can therefore allow you to bypass initial drawing setups when starting new drawings with standard drawing setup characteristics, making you more efficient in your drafting.

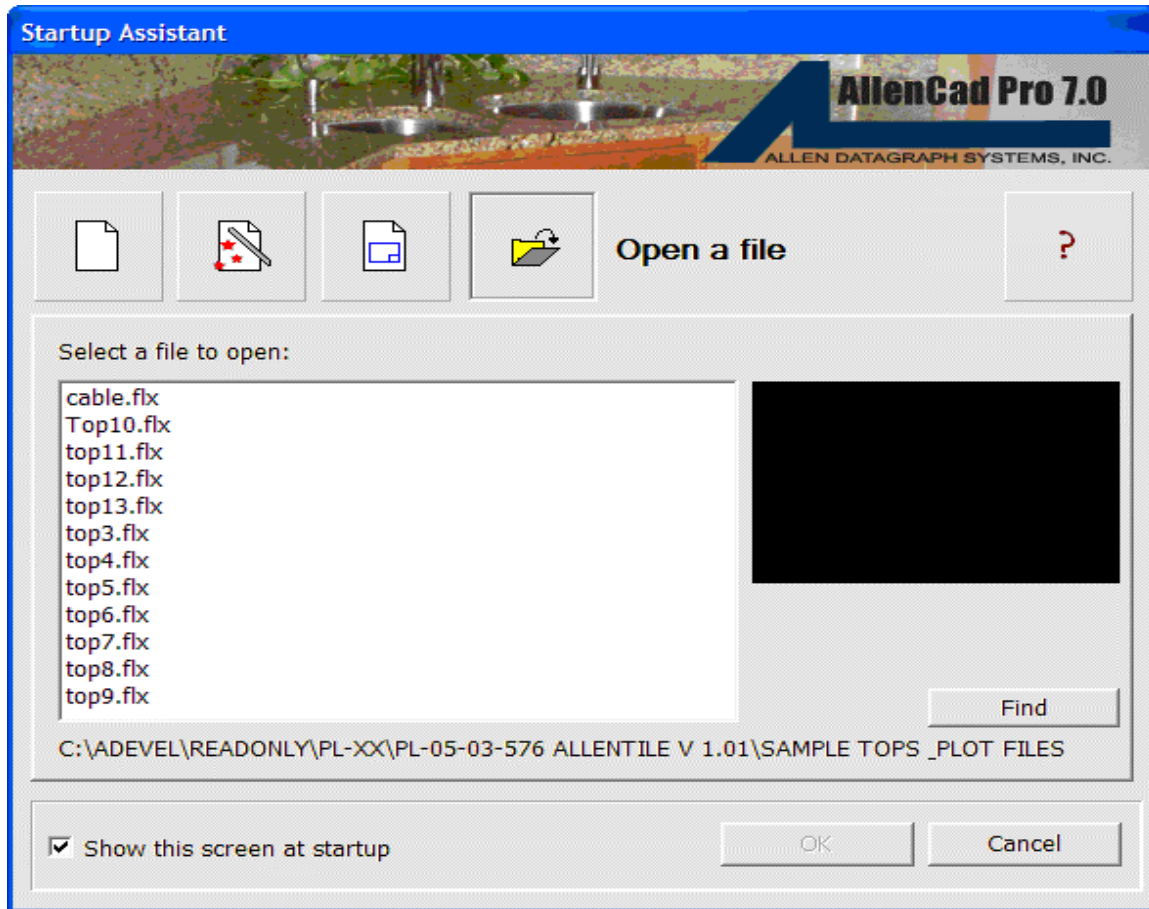


Select the Find... button to open the navigation dialog to the location of your template drawings.



The fourth and final selection button is the File Open selection button. This corresponds to the [OPEN](#) command, used to open an existing drawing. By default, AllenCad starts the ASSISTANT command by opening the dialog to this selection. If you prefer not to use the Drawing Assistant each time you start the program, simply uncheck the selection box at the bottom of this dialog.





Edit Attribute Definition...

**Command: ATTDEDIT**

**Alias ATD**

**Located:**

- 1. Pull-down Menu: Parts – Modify Attribute Definition**
- 2. Or Enter Command or Alias Directly at the Command Line**

You can modify an existing attribute definition by using the ATTDEDIT command. Select an attribute definition in the drawing. Immediately a dialog box is displayed that is similar to the dialog to create attribute definitions.

If the check box labeled Interactive is marked, you will be prompted to relocate the attribute definition when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

> ATTDEDIT

Select a attribute definition:

*<Dialog box appears to edit attribute definition parameters>*

Insertion point:

Hint: ATTDEDIT makes it easy to create several similar attribute definitions. Just copy a master attribute definition, and then edit the name and request parameters with the ATTDEDIT dialog.

---

**See also:**

ATTDEF

## Define Attribute...



**Command: ATTDEF**

**Alias AD**

### **Located:**

- 1. Pull-down Menu: Parts – Define Attribute**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Attributes are, in general, text objects that are related to, or joined to, Parts (Blocks). They are treated together with the Part as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single Attribute with a Part, or several.

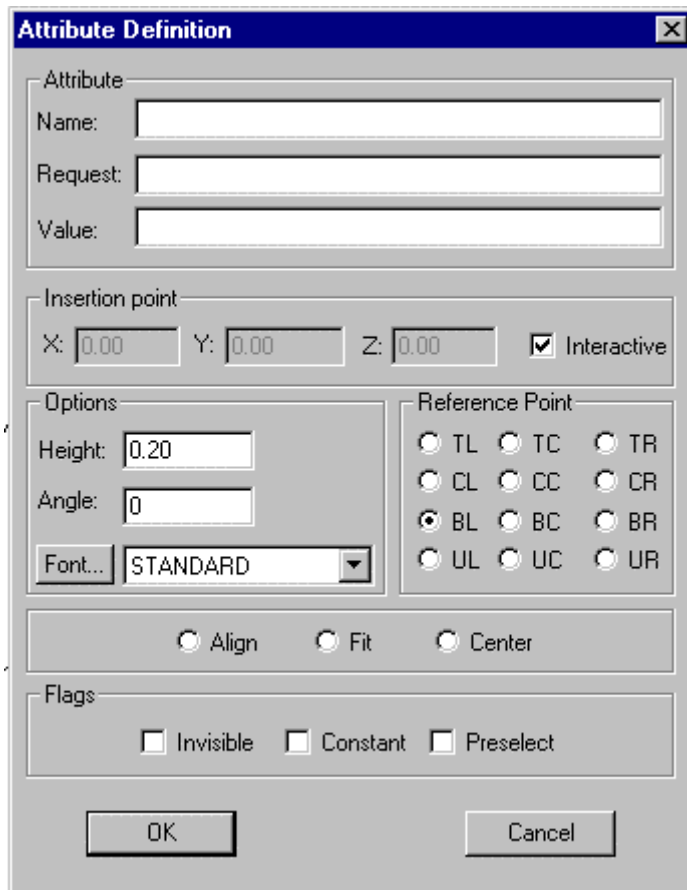
Using Attributes, the parts can be further defined by textual information. This information may be used to label Parts in the drawing, or may be used to contribute to a data base program for analysis later in the engineering process.

An Attribute consists mainly of three components; its name; its request; and the value. There are also a series of options that can be elected during the creation of an Attribute.

- The **Name** is the designation of the Attribute under that it is integrated in a part. Especially if a part contains more than one Attribute, the name is the most important criteria for the effective usage of the Attribute.
- The **Request** is a component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally, the request contains the invitation to enter a variable Attribute value.
- The **Value** is that portion of the Attribute that is written, together with the Part, into the drawing. It can be constant or variable, visible or transparent.

The following parameters for an Attribute may be defined during the creation of the Attribute, and edited later in the design process.

- Insertion point;
- Font options, specifically Font name, font height and font angle;
- Reference point for insertion;
- Visibility, whether visible or hidden; and
- Attribute value, whether constant or variable.



The image shows a software dialog box titled "Attribute Definition". It contains several sections for configuring an attribute:

- Attribute Section:** Includes input fields for "Name:", "Request:", and "Value:".
- Insertion point Section:** Includes input fields for "X:", "Y:", and "Z:" (all set to 0.00), and a checked checkbox for "Interactive".
- Options Section:** Includes "Height:" (0.20), "Angle:" (0), and a "Font..." dropdown menu currently set to "STANDARD".
- Reference Point Section:** A grid of radio buttons for selecting a reference point: TL, TC, TR, CL, CC, CR, BL (selected), BC, BR, UL, UC, UR.
- Alignment Section:** Radio buttons for "Align", "Fit", and "Center".
- Flags Section:** Checkboxes for "Invisible", "Constant", and "Preselect".
- Buttons:** "OK" and "Cancel" buttons at the bottom.

## Dialog Options

### Attribute Name

Enter the name of the Attribute in the input field. This is the name that the Attribute will carry as it is joined to the Part, so it is important to use as descriptive a name as possible. When a part contains several Attributes, the name is an important criteria for effective use of the Attribute. An Attribute name can consist of up to 31 characters, including the special characters \_ (Under line) and \$ (Dollar).

### Request

The Request is a textual component of the Attribute that can be displayed during the insertion of the Part in the command line area. Normally the Request contains an invitation to enter a variable Attribute value.

Request can consist of up to 256 possible characters. Blanks and special characters are allowed.

The user may enter in this input field, for instance, "Enter norm term". This would be a variable. A typical answer to Request would be to enter information like "Schedule 40 steel pipe".

The answer to Request is interpreted as an Attribute value and as such inserted in the drawing (see below).

Occasionally the entering of a request can be omitted. This would be the case especially when the part is using a constant Attribute value (see below).

### Value

The Value is that component of the Attribute that is written, with the Part, into the drawing. It serves as a designation of the Part and contains information to be analyzed. Its content is determined either by the input in the input window Value or by the answer to Request (see above). An Attribute Value can

consist of up to 256 possible characters. Blanks and special characters are allowed.

### **Insertion Point**

There are two options in defining this point in the Dialog Box; the entry of absolute coordinate values in the X, Y, and Z axes, or Interactive.

Interactive allows the user to define the Insertion Point by the cursor. By filling the control box marked Interactive, the user can exercise this option. The advantage to this option is that Dragmode is active as well, offering a preview of the positioning.

### **Text Options**

With the help of the text options the user may determine the height (font size), the rotation and the Font style of the Attribute text.

**New text styles** can be directly created from the **ATTDEF** command

The **FONT** command can be issued by depressing the font button on the text dialog

Define the font size by the input field Height. Enter the desired text height in drawing units, not in text points.

Via the input field Angle, specify the insertion angle of the Attribute value. For an counter-clockwise rotation enter a positive value, for a clockwise rotation enter a negative value, between 0 and 360 degrees.

A selection window is available to define the Font style. A list of the available fonts is displayed. Select the desired Font style.

### **Reference Point**

The Reference Point determines the position and alignment of the text object in relation to the Insertion Point of the Attribute. To this end, a Reference Point is defined on the text object. During insertion, this point will be coordinated with the Insertion Point.

If the reference point, for instance, is the lower left corner of the text object, then the text will run from the Insertion Point to the right, and the lower edge of the text will be located at the height of the Insertion Point.

In total there are twelve Reference Points available. Select one of them with the cursor. The Reference Points represent the initials of the position on the text object. BL, for instance, represents Bottom Left, while UR represents Upper Right. Center positions are available as well.

### **Align**

Text objects may be rotated or scaled during the process of insertion by using the function Align. Determination of rotation angle, as well as a scaling factor, is made by indicating two points in the drawing using the cursor.

Activate the option Align by filling the circle marked "Align". Activating this option will remove the options Insertion Point, Height, Angle, and Reference Point. Align requires that the user pick two points on the drawing. The text object will be placed left justified on this baseline.

Confirm the use of the option Align after the input of the string by clicking OK.

Then simply follow the prompts for a first point, then a second point. The first point is interpreted as the Insertion Point, the second point sets the angle of insertion. The second point will also determine a scaling factor that will, in turn, determine the text height.

### **Fit**

Fit enables the user to rotate and expand a text object during the insertion. The text object is not scaled, so the text height remains unchanged, but the text string is expanded or compressed as required by the Fit. The procedure for this option is:

Activate Fit by clicking the panel with this name at the lower left side of the dialog window Text. The options Insertion Point, Angle and Reference Point will not be available. The parameters required for Fit are two points that must be defined during the inserting of the text object. The reference point of the text object is automatically positioned left justified on the basis line;

Answer the prompt for "first point". This point is interpreted by the program as the Insertion Point. It is also the point of rotation for the rotation angle and the first point for the expansion factor; then

Indicate the second point. This will determine the rotation angle and expansion factor, and set the text height.

### **Center**

Center allows fast insertion of a text object with the vertical and horizontal alignment centered. Text height and rotation angle may be specified in the normal fashion.

### **Flags**

#### **Visibility**

Invisible is a switch panel that controls the visibility of the Attribute Value. A cross in the control box displays active status of the option.

If Invisible is activated, the Attribute Value will not be displayed during the insertion of the Attribute in the drawing. this is the option to choose if the Attribute contains information to be analyzed later that does not have to be displayed on the drawing.

It is also convenient to use this option when there are so many Attributes that to show them all would make the drawing to "busy", confusing the drawing.

#### **Constant**

This switch panel is used for switching between constant and variable Attribute values (see above). A cross in the control box displays the active status of the option.

With Constant activated, the text entered in the Value field will be written as a constant Attribute Value in the drawing. There will be no opportunity to insert any variable data.

With Constant de-activated, the request at the command line to insert variable Attribute Values will appear.

#### **Preselect**

The Preselect mode allows the user to create Attributes that accept their default values.

With Preselect active, Attribute Values are not requested. The Preselect mode is the most flexible mode, since the mode allows the user to edit pre-selected Attribute Values after insertion by using the editing commands.

Verify the entries by OK switch or by pressing RETURN. The Attribute will be inserted as specified.

**See also:** QATTDEF

## **Modify Single Part Attributes (one by one) ...**



**Command: ATTEDIT**

**Alias ATT**

#### **Located:**

- 1. Pull-down Menu: Parts – Modify Part Attributes (1 by 1)**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Enter the command ATTEDIT in the command line area or select **Modify Attributes...** from the *Part* menu.

The first prompt will request that the part be selected. Make this selection either by naming the part, or by selecting the part with the cursor.

After the part selection, a dialog box will appear. The editing process from this point is very similar to the procedure followed in the creation of an Attribute, except that the name is now displayed, and cannot be changed.

If a part contains several Attributes, individual Attribute values can be displayed in the input field *Value* using these external. Clicking the right double arrow will display the next Attribute, while clicking the left double arrow will display the previous Attribute is displayed. That Attribute can at this moment be edited results from the name display above the input field.

---

**See also:**

Edit Part Attribute

Export Attribute Information



**Command: ATTEXP**

**Alias ATX**

**Located:**

1. **Pull-down Menu: Parts – Export Attribute Information**
2. **Palette: Parts**
3. **Or Enter Command or Alias at the Command Line**

Writes attribute information to a file.

With this function, the user can extract attributes for parts or entities from the user drawing and save these in a CDF-format file. This is important when transferring and analyzing information from the drawing to or from programs for further processing

Proceed as follows

Enter the command **ATTEXP** or enter the function by selecting the menu point **Export Attribute Information...** in the pull-down menu Part.

Select the objects whose attributes are to be analyzed and the information that to be exported. The usual object select functions in the Options Bar are available.

**Select attributes**

The left-most table lists the attributes that have been selected for export. An *N* indicates that it is a numeric value, and a *C* indicates that a character string is to be exported.

The next number displays the length of the field input for the attribute. For numeric values, the number of the decimal digits is also displayed.

Switches for Adding and Deleting attributes to the selection list are placed underneath the selection table. To delete attributes from the list, mark one or more lines in the list and select the switch Delete.

**Select Part Properties for Export**

The user can select that part properties are to be included in the CDF-file in a manner similar to adding attributes.

Select the switch Adding / Changing below the display and selection field for the parts and properties. All of the properties that can be exported will be shown in the selection area. Select a property by selecting the marked field. Enter the desired number of characters for the field width and, if necessary, edit the number of decimal digits.

### **Change Field Separators**

In the CDF file, all fields are separated by *Field separators*. Non-numeric fields are included in *text separators*. These values can also be changed in this box.

### **View and Write CDF Files**

When the user has selected all of the information desired, determine the name of the output file. A list will appear in the field CDF file. The switch '...' on the right side next to the input field for the file name displays a file selection box.

The switches Display and OK allow the user to search the drawing for the parts with the selected attributes, and the output in the CDF format is prepared. If the user selects the switch Display, a file will not be written, but will be displayed in a tabular Dialog Window.

From this window, individual lines, or all lines, can be copied to intermediate storage (Clipboard) for processing by other Windows programs.

### **Attribute Modification**

#### **Command: ATTVALUE**

**Alias AV**

##### **Located:**

- 1. Pull-down Menu: Parts – Modify Attribute Text – Modify Attribute Text Value**
- 2. Or Enter Command or Alias at the Command Line**

#### **Command: ATTMOVE**

**Alias ATM**

##### **Located:**

- 1. Pull-down Menu: Parts – Modify Attribute Text – Move Attribute Text**
- 2. Or Enter Command or Alias at the Command Line**

#### **Command: ATTROT**

**Alias ATR**

##### **Located:**

- 1. Pull-down Menu: Parts – Modify Attribute Text – Rotate Attribute Text**
- 2. Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

### **Edit Attribute Text Value**

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu

*Modify Attribute Text.* ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

### **Move Attribute Text**

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

### **Rotate Attribute Text**

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

## **Attribute Modification**

**Command: ATTVALUE**

**Alias AV**

#### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Modify Attribute Text Value**
2. **Or Enter Command or Alias at the Command Line**

**Command: ATTMOVE**

**Alias ATM**

#### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Move Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

**Command: ATTROT**

**Alias ATR**

#### **Located:**

1. **Pull-down Menu: Parts – Modify Attribute Text – Rotate Attribute Text**
2. **Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.



### **Edit Attribute Text Value**

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu *Modify Attribute Text*. ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

### **Move Attribute Text**

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

### **Rotate Attribute Text**

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

### **Attribute Modification**

**Command:** ATTVALUE

**Alias** AV

**Located:**

1. **Pull-down Menu:** Parts – Modify Attribute Text – Modify Attribute Text Value
2. **Or Enter Command or Alias at the Command Line**

**Command:** ATTMOVE

**Alias** ATM

**Located:**

1. **Pull-down Menu:** Parts – Modify Attribute Text – Move Attribute Text
2. **Or Enter Command or Alias at the Command Line**

**Command:** ATTROT

**Alias** ATR

**Located:**

1. **Pull-down Menu:** Parts – Modify Attribute Text – Rotate Attribute Text
2. **Or Enter Command or Alias at the Command Line**

The program offers several utilities to edit and modify attribute text entities. These utilities can be chosen from the Modify Attribute Text sub-menu in the pull-down menu Parts.

### **Edit Attribute Text Value**

Call the utility by keyboarding ATTVALUE or select **Edit Attribute Text Value** from the sub-menu *Modify Attribute Text*. ATTVALUE allows the user to select a single attribute of a part insertion and change its value at the command line:

```
> ATTVALUE
Select attribute of part: P1
New value for attribute <47>: 48
```

### **Move Attribute Text**

Call the utility by keyboarding ATTMOVE or select **Move Attribute Text** from the sub-menu *Modify Attribute Text*. Use the command ATTMOVE to select a single attribute of a part insertion and modify its location:

```
> ATTMOVE
Select attribute of part: P1
Target point: P2
```

### **Rotate Attribute Text**

Select Rotate Attribute Text from the sub-menu Modify Attribute Text or call the utility by keyboarding ATTROT. The command ATTROT allows the user to select a single attribute of a part insertion and modify its rotation angle:

```
> ATTROT
Select attribute of part: P1
New angle <0>: 90
```

Edit Part Attributes...



**Command: ATTXEDIT**

**Alias AX**

#### **Located:**

- 1. Pull-down Menu: Parts – Edit Part Attributes**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

If only the Attribute values shall be changed, please use the command ATTXEDIT or call the function **Edit Part Attributes...** from the menu *Part*.

After the calling of the command you will be requested in the command line area to select the part of that the Attributes shall be edited. For this it is enough to click the desired part with the cursor at any spot. You can always select only one part.

After the selection of the part the program calls a dialog box.

In the window the Requests (left) and the current Values (right) of the Attributes included in the selected part are listed. Select the Attribute value to be changed. The selected value is transferred to the editing field (below) and can be edited.

---

#### **See also:**

Modify Part Attributes

QATTEDIT

"B"

BATPROC

### Command BATPROC (Batch Processing)

Alias BT

#### Located:

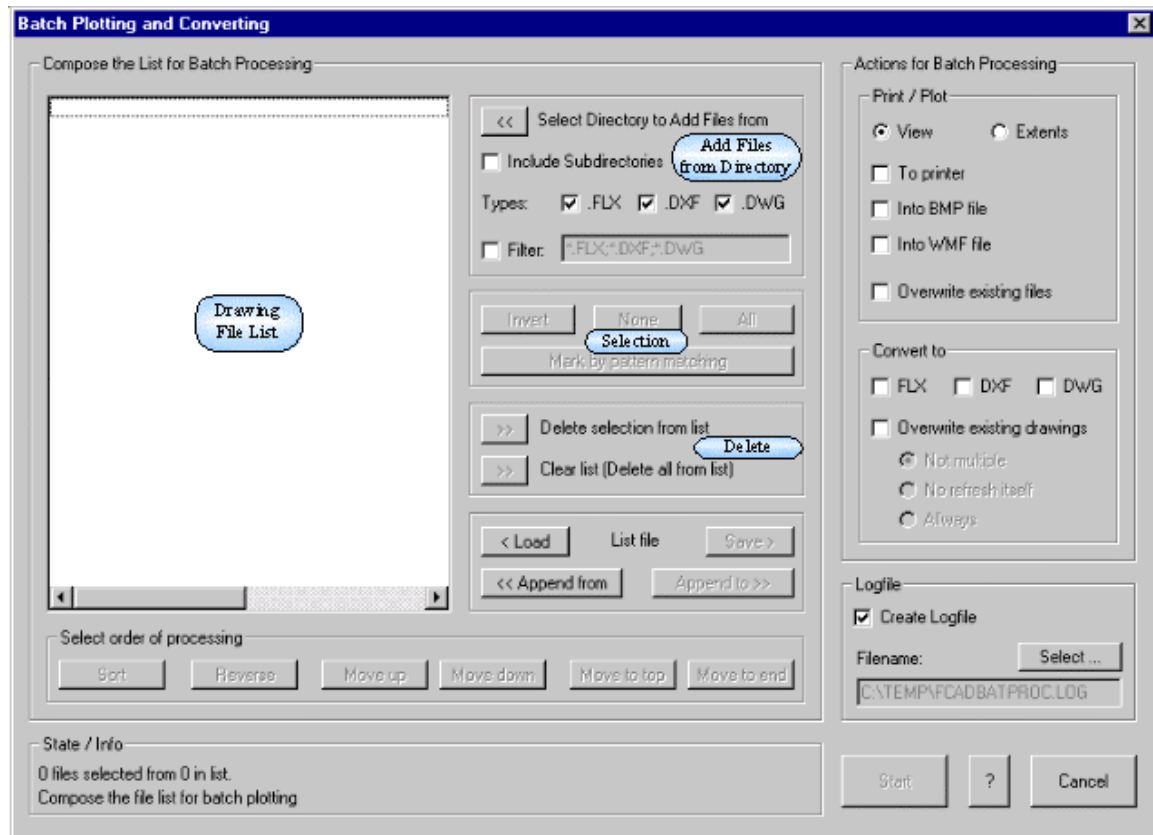
1. Pull-down Menu: File – Batch Processing
2. Or Enter Command or Alias at the Command Line

### Introduction

The command BATPROC makes it possible for you to arrange a list of drawing files and to do something with each drawing on this list. There are many built in capabilities to build the list and manipulate it. After arranging the list you can plot each file on the list not only to the printer but also to a Bitmap file (.BMP) file or into a Windows Metafile (.WMF). Moreover you can batch-convert drawings from one format (of the set DXF, FLX or DWG) to another of this set.

### Preferences and Start of the Command

You can start the command simply by typing "BATPROC" or by selecting the menu point BATCH PROCESSING inside of the menu TOOLS. Now the following dialog box appears at the screen:



This dialog is divided in two main parts, on the left side you find the functions to compose and manipulate the list, while on the right part the actions for the batch process can be defined. Following the

meaning of the fields will be described.

## Boundary Hatching



**Command: BHATCH**

**Alias BH**

### Located:

1. **Pull-down Menu: Detail – Boundary Hatch**
2. **Palette: Detail**
3. **Or Enter Command or Alias Directly at the Command Line**

The command BHATCH fills a region with a hatch pattern or solid, bounded by linear entities, just by pointing to any point within the boundary.

The BHATCH command defines a boundary by creating a polyline that follows the entities that make up the boundary. Select each entity to define a boundary around the area you want to hatch.

> BHATCH

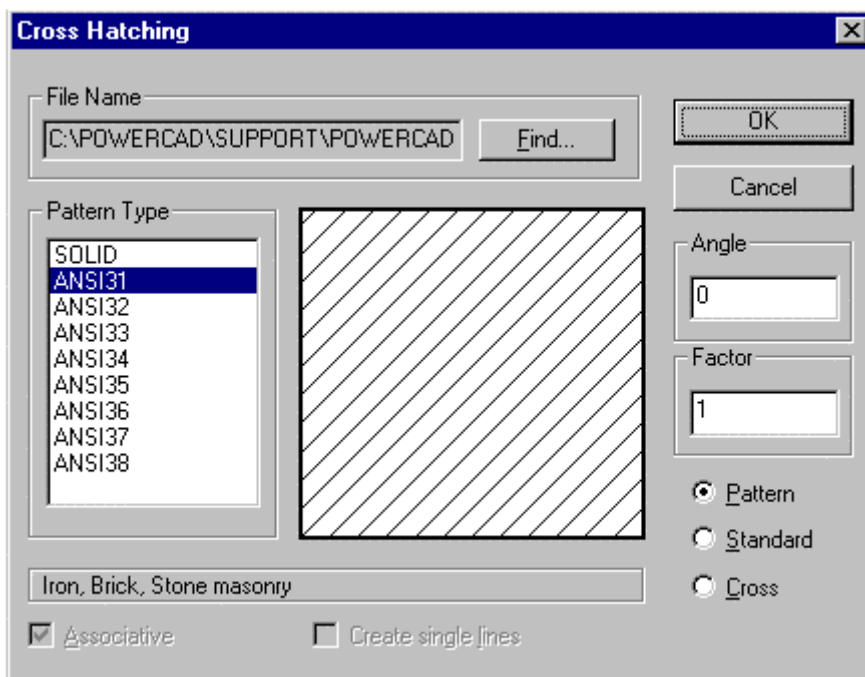
Select objects:

Select point inside boundary contour <+X>:

Select additional elements? <N>:

<Dialog box appears>

Note: to exclude internal areas from hatch application, select the internal boundaries to be ignored when prompted for “Select additional elements.”



Hatch patterns are stored together in .pat files. The file name input at the top of this dialog allows you to select a pattern from any of a number of .pat files that you may have available. Having selected the file, all patterns included in that file are listed in the Pattern Type window on the left. Clicking on any of these causes a representation of that pattern to appear in the central window, making selection easier.

The selections to the right on this dialog modify the pattern, in which:

- Angle – Will cause a counter-clockwise rotation of the pattern by the angle entered;
- Factor – Will cause the pattern to become more condensed or expanded, depending upon the entered factor value;
- Pattern will display a non-uniform pattern (if that is available in the particular selection);
- Standard – Will modify the pattern to a uniform spacing and
- Cross – Will create a "criss-cross" pattern effect.

The hatch will be applied immediately after the OK button is pressed.

### **Hatch Color**

Hatch color is, as with other drawn entities, determined by current layer color setting, or else by a set color that over-rides bylayer color settings. Color fills are achieved by making the selection of "Solid" in the pattern name window.

Further information on hatch pattern definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

---

#### **See also:**

Hatching  
Modify Hatching  
Polyline Boundary  
Polyline Contour

## **Command BMAKE**

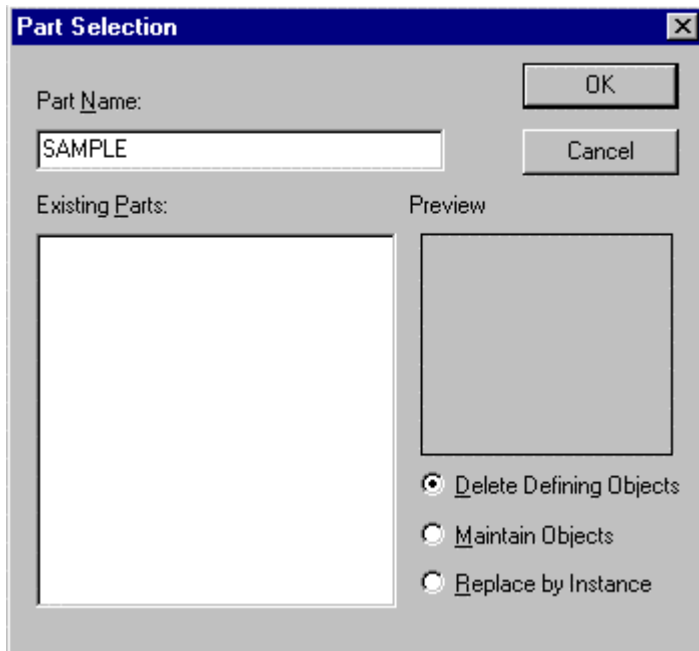
**Command: BMAKE**

**Alias BM**

#### **Located:**

- 1. Enter Command or Alias Directly at the Command Line**

This command is used to create internal part definitions, with the assistance of the dialog below. First, the command asks you to define the new part's insertion point.



In the dialog, we input the name of our new part and have used "Sample" for this illustration. The program lists all other defined parts, in case we want to include them in the new part that we are creating. You will also notice three alternatives that are available, concerning how the program treats entities chosen to be part of the part. Whether the selected objects are deleted, retained or replaced by the new part that is being created, can be set by clicking on the appropriate "radio button" in the dialog.

When you click on OK, you are asked to select the objects that will comprise the new part. Part creation is completed immediately, once these objects have been selected.

Sample Input:

```
> BMAKE
Insertion base point:
Select objects:
Other corner:
3 selected.
Select objects:
*** 3 selected. ***
```

## Command BMP2RLC

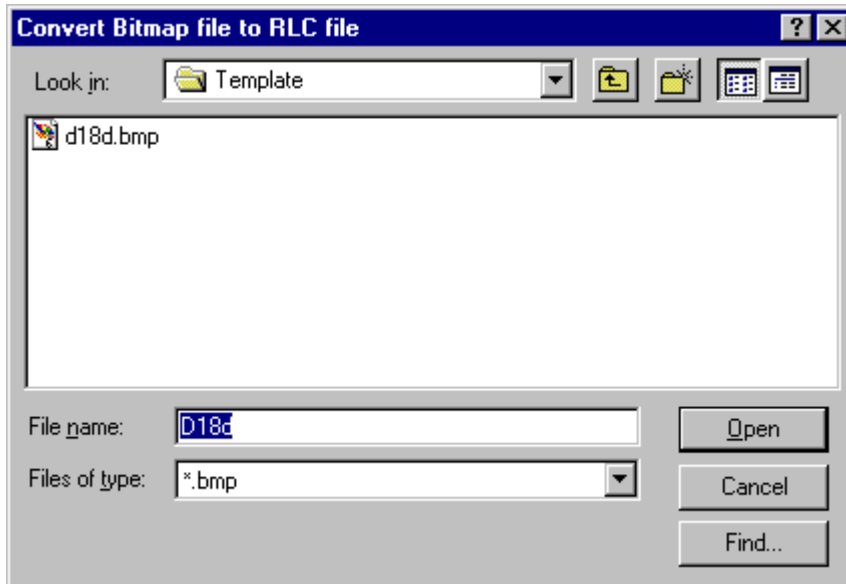


**Command BMP2RLC**

**Located:**

- 1. Palette: Raster**
- 2. Or Enter Command Directly at the Command Line**

Like its counterpart (RLC2BMP), BMP2RLC simply converts an external monochrome bitmap (BMP) file to RLC format. When issued, the command calls the following dialog:



To complete command execution, simply navigate to your BMP file through the "Look in:" drop down window and select the file. The file is then immediately converted with the new file remaining in the same location as the original and having the same filename but with .rlc extension.

## Command BMPEDIT

### Command: BMPEDIT



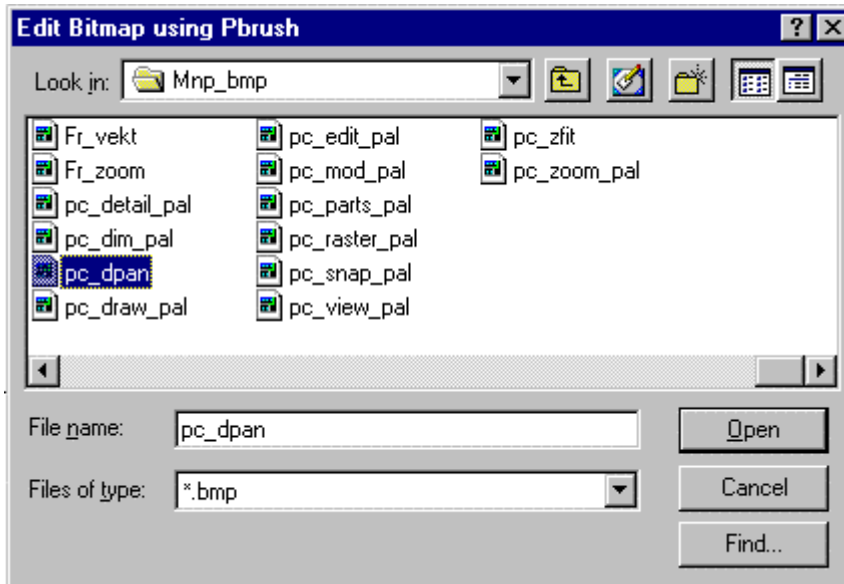
Alias BPE

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Run Bitmap Editor**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

This command allows the user to edit an external Bitmap graphic (.bmp), using the Windows Paintbrush program. In this way, you can make necessary changes to the graphic before importing it into AllenCad, without ending your AllenCad session.

This command can be run from the "Resource Manager - Run Bitmap Editor" selection on the Tools pull-down menu or it can be typed at the command line. The following dialog appears onscreen once the command is issued, allowing you to browse and locate the bitmap file that you need to edit.



The bitmap is opened in the Paintbrush program once the selection has been made. Please consult Windows' documentation for instructions on the use of the Paintbrush program.

## Copy Region to Bitmap



**Command: BMPOUT**

**Alias BO**

**Located:**

1. **Pull-down Menu: File – Export Region To Bitmap**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

Call the command either by keyboarding BMPOUT or by selecting **Copy Region to Bitmap...** from the menu *Edit*.

The following options allow all **entities** in the region to be copied into a bitmap:

### Window

Define a window by setting two points.

### View

The complete current view will be exported into a bitmap.

### Extents

The region to be copied is defined by the extent of the drawing.

Next, specify the **Width** and **Height** of the bitmap in Pixel Units (10...2000 pixels).

Finally, you may choose whether the bitmap should be written into a **File** or copied to the Windows **Clipboard**.

### Notes:

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The bitmap remains in the clipboard until you clear it. Place new objects on it with the



RENDER, BMPOUT, WMFOUT or other command, or Cut or Copy in other Windows applications.

## Command BOUNDARY

**Command: BOUNDARY**

**Alias BD**

**Located:**

1. **Enter Command or Alias Directly at the Command Line**

This command functions in an identical fashion to another command, PCONTOUR. Both commands draw a closed polyline around the perimeter of an area enclosed by selected entities. The user is asked to select the entities that form the boundary to the area and then to pick a point within the bounded area. The program defines the boundary and draws the polyline based on this input information.

### Create Boundary Polyline for Hatching



**Command: BPOLY**

**Alias BP**

**Located:**

1. **Pull-down Menu: Draw – Polyline Contour**
2. **Left SideBar**
3. **Palettes: Draw and Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

The command BPOLY creates a boundary polyline from existing objects that form an enclosed area, most often for use with the HATCH or AREA commands.

The original entities can be made up from a combination of lines, arcs, circles, and polylines.

They can be in any arrangement as long as the boundary will make up an enclosed area without looping back upon itself.

Note: The BHATCH command can be used to create its own boundary polyline, while applying a hatch pattern to an enclosed area.

---

#### See also:

Command PCONTOUR

PCONTOUR is the command line version of the BPOLY command.

Break

**Command: BREAK**

**Alias B**

**Located:**

1. **Pull-down Menu: Modify - Break**
2. **Or Enter Command or Alias Directly at the Command Line**

The **BREAK** command operates identically to the command **DELPARTIAL** – One Point option. Like this command, the command **BREAK** breaks objects at a point or points you select. **BREAK** has been added for convenience of those users who are familiar with other CAD programs.

The main difference is that the command **BREAK** assumes that you might want to use the pick point when selecting the entity as the first point of the part to delete:

```
> BREAK  
Select entity (line, circle, arc, 2D-polyline):  
Second point (or F for first point):
```

However, you can respond to the prompt for the second point the Option *First* (short *F*) to explicitly specify both break points independent from the entity selection:

```
> BREAK  
Select entity (line, circle, arc, 2D-polyline):  
Second point (or F for first point): F  
First point:  
Second point:
```

Also, you have the choice to break an object into two pieces without erasing any part of it. To do this, use the command and specify the same point for both the first and the second points; type **@** at the request for the second point that signifies the previous point entered. This is useful especially if you want to break an entity at an intersection point with another entity:

```
> BREAK  
Select entity (line, circle, arc, 2D-polyline):  
Second point (or F for first point): F  
First point: int  
of object:  
Second point: @
```

---

**See also:**

**DELPARTIAL**

## "C"

Command **CALLOUT**

### ■ Command **CALLOUT**

**Alias CA**

**Functional in: AllenCad CE Pro YES**

**AllenCad CE YES**

**Located:**

- 5. Draw Menu > Markup> Insert Callout**
- 6. Left Side Bar**
- 7. Palette Draw**
- 8. Enter Command or Alias Directly at the Command Line**

The **CALLOUT** command offers an easy way to place boxed markup text in your drawing, that has an arrow pointing to a reference object. All of this can be accomplished with a single **CALLOUT** command.

After issuing the CALLOUT command, you are asked to specify the arrow (point) location and then the other end of the arrow. You are then prompted to insert your text in the dialog box presented, after which the command terminates and the Callout is placed on your drawing.

Callouts are inserted on the current layer in red to conform with redlining convention. Color can be changed as necessary afterwards, using the PROPEDIT command.

## Chain



**Command: CHAIN**

**Alias CN**

### Located:

1. **Pull-down Menu: Draw - Chain**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The Chain command is a combination of the commands Line and Polyline.

The Chain command makes it possible to draw continuous lines in a manner similar to the command LINE using the option *Continuous*. The procedure is the same as drawing continuous lines.

But there are some important differences in drawing lines, polylines, and chains:

- Firstly, when you draw a continuous line, each different segment is a separate element. That way, each element can be chosen separately, using the object selection function.
- With a chain, in contrast, the resulting entity is a single entity, either for editing or for other purposes.
- Second, for a chain, it is possible to choose the line width. This is similar to the command Polyline. The last width value entered for a polyline will be offered as the default width for the next chain, and
- Finally, (differently from a Polyline) it is only possible to choose a single width of line for the entire line when drawing a Chain.

A chain is drawn by describing the points that will be the starting and end point of each chain segment. The end point for each segment will serve as the starting point for the next segment. A single width of the line must also be specified.

```
> CHAIN
First point: P1
Next point: P2
Next point: P3
...
```

You can terminate the Chain command by pressing RETURN or ESC.

### Options:

The following options will be available in the options panel:

#### Width

The option *Width* allows the user to define the width of the entire line. **The width must be specified before entering the first point.** Picking or entering the first point is understood as a verification of the required value.

The previous entered value from either a Chain or Polyline command will be offered as the default

width value for the new chain. Verify this predetermined value by pressing RETURN or enter a new value for the width.

### Undo

The option *Undo* lets the user reverse the last step taken. In other words, the last-drawn chain segment will be erased. The end-point of the previous segment is then used as a starting point of the next segment. Undo's can be repeated as long as segments exist.

### Close

The option *Close* allows the user to close the chain to the start point. The two end points of the chain will be connected to each other and the command will be terminated.

## Chamfer



**Command: CHAMFER**

**Alias CH**

#### Located:

1. **Pull-down Menu: Modify – Chamfer or  
Modify – Edit Polyline – Chamfer Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

When executing the command, the first prompt will ask for the entities to be chamfered. Select those entities by picking them with the cursor.

In the Options Bar, the option **Distance** will appear.

There are two choices: either select, and enter, a distance for the chamfer, or choose to "pick" the distance for the chamfer. The distance back from the point of intersection for the chamfer must be specified for both of the intersecting entities, since there are often non-symmetrical chamfers.

The option **Trim (Off)**, or **Trim**, functions in exactly the same manner as described with the command **Fillet**, as does the option **Polyline**.

The option **Angle** specifies the relative angle that will be applied to the chamfer, a given distance back from the intersection, on object 1.

---

### See also:

FILLET

Change



**Command: CHANGE**

**Alias CV**

#### Located:

1. **Pull-down Menu: Modify – Change**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Change** permits the editing of entities including lines, circles, arcs, text, attributes and

parts with regard to their geometrical character or their attributes.

The command is called by typing the command **CHANGE** or by selecting **Change** from the *Modify* menu.

Select the entity to be edited using the functions shown in the Options Bar.

During the changing operation the user may wish to be supported by the preview provided by **Dragmode**. Select this mode from the Options Bar.

The attributes or geometry that can be changed depends on the entity. In the following paragraphs is a summary of the possible operations, listed by the type of entity

### **Line**

One of the two ending points of the line may be moved to a new position, while the length of the line remains the same.

If the end point of a line is moved without using **Dragmode**, the point to be moved must be the point nearer the selection point. If **Dragmode** is activated, the point lying nearer to the cursor position will be moved. By changing the cursor position the end point of the line to be moved can be changed.

### **Circle**

**Change** reference geometry can only alter the diameter or radius of a circle. After the object selection, determine a point by either "picking" it, or by coordinate input. The diameter or radius will be enlarged or reduced accordingly.

### **Arc**

Arcs may be changed as were circles.

### **Text**

To change text, these attributes, or parameters, may be changed. Existing values are displayed as parameters and can be taken over by pressing RETURN.

- Insertion point of the text object
- Text style
- Text height
- Rotation angle
- Text string (character sequence)

### **Attribute definition**

These attributes can be changed;

- Inserting point;
- Text style;
- Text height; and
- Rotation angle

These are specific characters of those attributes that may lend themselves to change:

- Name
- Request
- Parameter value

The constant or variable (parameter) value of an attribute can be changed. **Value** is that part of the attribute that is written, together with the part, into the drawing. It may either name the part or contain information about the part. Enter new values (sequence of characters or numeric input).

You find detailed information to the characters of attributes in the section *Parts and Attributes*.

### **Part**

The user may change the insertion point and/ or the rotation angle of a part. The existing values are displayed as parameters and can be repeated by pressing RETURN.

- Insertion point of a part

Rotation angle

**Dragmode**, with its dynamic preview capability, is automatically activated during the change of this attribute.

See also: PROPEdit and CHPROP

## Change Properties (Command Line Version)

**Command:** CHPROP , QPROPEdit

**Alias** CHP, QPR

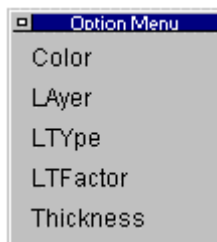
**Located:**

### 1. Entered Directly at the Command Line

These commands modify the properties of drawing objects.

They are the command line variants of the PROPEdit command.

When you issue the command, you will be asked to select objects, and be presented with the object selection dialog to help you make this selection. Next, you will be prompted at the command line to identify which properties you want to change. A dialog appears (as below) on screen to help you enter your selection.



After making your selection, you will be asked to enter a new value for the property you want changed.

As with most commands, command line input can be used, instead of using the dialog. You simply type in the option abbreviations as are shown bold in the following text screen recording. The requests of these commands are as follows:

```
> CHPROP
Select objects:
Specify property to change (Color, LAyer, LTtype or Thickness):
```

---

**See also:**

PROPEdit

Circle



**Command:** CIRCLE

**Alias** CI

**Located:**

1. **Pull-down Menu:** Draw - Circle
2. **Palette:** Draw
3. **Or Enter Command or Alias at the Command Line**

There are several ways to describe a circle, geometrically. The program allows the user to choose from among these ways, and draw the circle in the most convenient manner.

A circle can be drawn by choosing one of the following options:

1. Center of the circle and radius or diameter;
2. Two Points on the circumference (that define the diameter and center point)
3. Three Points on the circumference
4. Two Tangents

These options will appear on the *Options Panel* when the command circle is active.

## Options

### Center

Since the *Center Point* option is the most often chosen, it has been selected as the default option and is activated automatically if no other option is chosen. The first prompt for this option will be the location of the center point; the second will ask for the radius. If the circle being drawn is preceded by another circle, the last radius will appear at the prompt as the default radius value. To verify the default, just press RETURN, or to change the option, enter or "pick" a new radius.

If it is more convenient to enter the diameter than the radius, the user may choose that option from the options panel.

### 2P

If the circle is to be tangent to two other lines, this option is handy.

The prompt will ask for two points in succession. Again, the points may be picked using the object snap options, or may be entered from the keyboard. The mid-point between the picked points will be the circle center and the distance between the points, the circle diameter.

### 3P

This option allows the user to draw a circle by choosing three points on the circumference of the circle. It is convenient to use when the circle must intersect three tangent lines, or when drawing the circle within a polygon.

The prompt will ask for three points successively. The user may point to these points on the screen, perhaps using the option **object snap**, or enter the coordinates through the keyboard.

### Radius

This geometry is particularly useful when the radius of the circle is a known, and the center point is undefined, or when the radius follows from geometry already established.

You are first asked to pick two points that define the circle radius length. These options are for entering the radius:

- Verify the default radius value (the radius of the last circle drawn) by pressing RETURN;
- Enter a new value for the radius manually; or
- Pick the radius on the screen. This option is especially useful when the radius follows from the geometry and only has to be copied.

Next, you are asked to locate the circle center point.

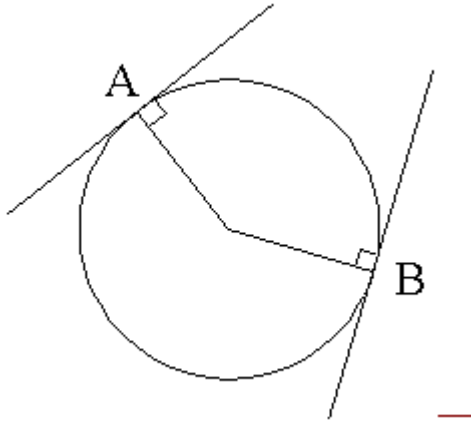
The circle is drawn once the center has been located, however the command assumes that you may wish to draw another circle with the same radius and it asks for another center point. This continues until the command is terminated by pressing Esc, Return, or selecting another command.

### Diameter

The use and the implementation of this option follows exactly the use and implementation of the option **Radius** except that instead of choosing the radius of the circle, the user must specify the diameter.

## Tangent

This option is frequently useful, as it will draw a circle of given radius or diameter **tangent to two lines**.



The first prompt will ask for the objects that the circle is to be tangent to. These objects must exist in the drawing before this command is issued. Pick both objects on the screen. In the example shown, we pick the two lines.

After choosing the second object, enter the radius of the circle to be drawn. The radius of the last circle drawn is offered as default by the program. Verify this choice by pressing RETURN or enter a new value manually, or by "picking" it on the screen.

The circle will be placed so that it touches, tangentially, the two chosen elements with its circumference. In other words, the line from the circle center to the intersections is at right angles to the objects. This is illustrated in the example above, where points A and B are the points of calculated tangency.

If the chosen radius is smaller than the distance of the two objects to each other, the circle will be moved in the direction of the imaginary point of intersection, so that the defined radius will touch the extension of the elements. Note: the original objects are not extended. The extensions are only calculated to determine circle location.

If a tangential intersection is impossible (parallel lines, two concentric circles or arcs as chosen objects, etc.), the program gives the following message: "Circle does not exist!"

**Note:** The "smoothness" with which the circle is represented onscreen is controlled by the variable CIRCLES. By issuing the SETTINGS command, users can go to the Draw Modes tab and enter a new value (between 8 and 1024) in the Arc Segments box that controls smoothness. A low value will result on a rough arc or circle, while a higher value will produce a smooth arc or circle.

## Close a Drawing



**Command: CLOSE**

**Alias CL**

**Located:**

1. **Pull-down Menu: File - Close**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**



Closing a drawing indicates that the work on this drawing is finished. The program remains open. If a drawing has been changed before closing there will be a prompt asking whether the file should be saved. This is to prevent unintended loss of work.

If the user decides to save the drawing, the program will automatically switch to the standard routine to save the drawing under a new name.

Select the command CLOSE in the menu *File* or press the keys CTRL+F4. It is also possible to close the drawing window, as in all Windows programs, by double clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

---

**See also:**

OPEN  
NEW  
SAVE  
SAVE AS...  
SAVE ALL

## Configuration

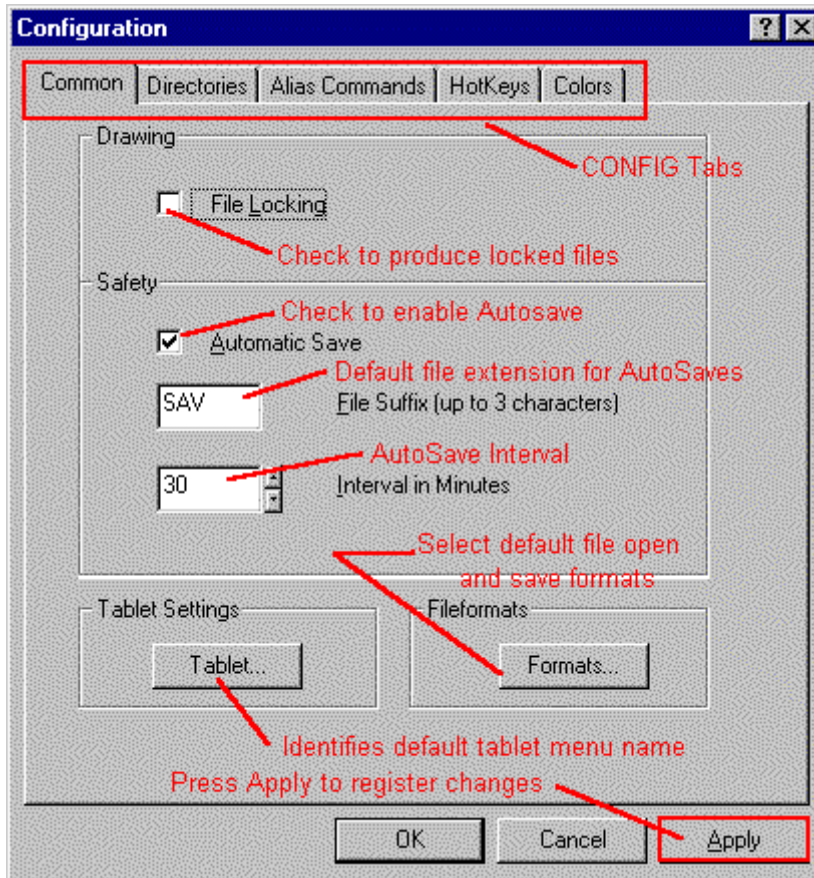


**Command: CONFIG**  
**Alias CFG**

**Located:**

1. **Pull-down Menu: Tools – Configure**
2. **Or Enter Command or Alias Directly at the Command Line**

By calling the CONFIG command or selecting Configure on the File pull-down menu, the Configuration dialog will appear.



As can be seen, this dialog contains five tabs, each presenting separate dialogs under the headings: Common, Directories, Alias Commands, Hotkeys and Colors.

Note: Changes made in these dialogs will only become effective after the APPLY button has been selected. No changes will become effective if you close the dialog window by pressing OK, the Cancel button or ESC key.

## Copy



**Command: COPY**

**Alias C**

### Located:

1. **Pull-down Menu: Modify – Copy**
2. **Top Function Bar**
3. **Palette: Modify**
4. **Or Enter Command or Alias Directly at the Command Line**

### Duplicating drawing objects

This function allows the user to copy drawing elements to AllenCad's clipboard and later put them at a specified place.

The copies created by this command are exact copies of the selected entities. All entity attributes such as layers, line type, color, and all other attributes are copied as well as the outline of the object(s).

### Options:

## Basepoint

If **Basepoint** is the selection, enter the base point, then, as prompted, the **Target Point** (the position to which the Copy is to move). The copy will be made in the position specified.

## Rel.Point

**Rel.Point** may be chosen if the copy is to be placed in a position relative to the current image of the object.

The prompt will ask for the relative points by specifying the move in the X, Y, and Z directions. If the move is to be made in the X and Y directions only, simply do not specify a Z delta. The copy will be placed in the specified position.

## Copy Object Properties



**Command: COPYPROP**

**Alias CP**

### Located:

1. **Pull-down Menu: Edit – Copy Object Properties**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

This command helps you to change the properties (layer, linetype and color) of selected objects to those of a selected entity.

### To change the properties of a selection:

1. Select a single reference entity.
2. Select one or more objects to be changed.

### Command line example:

```
> COPYPROP  
Select entity for reference properties:  
Select objects to modify. Select objects:
```

## Command COPYSELECT



**Command: COPYSELECT**

**Alias CST**

### Located:

1. **Pull-down Menu: Edit – Copy Selection to Clipboard**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

The command COPYSELECT copies selected objects from the drawing, placing them onto a **AllenCad-specific** clipboard. The selected objects remain in the drawing.

The command COPYSELECT is useful especially to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

### To copy objects onto the clipboard:

1. First specify the base point for later insertion of the objects.
2. Then select the objects to be copied to the clipboard.

### Command line example:

```
> COPYSELECT  
Insertion base point:  
Select objects:
```

### Note:

Do not confuse this command with the COPY command found in the *Modify* menu that makes copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another drawing. They will remain on the clipboard until you place new objects on it with the COPYSELECT command.

---

### See also:

Command PASTE  
Command CUTSELECT

## Cut Selection to Clipboard



**Command:** CUTSELECT

**Alias** CT

### Located:

1. **Pull-down Menu:** Edit – Cut Selection to Clipboard
2. **Top Function Bar**
3. **Palette Edit**
4. **Or Enter Command or Alias at the Command Line**

With the command CUTSELECT selected objects are copied to a **AllenCad-specific** clipboard (as with the command COPYSELECT)

However, unlike COPYSELECT, with CUTSELECT, the originally selected objects are deleted from the drawing.

The command PASTE will insert the objects from the PCAD clipboard into any open drawing. PASTE will request the insertion point.

---

### See also:

Command COPYSELECT  
Command PASTE  
Command DELETE

## "D"

Delete



## **Command: DELETE**

**Alias DEL**

### **Located:**

- 1. Pull-down Menu: Edit – Delete**
- 2. Top Function Bar**
- 3. Palette: Edit**
- 4. Or Enter Command or Alias at the Command Line**

During the creation of drawings, or, later during the editing, changing or detailing of drawings, it may be necessary to delete certain parts of the drawing.

After calling the command, the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for object selection. These same options will alternately appear in a dialog menu, depending upon your preferences specified in Desktop Configuration.

WPolygon	CPolygon	Fence	Last	Previous	Remove	Window	Crossing	Single	All
----------	----------	-------	------	----------	--------	--------	----------	--------	-----

When deleting, as many objects can be selected as the user desires. When the object selection is complete, the selected entities will be deleted without another confirmation, by simply pressing ENTER.

---

### **See also:**

Command UNDELETE

## **Delete Lock-Files**

### **Command: DELFLK**

**Alias DLK**

### **Located:**

- 1. Pull-down Menu: File – Delete Lock Files**
- 2. Or Enter Command or Alias at the Command Line**

The program contains a function for deleting the so-called lock-files, called from the menu File, or entered by typing the command DELFLK.

Lock-Files are security files created by the program when a drawing file is opened. They carry the same name as the original file and can be identified by the extension .flk. Lock-Files will be deleted automatically, if the drawing files are closed in the normal manner.

This procedure avoids two or more users working on one drawing at the same time when in a networking environment.

If the program is unexpectedly interrupted and the files have not been closed before the interruption, the same situation will occur. If the user tries to re-open a file after an interruption he or she will get the message file locked.

In order to re-gain access to the drawing, the corresponding Lock-Files must first be deleted.

In order to delete these files, first enter the command. Next, the user must perform the standard steps for the selection of files (similar to the steps in opening files). The program will offer the extension .flk as the desired file type.

Delete the offending Lock-File, and the drawing file can again be opened.

## **Delete Partial**



**Command: DELPARTIAL**

**Alias DPL**

**Located:**

1. **Pull-down Menu: Modify – Delete Partial**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Delete Partial** will cause a part of a line, circle, or arc to be erased.

These entities can also be **divided** by this command. The result of such a division will be two independent entities that can be edited, or have another operation performed on it, separately.

#### **Deleting Part of a Line**

Deleting part of a line is the standard option for this command. to perform this deletion, determine the end points of the part of the line to be deleted. The line will be divided at this point, and the portion of the line between the two points will be deleted.

#### **Deleting Part of a Circle or Arc**

If the entity is a **circle**, you have only the two-point option available that removes a section of the circle circumference, as a 360o arc is not supported. It must therefore be some angular dimension less than 360o. Arcs, on the other hand, can be either broken into two at a specified point (the one-point option), or else a section can be removed, also leaving two arc objects (the two-point option). With arcs or circles, removal of a section is measured counter anti-clockwise from picked points 1 and 2.

---

**See also:**

BREAK

Desktop Configuration

**Command DESKCONF**

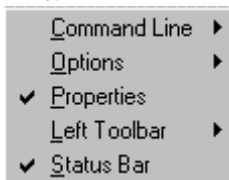
**Alias DSK**

**Located:**

1. **Pull-down Menu: Tools – Desktop Manager**
2. **Or Enter Command or Alias Directly at the Command Line**

## **Modifying the Desktop Layout**

The DESKCONF command's pop-up menu enables you to re-arrange the desktop to suit your needs. This pop-up menu allows you to set various layout options for the programs user interface, including the display of the Command Window, Options Menu or Options Bar, Property Bar, Left Sidebar, and Status Line.



**The Desktop Layout pop-up menu**

**Caution!** The following sections describe how to turn off desktop elements. The user should exercise caution in doing so, however, as these actions will remove visual

references to command inputs. If all desktop elements are turned off and the user wishes to reinstate them, the DESKCONF command will need to be retyped and executed with the relevant options selected. Pressing F2 will call the text window where the user can see the command prompt, as needed. Likewise, online Help can be called as needed, by pressing the F1 key.

#### Turning desktop items on and off:

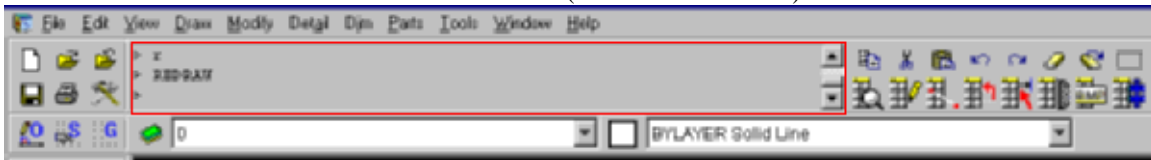
To achieve more space to display drawings on the desktop, or as an advanced user, who is familiar with the names and options of the commands, you may want to turn off certain items of the desktop. You can turn off and on (at runtime) the display of the command line, the Options Menu or context toolbar, the property bar, the left toolbar, and the status bar as the figure above shows.

#### How to open the *Desktop Configuration* pop-up menu:

You can call the *Desktop Configuration* menu by typing the DESKCONF command or selecting the Desktop Configuration selection on the Tools pull-down menu.

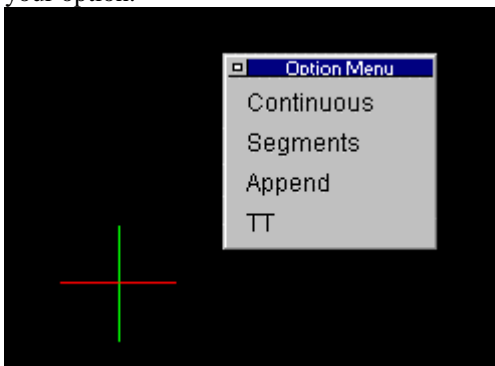
You can also open the *Desktop Configuration* pop-up menu by clicking the right mouse button in a "non-functional" area of the program's desktop. Such area is either the background area upon where the drawing viewports are displayed or areas in the property bar, Options Bar, or toolbars that are not covered by buttons or drop-down lists. If necessary, reduce the size of the current drawing(s), cascade the drawing(s), or minimize the drawing(s) to get some free space of the drawing area background. Right-click to this area to open the *Desktop Configuration* menu. You exit this pop-up menu by left clicking to some other area or component on the desktop.

The **Command Line** can either be placed at the top of the screen (below the menu bar as pictured below), or in the default location at the bottom of the screen (above the status line).



Where you place the bar is strictly a matter of personal preference, as there are no specific advantages in choosing either location. If you are an advanced user, who is familiar with the program's command set, you might even turn the command line off to provide a larger drawing edit window. In this instance, you can always check for program messages or inputs by toggling the text window on/off using the F2 key.

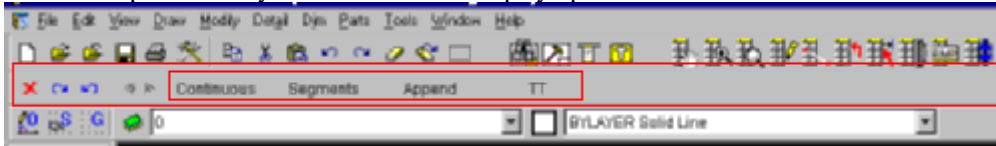
The items of the **Options** sub-menu allow you to determine whether you want to be prompted for command options via an *Options Dialog Menu* or within a *Context Toolbar*. By default, command options are displayed in a pop-up menu that appears in a dialog window in the edit window. These command option dialogs (as pictured below) will disappear or change to offer new options, once you have selected your option.



An Options Dialog Menu - The benefits to this option are that it places the options right on the editor

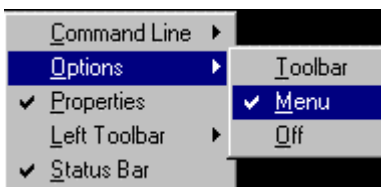
where you are working, and that it doesn't reduce the overall size of your drawing edit window. The drawback is that you sometimes may have to move (drag and drop) the dialog if it is in your way as you draw.

If you choose to display options on an *Options Bar*, you will see a new toolbar inserted between the horizontal toolbar and the Properties Bar, as pictured below. This toolbar will remain open to display command options until you choose another display option.

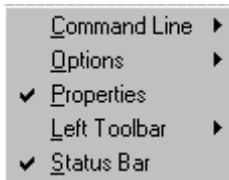


The Options Bar - Note the options presented on the bar.

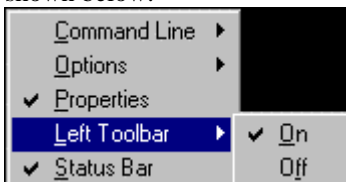
Note: You can recall the display of the context pop-up menu by selecting the option *Options>Menu* from the *Desktop Configuration* pop-up menu.



Both the **Properties Bar** and the **Status Bar** can be toggled off and on by clicking on the selection in the Desktop Configuration dialog. When toggled on, the selection will appear with a check mark as shown below:



The **Left Sidebar** can be switched off and on by clicking on the on or off options that appear in a secondary window when you click on the Left Toolbar option in the Desktop configuration dialog, as shown below:



See Also: Emulating the AutoCAD Desktop

## Command DIGICONFIG

### Command DIGICONFIG

#### Alias DC

#### Located:

#### 1. Enter Command or Alias Directly at the Command Line

To digitize existing drawings, you need to have two defined coordinate locations on your drawing. Place your drawing on the tablet and tape it down so it cannot move. Enter the DigiConfig command. The



program asks for the value of two coordinates and then asks you to pick the corresponding location on the plan.

You turn on and off the digitizing function with the DIGION and DIGIOFF commands. The program has a digitizer palette called the "Digitize Paper Drawings" that can make the process easier.

## Command DIGIOFF

### Command DIGIOFF

#### Alias DGF

#### Located:

1. Entered Directly at the Command Line

The DIGIOFF command switches the digitizer from Digitizer mode to Tablet mode after you have digitized a paper drawing. See also Command DIGICONFIG.

## Command DIGION

### Command DIGION

#### Alias DGN

#### Located:

1. Entered Directly at the Command Line

The DIGIOFF command switches the digitizer from Digitizer mode to Tablet mode after you have digitized a paper drawing. See also Command DIGICONFIG.

## Angle Dimensioning (3 Points)

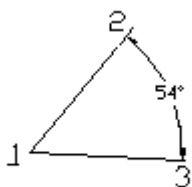


### Command: DIMA3P

#### Alias D3P

#### Located:

1. Pull-down Menu: Dim – Angular 3 Points
2. Palettes: Create Dimensions and Dim
3. Or Enter Command or Alias Directly at the Command Line



Angular- 3-Point

This function uses 3 measurement points that are placed at the apex and legs of an angle. This function not only allows the user to calculate the interior angle between the two legs, but also the exterior angle.

The user will be asked to select the vertex position. Either type in the coordinates, or select with the cursor. It is suggested that the user use the Object Snap options whenever applicable.

Define a point on each of the legs of the angle.

Define the location of the dimensioning arc by selection with the cursor. The dynamic preview function will assist the user in gaining a visual impression of the result. The position chosen for the dimensioning arc will also decide that angle, (interior or exterior) is to be dimensioned.

After confirmation, the program will display the measured value. This can be confirmed with RETURN, or amended as required.

```
> DIMA3P
Identify vertex: end
of object: P1
First point: P2
Second point: P3
Position dimension arc: P4
Dimensioning text <60>: RETURN
```

## Command DIMA4P



**Command: DIMA4P**

**Alias D4P**

**Located:**

1. **Pull-down Menu: Dim – Angular 4 Points**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias at the Command Line**



As with the DIMA3P command, DIMA4P creates an angular dimension on the drawing, but with different inputs. With this command, you pick two objects that define the angle being measured and detailed.

This command is found on the Dim – Angular 4 Points... pull-down, on the Dimension palette, or it can be entered at the command line.

Command input sequence is as follows:

```
> DIMA4P
Select line, polyline, or arc:
Select line or polyline:
Position of dimension arc: [Here you indicate where the notation is to be made]
Dimension text <71>:[Here you can modify the note as required – 71o has been measured in
this example]
```

## Command DIMALI



**Command DIMALI**

**Alias DAL**

**Located:**

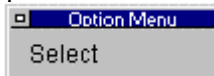
1. **Pull-down Menu: Dim – Aligned**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias at the Command Line**



Aligned dimensioning is a type of linear dimensioning that measures and labels the absolute distance between two points, regardless of the relative position of the axes. This makes it possible to dimension lines, edges and spaces etc. that are not located on the principal axes of the coordinate system (IE. not parallel to x, y or z-axes).



The user will be asked to define two points of measurement (DEFPOINTS). As an alternative the program offers the user the chance to select a specific dimensioning object function. Activate this function by



choosing Select from the options bar. This allows lines, polylines, circles and arcs to be selected. Point to the desired object with the cursor. In the case of lines, polylines and arcs the start and end points of the object will be interpreted automatically and used for the measurement calculations.

The first point of a circle will be defined as a point situated along the perimeter of the circle. The second point will be obtained by drawing a straight line from the first point, through the center point of the circle, to the opposite side of the perimeter of the circle.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line. This can be done either by entering the relevant coordinates, or with the cursor.

The absolute measurement between the two points will be displayed. Pressing the RETURN key will accept the calculated value. The dimension will be inserted into the drawing and shown on a horizontal plain.

> DIMALI

1st Point: P1

2nd Point: P2

Dimension line positioning: P3

Dimension text <2.5000>: RETURN

Or

> DIMALI

st Point: SELECT <CR>

select line, polyline, circle, arc: P1

Dimension line positioning: P2

Dimension text <2.5000>: RETURN

## Baseline Dimensioning

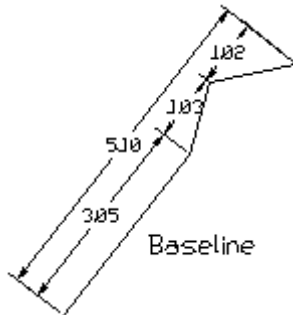


**Command:** DIMBAS

**Alias** DBA

### **Located:**

1. **Pull-down Menu:** Dim – Baseline
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**



**Baseline** adds a second measurement line to the drawing, parallel to the already existing dimensioning lines. This allows the user to read the total dimension between two end points, extending over a number of individual dimensions within a chain.

This type of dimensioning assumes that a form of linear dimensioning already exists. The appendices dimension will inherit the same properties as the existing measurements.

With the cursor, mark the starting point of the dimensioning chain, taking care to utilize the same direction of measurement as laid down during the creation of the individual dimensions within the chain.

Select the second point to be used for the baseline dimension by entering the coordinates at the keyboard or with the cursor. Whenever possible enlist the help of OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as this second line will automatically be drawn at a given distance from the existing dimensioning lines.

```
> DIMBAS
Select a dimensioning line: P1
Second Point: P2
Dimensioning text <20.50>: Return
```

## **DIMCENT**



**Command:** DIMCENT

### **Located:**

1. **Pull-down Menu:** Dim – Center
2. **Palettes:** Create Dimensions and Dim
3. **Or Enter Command or Alias Directly at the Command Line**

This command places a cross at the radius point (center) of selected arcs or circles.

## Continuing Dimensioning

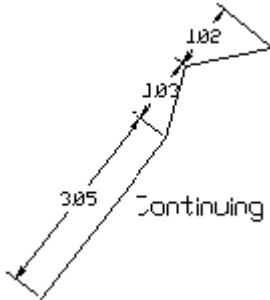


Command: DIMCON

Alias DCN

### Located:

1. Pull-down Menu: Dim – Continuing
2. Palettes: Create Dimensions and Dim
3. Or Enter Command or Alias Directly at the Command Line



**Continuing** dimensioning is a variation of linear dimensioning in which existing dimensioning is continued or extended. Here a second dimension is linked to an existing dimension so as to create a dimensioning chain. Continuing dimensions can only be created, when a drawing already contains an existing linear dimension that can be used as a basis for the chain.

A continuing dimension always inherits the same properties as the existing dimensioning. For this reason it is only possible to connect dimensioning of the same type. For instance, a new horizontal dimension can only be added or linked to an existing horizontal dimension.

A message will appear in the text window requesting the selection of an existing dimension. Select the desired dimension, taking care to place the selection cursor as close as possible to the point at that the continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the continuing dimension will be the point that lies closest to the selection point at the time the object is selected

Choose the second point by entering the relevant coordinates, or by selecting with the cursor. Whenever possible, employ the object snap-to function.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as this will automatically be placed in line with the existing dimensioning.

```
> DIMCON
Select a dimension: P1
2nd Point: P2
Dimension text <2.50>: RETURN
```

## Diameter Dimensioning



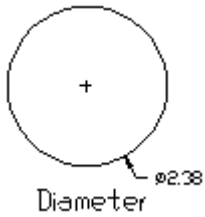
Command: DIMDIA

Alias DDA

### Located:

1. Pull-down Menu: Dim – Diameter

2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Diameter dimensioning is used to display dimensions of circles and arcs. Unlike the horizontal or vertical dimensioning functions, no measurement lines are shown. In this case a single extension line is used to show the relationship between the value and the object itself. The diameter symbol signifies that this is a special dimensioning feature used only in circle and arc dimensioning.

The first prompt will ask for the specific circle or arc. Select with the cursor. Please note that the point at that the user selected the object will be defined as the point at that the dimensioning extension line will connect to the element.

QUADRANT OSNAP will help the user to select specific points on the circle or arc. After selection, the program will display the corresponding diameter measurement. This can be confirmed with the RETURN key or amended as required.

The second prompt will ask for the position of the extension line. Select a coordinate at the required distance from the object

The alignment of the extension line will have been fixed by the position of the two selected points and cannot be altered. The extension line will be drawn along an imaginary straight alignment that extends from the middle point of the object to the measurement point as defined during object selection (DEFPOINT).

The extension line and measurement text will be displayed and a cross (+) will automatically appear at the center of the circle or arc.

```
> DIMDIA
Select circle or arc: P1
Dimensioning text <24.00>: RETURN
Extension length: P2
```

## Command DIMDRAW

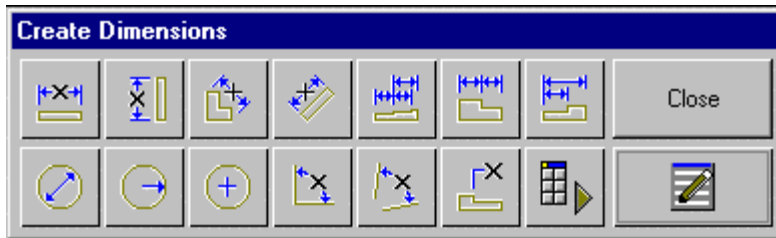


**Command DIMDRAW**

**Alias DDR**

### Located:

1. **Pull-down Menu: Dim – Dimensioning**
2. **Left SideBar**
3. **Palettes: Detail and Dim**
4. **Or Enter Command or Alias at the Command Line**



This command calls the toolbar shown above, which is not unlike the Dimension palette, in function. As with all command icons, holding the cursor above the icon for a second or so, will produce a "tool-tip" that identifies the command function.

Where the Dimension palette remains on the desktop for repeated use, however, the Dimdraw palette is only available for a single command and must be re-called if needed again. Pressing Enter immediately after the palette command has completed, will re-call the Dimdraw palette, however. When placing dimensions on the drawing, you can select the appropriate dimension command by clicking on the associated toolbar icon.

## Command DIMEDIT

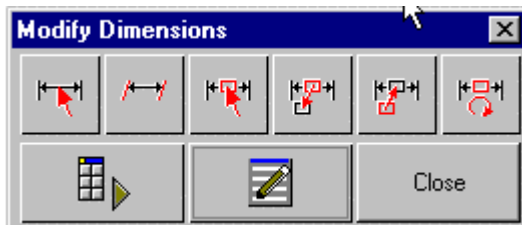


**Command DIMEDIT**

**Alias DED**

### Located:

1. Pull-down Menu: Dim – Modify Dimensions
2. Left SideBar
3. Palettes: Modify Dimensions and Dim
4. Or Enter Command or Alias at the Command Line



The DIMEDIT command produces this palette containing icons for commands used to modify dimension entities. The palette remains onscreen for a single selection only and the command is repeated for subsequent modifications.

As with all command icons, holding the cursor above the icon for a second or so, will produce a "tool-tip" that identifies the command function.

## Horizontal Dimensioning



**Command: DIMHOR**

**Alias DH**

### Located:

1. Pull-down Menu: Dim – Horizontal

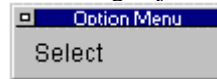
2. **Palette: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Draws horizontal dimensions. Horizontal dimensioning is linear dimensioning that both measures and labels the distance between two points parallel to the X -axis.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button . Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the x-axis and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMHOR
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMHOR
1. Point: Select
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: Return
```

## Linear Dimensioning



**Command: DIMLIN**

**Alias DL**

### **Located:**

1. **Pull-down Menu: Dim – Linear**
2. **Palette: Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



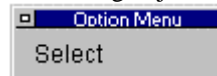


Linear

The DIMLIN command creates a horizontal or vertical dimensioning depending on the position of the dimension line. This command is a combination between both orthogonal modes horizontal and vertical.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points is measured along the X-axis for horizontal and Y-axis for vertical dimensioning and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMLIN
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMLIN
1. Point: Select
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: Return
```

## Modify Dimension Orientation



**Command: DIMOBL**

**Alias DOB**

### Located:

1. **Pull-down Menu: Dim – Modify Orientation**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to modify the orientation of existing linear dimensions.

Call the command and enter the angle for the dimension orientation. Select one or several dimensions, to be modified in its orientation.

## Ordinate Dimensioning

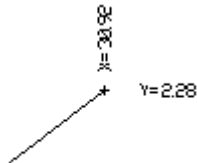


**Command: DIMORD**

**Alias DOD**

**Located:**

1. **Pull-down Menu: Dim – Update Dimension**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



**Ordinate**

Ordinate can be used to determine the X and Y coordinates of a point and enter it into the drawing. These coordinates can either represent a point within the drawing, or be that of a geometrically significant point of a drawing entity.

The X and Y options will be displayed within the options bar. By selecting one of the options the user can determine whether the X or Y coordinate value is to be defined. Click the corresponding button, bearing in mind that the X value is set as the default and will therefore automatically be used if no other option is selected.

The user will then be prompted to select the point to be measured. Type in the coordinate values, or select with the cursor. If the location represents a specific point on an entity, enlist the help of OSNAP.

The program will ask for LEADER LENGTH; this requires that the user enter the EXTENSION LINE position and also determines the physical location of the measurement value text. The extension line originates at the point to be measured and connects to the measurement value text.

Use the cursor to select the position at that the measurement value is to be displayed. This figure can be positioned at any alignment and distance from the dimensioning line. To help, the program draws a line from the selected coordinates to the current cursor position. This illustrates the distance and alignment between the two positions. Select the desired position with the cursor.

If the dimension is not placed horizontally or vertically in relation to the measurement point, then the extension line will appear in orthographic mode and not as a straight line. An example of this is shown in the above diagrams.

The program will display the calculated coordinate. The value can be confirmed by pressing the RETURN key, or amended as required. The user may type in any alphanumeric sequence, thus enabling the point to be titled as required.

```
> DIMORD
Coordinate point: X
Coordinate point: P1
Extension length: P2
Dimensioning text <20.00>: RETURN
```

**Radius Dimensioning**

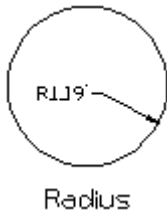


**Command: DIMRAD**

**Alias DRA**

**Located:**

1. **Pull-down Menu: Dim – Radius**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Radius dimensioning is carried out in the same way as diameter dimensioning. The only difference is that a radius dimension is calculated instead of a diameter measurement.

```
> DIMRAD
Select circle or arc: P1
Dimensioning text <24.00>: RETURN
Extension length: P2
```

## Restore a Dimensioning Style



**Command: DIMREST**

**Alias DRE**

**Located:**

1. **Palette: Dim**
2. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to reinstate a stored style file so that it can be reinstated as the current style. All of the stored style settings will be retrieved and used by the system for future dimensioning. Any dimensions that were created prior to the definition of the new style can also be updated (DIMUPD) so that they, too, adhere to the conventions of the new style settings.

This is done by using the DIMUPDATE command, explained in the following section.

Restoring a dimensioning style is achieved up by entering the DIMREST command.

The program will follow this command by displaying the name of the current dimensioning style in the text window. The user will then be requested to choose the dimensioning style that is to be used as the current default. The program options bar offers the user two methods by that to select the required dimensioning style.

- Either type in the name of the stored dimensioning style, remembering that the user can list all of the dimensioning styles already contained within the particular drawing; or
- Use the cursor to select the desired dimensioning style.

### Select the Style by Typing the Name.

Enter the style name at the appropriate input field. By clicking the list button shown in the options bar the user can display all of the available dimensioning styles.

In order to achieve a clearer overview, it is possible to list all of the file names that begin with a particular character chain by entering the chain at the input field. The standard DOS default \* can also be used to substitute a complete character chain.

When used in a search routine this will cause all of the existing file names to be listed. If the user wishes to search for specific dimensioning style files, type in the desired character chain followed by the \* character. For example, ar\* would result in a list of all files beginning with AR.

```
> DIMREST
Current dimensioning style: Architecture03
Name of the dimensioning style: List <?>
List dimensioning styles <*>: arch*
ARCHITECT01
ARCHITECT02
ARCHITECT03
```

Listing the existing dimensioning styles will complete the **Save Dimensioning Style** function. Pressing RETURN will restart the option and save the current dimensioning settings.

```
> DIMREST
Name of the dimensioning style: Architektur03
```

### Selection by Pointing

After the program has displayed the current dimensioning style, choose the **SELECT** option field and choose the desired dimensioning style with the cursor. The program confirms the user selection and verifies it by displaying the new dimensioning style in the text window.

```
> DIMREST
Current dimensioning style: Architecture
Name of the dimensioning style: Select
Select a dimension: P1
Current dimensioning style: Arch_Detail
```

---

### See also:

#### Command DIMTYPE

### Rotated Dimensioning



Command: DIMROT

Alias DRO

#### Located:

1. Pull-down Menu: Dim – Rotated
2. Palettes: Create Dimensions and Dim
3. Or Enter Command or Alias Directly at the Command Line



Rotated

Rotated dimensioning is also a type of linear dimensioning that measures the distance between two points, at a specified angle.

When dimensioning the distance between two points (or an object with linear extension) at an angle

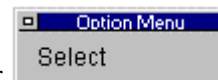
greater or less than 90°, the space allocated to the dimension will always be smaller than the absolute dimension measurement. This procedure is equivalent to looking at an object from a specific angle and establishing the visible length from a different viewpoint.

The user will be asked to define the angle at that the dimensioning is to appear in relation to the axial alignment of the coordinate system. This can be done by entering the coordinates at the keyboard, or by using the cursor to click on two points.

The first figure entered defines the angle between the direction of the negative Y-axis and the first dimension line as seen in a clockwise direction. The apex of this angle is determined by the position of the first point of measurement.

If the user defines the angle at that the dimensioning is to take place by selecting two separate points, then a perpendicular line will be drawn at 90° to the extension lines.

After having determined the dimensioning angle, the user will be asked to define two points of measurement (DEFPOINTS). As an alternative, the program offers the user the chance to select a specific



dimensioning object by clicking the Select function from the Options Bar. This will allow lines, polylines, circles and arcs to be selected. Point to the desired object with the selection cursor.

The start and end points of a line, polyline or arced object will be interpreted automatically and used for the measurement calculation.

The first point of a circle will be defined by the program as the next point situated along the circle perimeter. The second point is obtained by drawing a straight line from the first point, through the center intersection of the circle and on to the opposite side of the circle perimeter.

In contrast to objects that have a linear extension, this method will always measure and display the exact diameter of a circle. Only the position of the dimensioning extension lines will be affected when using rotated dimensioning.

Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line.

The distance between the two points will be displayed at the designated angle orientation. Pressing the RETURN key will accept the calculated value and cause it to be inserted horizontally into the drawing.

```
> DIMROT
Angle of rotation <0>: 60
First Point: P1
Second Point: P2
Dimension line positioning: P3
Dimension text <2.50>: RETURN
```

or

```
> DIMROT
Angle of rotation <0>: 60
First Point: Select
Select line, polyline, circle or arc: P1
Dimension line positioning: P2
Dimension text <2.50>: RETURN
```

## Command DIMSAVE



**Command: DIMSAVE**

**Alias DS**

**Located:**

1. **Palette: Dim**

## 2. Or Enter Command or Alias Directly at the Command Line

The DIMSAVE command allows the user to combine the individual settings or parameters and store them in a named dimensioning style. This facility enables the user to store and recall a number of different styles, each of that can be used for an individual working environment.

Architectural dimensioning, tolerance dimensioning or country and language specific options are examples of this (such as for individual units of measurement or labeling).

The use of pre-defined dimensioning styles removes the need to constantly change the dimensioning settings, simplifies the actual drawing procedure and in general makes the job of dimensioning easier.

It should be noted that dimensioning styles can only be stored together with the current drawing and are, therefore, only available for use from within that drawing. In order to use pre-defined dimensioning styles in other drawings, they should be created and stored in one of the template drawings, or loaded from a pre-defined prototype drawing.

### Procedure for Saving

Select the command **Modify Dimension** from the menu *Detail*. From the palette, select the icon representing the Dialog Box *Dimension*. Enter the proper settings and values from this dialog box. In the first field in this box, name the dimensioning template with a descriptive name.

Before entering the name of a dimensioning style, it is possible to list the file names that have already been allocated. To do so type a question mark (?) in the input field and press RETURN.

Dimensioning style names may be up to 31 characters long and can consist of any alphanumeric input except special symbols. Up to a maximum of two umlauts can be used in any one name, but these should be avoided if at all possible. No differentiation is made between upper and lower case characters as all entries are automatically converted to upper case.

Confirming the selected name will cause the dimensioning style to be stored.

In order to achieve a clearer overview, it is possible to list all of the file names that begin with a particular character chain by entering the chain at the input field. The wildcard pattern option \* can also be used to substitute a complete character chain. When used in a search routine this will cause all of the existing file names to be listed. If the user wishes to search for specific dimensioning style files, type in the desired character chain followed by the \* character. For example, ar\* would result in a list of all files beginning with AR.

```
> DIMSAVE
Name of the dimensioning style or ? for a complete list: ?
List of dimensioning styles <*>: ar*
ARCHITECTURE01
ARCHITECTURE02
```

Listing the existing dimensioning styles will complete the save dimensioning style function. Pressing RETURN will restart the option and save the current dimensioning settings.

```
> DIMSAVE
Name of the dimensioning style, or ? for a complete list:
ARCHITECTURE03
```

---

See also:

**Command DIMSTYLE**

### Serial Dimensioning

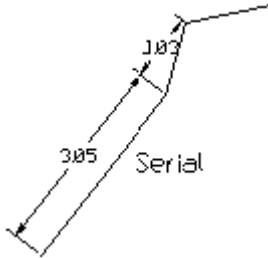


**Command: DIMSER**

**Alias DSR**

### Located:

1. **Pull-down Menu: Dim – Serial**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Serial dimensioning is carried out in the same way as described above for Continuing dimensioning. However, where only one additional dimension can be added to an existing dimension with Continuing dimensioning, Serial dimensioning enables the user to link as many dimensions as required. This assumes that at least one dimension already exists that can be used as a basis for the rest.

It is only possible to add dimensioning of the same type since the additional dimensioning will inherit the properties of the existing dimensioning:

Select the dimensioning that is to act as the basis for the series, taking care to place the selection cursor as close as possible to the point at that the Continuing dimension is to be linked.

The dimensioning is based on two points of measurement. The starting point for the series dimensioning will be the point that lies closest to the click point at the time of object selection.

In addition to Select and End that are the same for Serial dimensioning as for Continuing, two other selections are possible from within the series dimensioning function bar:

#### **Continue and Baseline.**

1. **Continue** allows the user to link one additional dimension and should be selected when creating serial dimensioning. As this is the default option it only needs to be confirmed with the RETURN key. Alternatively, click the appropriate button as displayed in the options bar.
2. **Baseline** allows the user to create baseline dimensioning, parallel to the existing linear dimensioning. This function is described in the next section. Select the second point by typing in the coordinates or selecting with the cursor. Whenever possible use OSNAP.

The program will display the calculated dimension. This value can either be altered, or confirmed with RETURN. The user will not be asked to position the dimensioning line, as it will automatically be placed in line with the existing dimensioning.

```
> DIMSER
Select a dimension: P1
Series <continuous> RETURN
2nd Point: P2
Dimension text <2.50>: RETURN
Series <continuous>: RETURN
2nd Point: P2
Dimension text <2.50>: RETURN
...
ESC
```

By re-selecting the **Continue** option, it is possible to repeat the above steps for as many times as dimensions are required. The creation of additional dimensioning can be terminated by pressing the ESC

key.

## Undo Repositioning



**Command: DIMTHOME**

**Alias DTH**

### Located:

1. **Pull-down Menu: Dim – Modify Dimension Text – Realign DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This command undoes the moving of the dimensioning text (done with DIMTMOVE) and returns it to the original position.

After calling the command, select the dimensioning objects that dimensioning text was moved.

Close the object selection by pressing RETURN.

---

### See also:

Move Dimensioning Text

## Command DIMTMOVE



**Command: DIMTMOVE**

**Alias DTM**

### Located:

1. **Pull-down Menu: Dim – Modify Dimension Text – Move DimText**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to move the dimensioning text of existing dimensioning objects. It doesn't move the associated dimension arrows.

After calling the function, select a dimensioning object. Move the dimensioning text by "pointing" a new insertion point for the text object.

---

### See also:

Undo Repositioning

Command DIMTNEW



**Command: DIMTNEW**

**Alias DN**

### Located:

1. **Pull-down Menu: Dim – Modify Dimension Text – Modify Dim Text Value**
2. **Palettes: Modify Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



This command allows the user to edit and modify the dimensioning text value.

After calling the function, enter the new dimensioning text value. Select one or several dimensioning objects to be changed.

Press RETURN to close the object selection. The dimensioning text of the selected objects will be modified accordingly.

### **Rotate Dimensioning Text**



**Command: DIMTROT**

**Alias DTR**

#### **Located:**

- 1. Pull-down Menu: Dim – Modify Dimension Text – Rotate DimText**
- 2. Palettes: Modify Dimensions and Dim**
- 3. Or Enter Command or Alias Directly at the Command Line**

Using this function, it is possible to rotate the dimensioning text of a previously generated dimension.

After calling the function, first enter the rotation angle, then select the one or more dimensioning text objects. Press RETURN to finish the object selection.

The selected dimensioning text will be rotated around its insertion point.

### **Dimensioning Styles**



**Command: DIMTYPE**

**Alias DDR**

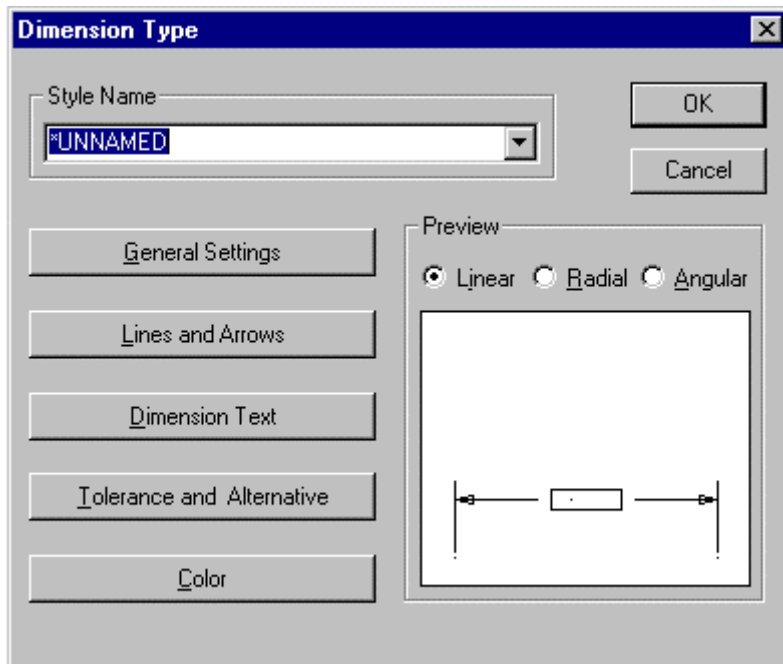
#### **Located:**

- 1. Pull-down Menu: Dim – Dimension Type**
- 2. Palettes: Create Dimensions and Dim**
- 3. Or Enter Command or Alias Directly at the Command Line**

Dimensioning format is controlled by the Dimensioning dialog box. Use the style name, input and selection dialog to either store the current settings as a dimensioning style, or to select a pre-defined style as the default.

The buttons described below each open additional dialog windows in which individual dimensioning parameters can be defined.

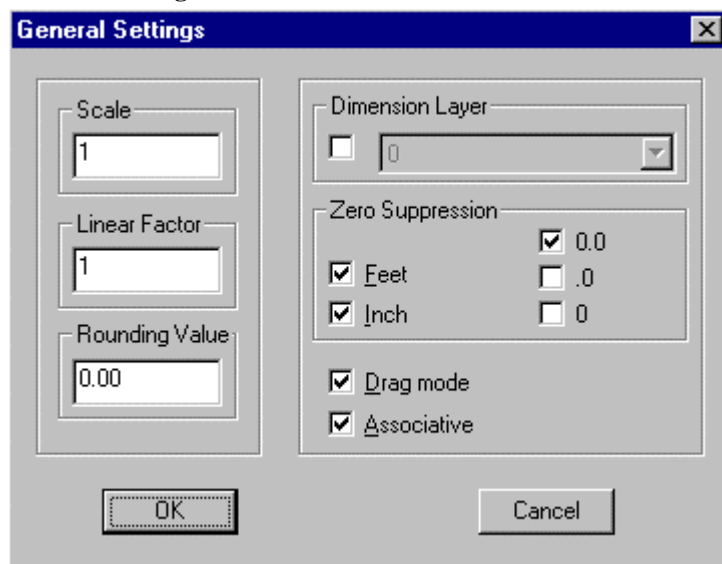
#### **Dialog Box Options:**



When you issue the DIMTYPE command, you are presented with the opening dialog. Here, you see five bars on the left side that lead to specific configuration areas. On the right, you can click on the Linear, Radial and Angular radio buttons, to see a representation of current settings in the window below, for each dimension type.

Clicking on each bar will produce the dialogs detailed below:

#### General Settings



#### Scale

Scale value enlarges or reduces the size of all dimensioning elements (apart from the dimension value itself) and allows the dimensioning elements to be scaled to suit the aspect ratio of the drawing.

#### Linear factor

Linear factor converts the measured drawing units into a dimension figure. The default value is 1.

#### Precision

Precision defines the number of decimal places to which the measurement will be carried.

### Dim. Layer

The layer in which the dimensioning values are stored is identified by the name entered in the dimensioning layer selection field. Select the layer required, or enter a new layer name. It is prerequisite that the option has already been activated by clicking the appropriate control box. A check mark indicates the current status of the option.

### Zero Suppression

The Zero suppression field decides the treatment of the display mode for zeroes.

### Drag mode/ Associative

The last two control boxes activate or deactivate either the "rubber band" mode and / or the associative character of the dimensioning.

### Lines and Arrows

The screenshot shows the 'Lines and Arrows' dialog box. It has a title bar with a close button. The dialog is organized into two main columns. The left column, titled 'Lines', contains three sub-sections. The first, 'Extension Lines', has input fields for 'Distance' (0.06) and 'Extension' (0.18), followed by checkboxes for '1. invisible' and '2. invisible'. The second, 'Dimension Lines', has input fields for 'Increment' (0.38) and 'Extension' (0.00), followed by checkboxes for 'Interior lines' and 'Always interior lines'. The third, 'Center Mark and Lines', has an input field for '0.09' and a checkbox for 'Lines'. The right column, titled 'Arrows', has four radio buttons: 'Arrow' (selected), 'Circle', 'Slash', and 'User'. Below these is a 'Blocks' section with a 'Size' input field (0.18) and two radio buttons: 'Same blocks' and 'Seperate blocks'. At the bottom of the dialog are 'OK' and 'Cancel' buttons.

Extension lines are the lines that extend the object dimensions outside the perimeter of the measured object, assuming that dimensioning is to take place outside of these perimeter constraints. These lines, and the nature of the arrowheads to be used, are defined in the Dialog Box *Lines and Arrows*.

### Lines

The fields in this area decide the characteristics of the extension and dimension lines.

#### Extension Lines

This area defines the spacing required for the extension lines.

**Distance** allows the user to choose the space between the object and the point at that the extension lines begin. A value = 0 indicates that the lines will start directly on the object itself.

**Extension** dictates how far the extension lines should extend past the dimensioning line.

The control boxes marked **Invisible** can be used to suppress either one or both of the extension lines in the drawing.

## Dimension Lines

Dimension lines either connect the object edges, or the extension lines of the object that is to be measured. The dimensioning text is written along this line.

**Increment** sets the spacing between successive dimensioning lines that use the same extension lines.

**Extension** allows the user to enter the measurement by that the extension lines should run past the dimensioning lines. This value can only be entered when the **Slash** option has been selected as the dimension arrow type.

The field **Interior Lines** or **Always Interior Lines** decides whether dimensioning lines should be allowed to extend past the extension lines.

## Center Mark and Lines

This area defines the existence and size of a cross-mark indicating a circle center point. It also defines whether or not dashes or strokes should be added to the center point cross or not.

## Arrows

This field allows the user to select the type of arrowhead to be used.

## Blocks

This area describes the length of the arrowhead. Only the size selection of this area is normally active. Block Name is active only when the arrow style selected is User. The size of the Block to be used is defined here.

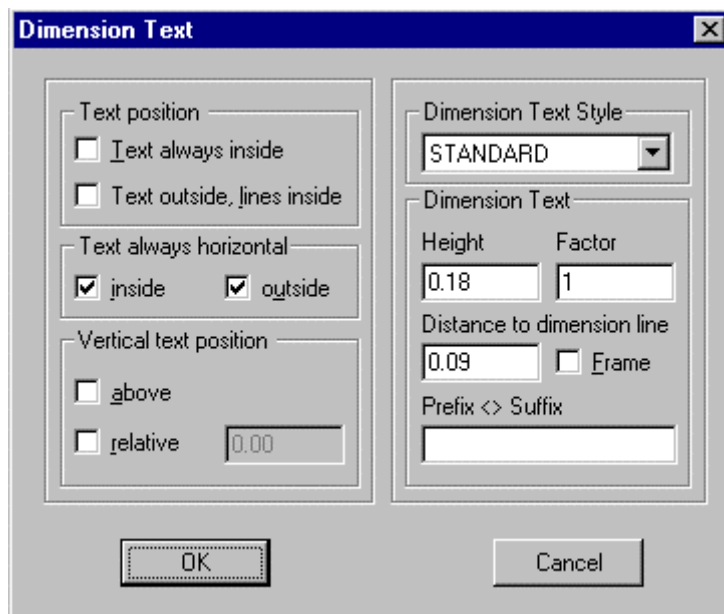
## Block Name

Active only when the arrow style selected is User, this field decides the Block Name. The name commonly in the field is ARROW.

## Separate Blocks

This field either allows or suppresses the use of divided or separate arrow blocks. When activated, it is also possible to define that elements should be used at either end of the dimensioning line.

## Dimension Text



The image shows a dialog box titled "Dimension Text". It contains two main sections. The left section is titled "Text position" and includes three sub-sections: "Text position" with checkboxes for "Text always inside" and "Text outside, lines inside"; "Text always horizontal" with checkboxes for "inside" and "outside", both of which are checked; and "Vertical text position" with checkboxes for "above" and "relative", and a text field for "0.00". The right section is titled "Dimension Text Style" and includes a dropdown menu set to "STANDARD", a "Dimension Text" section with "Height" (0.18) and "Factor" (1) text fields, a "Distance to dimension line" text field (0.09) and a "Frame" checkbox, and a "Prefix <> Suffix" text field. At the bottom are "OK" and "Cancel" buttons.

These dialog window fields determine the position and look (size, style, spacing) of the dimensioning text.

### Text position, Text always horizontal, and Vertical text position

These fields serve to define the position of the text with respect to the dimension lines.

### Dimension text style, Dim text

These fields define the style and size of the text, and the distance separating text lines.

The prefix and suffix fields enable the user to type in a character string that will appear before or after the dimensioning text. For example, this field may be used to display the unit of measurement.

The < > character decides whether the character string will appear before or after the dimensioning text. For example, **approx. <> ft.** entered in this field would produce the following dimensioning : **approx. 12 ft.**

### Tolerance and Alternative Dimensioning

The screenshot shows a dialog box titled "Tolerances / Alternative Units". It contains several input fields and checkboxes. Under the "Tolerance" section, there are checkboxes for "Variance" and "Limits". Below these are two groups of fields: "Positive Tolerance" and "Negative Tolerance", each with a "Value" field (containing "0") and a "String" field. The "Alternative Mode" section has a checkbox for "on / off". Below it is the "Alternative Units" section with a "Factor" field (containing "25.40") and a "Decimals" field (containing "2"). At the bottom is an "Alternative String" text area. At the very bottom are "OK" and "Cancel" buttons.

The tolerance settings control the tolerance, both the maximum and minimum values, allowed. The control fields activate or deactivate the display of these elements. A check mark in either of the control windows shows the current status of the individual options.

Values for the positive or negative tolerance are entered in the field marked Values. These fields also enable the user to type in a line of text (String).

The **Alternative mode** options control the use of a second, alternative dimensioning method. This mode, when activated, will allow dual dimensioning in metric and imperial values.

### Factor

The conversion factor between the two units of measurement can be entered in the factor field.

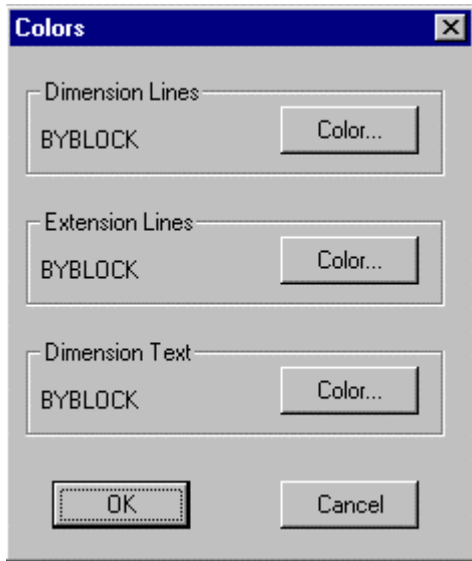
### Decimals

The number of decimal places selected will appear in the decimal display.

### Alternative string

Alternative string should contain a description or title of the second unit of measurement.

### Colors:



The dialog window allows colors to be assigned to the dimensioning lines, extension lines and dimensioning text (dimension value, alternative dimensioning, tolerances, limit values, unit of measurement etc.)

Select the appropriate element and allocate a color from the displayed selection box. 256 standard colors are available in addition to the BYLAYER and BYBLOCK options.

## DIMUPD



**Command: DIMUPD**

**Alias DU**

### Located:

1. **Pull-down Menu: Dim – Diameter**
2. **Palettes: Modify Dimensions and Dim**
3. **Left SideBar**
4. **Or Enter Command or Alias at the Command Line**

After changing the current dimensioning style, all new dimensions will be created in line with the new criteria. Using the Update Dimension function, it is possible to update all of the existing dimensions so that they comply with the new dimensioning style settings.

Using one of the object selection functions found in the options bar, mark or select the dimensions that are to be updated so as to reflect the new dimensioning style.

It is possible to update individual, group or all dimensions contained within a drawing at one time.

Confirm the selection process by pressing RETURN. The marked dimensions will then be updated to conform to the new dimensioning style. The program will display a x dimensions found notice telling the user how many dimensions have been updated.

The following dialog examples demonstrate the selection process and should clarify the use of the DIMUPD command.

> DIMUPD

Select Dimension(s): Fence

First Fence point: P1

Endpoint of the line: P2

Endpoint of the line: P3

RETURN

Select Dimension(s): All

2 Dimensions found  
 Or  
 > DIMUPD  
 Select Dimensions(s): All  
 Select Dimensions(s): RETURN  
 3 Dimensions found!

## Vertical Dimensioning

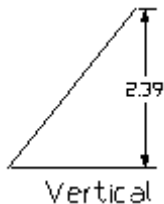


**Command: DIMVER**

**Alias DVR**

### Located:

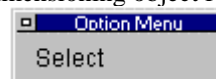
1. **Pull-down Menu: Dim – Vertical**
2. **Palettes: Create Dimensions and Dim**
3. **Or Enter Command or Alias Directly at the Command Line**



Vertical dimensioning is linear dimensioning that measures and labels the distance between two points or the linear extension of an object parallel to the Y-axis.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points will be measured along the Y-axis and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMVER
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMHOR
1. Point: SELECT <CR>
```

select line, polyline, circle, arc: **P1**  
Dimension line positioning: **P2**  
Dimension text <2.5000>: **Return**

## Preload Parts

### Command: DINSERT

#### Alias DIN

#### Located:

##### 1. Entered Directly at the Command Line

The command PINSERT incorporates an external block into the current drawing without creating an instance of the block in the drawing. This variant can be seen as discrete insertion.

The main purpose of this command is

- to preload parts (blocks) for later use
- to load table definitions defined in a template drawing (layers, linetypes, text styles, dimension styles, etc.)

The only request of the command is to specify the file name of the external part be brought in, but not actually be inserted into the drawing. Only the symbol tables will be enhanced by those (additional) entries found in the discrete inserted drawing.

This command is provided especially for use in the programming interfaces.

---

#### See also:

INSERT

Distance / Angle



#### Command: 'DIST

#### Alias D

#### Located:

1. **Palette: Edit**
2. **Left Sidebar**
3. **Pull-down Menu: Edit – Distance / Angle**
4. **Or Enter Command or Alias at the Command Line**

The transparent command **Distance / Angle** will determine the distance or angle between two points and displays them in the text window:

1. The absolute distance (range);
2. The Distance on the X-, Y- and Z-axis;
3. The Angle on the XY-plane; or
4. The Angle on the XY-plane.

The points may be arbitrary points that do not have to be part of a single drawing object.

The function Distance / Angle operates as follows:

1. Determines the two points required by selecting them with the Cursor, using the Object



Snap functions or by input of the coordinates; then

2. The distance and angle between the two points will be displayed.

The term *Distance* is understood to mean the absolute distance between points.

Example:

> DIST

First Point:

Second Point:

Distance: 0.37 Angle in XY plane: 41 Angle from XY plane: 0

Delta X: 0.28 Delta Y: 0.24 Delta Z: 0.00

The terms *Delta X*, *Delta Y* and *Delta Z* designate the distance between the two points in the direction of the respective axis of the coordinate system. All length values are given in drawing units.

*Angle in the XY-plane* measures the angle to the second point starting from the X-axis. It always displays the lesser-valued angle, no matter what the rotation direction.

*Angle from the XY-plane* measures the angle between the first and the second point from the XY-plane to the Z-axis. It is assumed that the first point lies on the XY-plane.

## Commands Measure and Divide

**Command: MEASURE, DIVIDE**

**Alias MR, DI**

**Located:**

1. **Pull-down Menu: Draw – Measure / Divide**
2. **Or Enter Command or Alias Directly at the Command Line**

The commands MEASURE and DIVIDE order points or multiple blocks (parts) along a 2D-Object. This object can be an arc, a line, a circle or a 2D-Polyline. Before a block may be used with these commands, **it must exist in the drawing database before the command is given**. This may be accomplished, either through insertion or part definition. When the command is given, the block can be inserted aligned, or not aligned to the object. Both commands work in a similar fashion. While the command MEASURE asks for a distance, DIVIDE requests for the number of points with the distance between points or parts dependent upon the length of the linear entity selected.

### Command sequence for MEASURE and DIVIDE

After typing in "**measure**" or "**divide**" (or selecting it from the menu DRAW), the following request appears:

/↻> measure

Select a line/circle/2d-polyline or an arc to measure:

After selecting the entity it is highlighted and the following question appears:

Block or distance:

Here you can input either a distance directly, (by putting in a real value or selecting two points), or you can input "**Block**" (or "**B**". "**BI**" etc), to use a block. In the first case the points at each distance from the

start to the endpoint of the object are inserted and so the command is completed. If you select the option **"Block"** you will be asked for the block name:

Name of block to insert:

Now type in the name of an **existing** block. You will be asked for the offset angle for the alignment of the block along the object.

Offset angle for alignment the block along the object (or ENTER for no alignment):

Here you can press return to avoid the alignment, or to put in any offset angle that is added to the tangential angle of the object at the point of insertion. Lastly, you will be asked for the distance again.

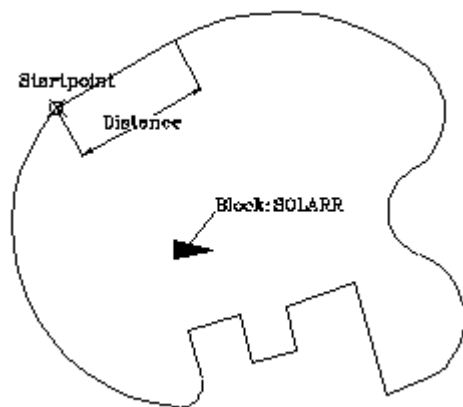
Distance for dividing the object:

You have to put in the distance here, either by giving a real value or by selecting two points. Now the Block at each position is inserted and the command will terminate.

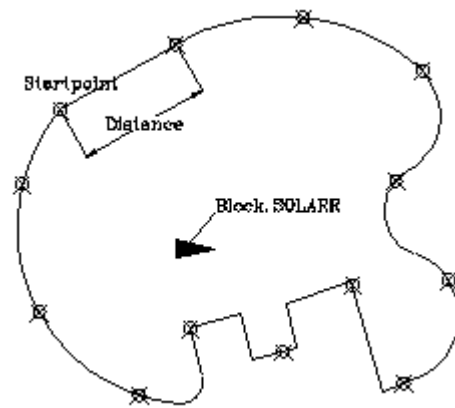
The sequence of the command **DIVIDE** is the same as above, except that you are asked for the number of points or blocks to insert, instead of a distance. In this case the command will compute the length between the insertion points itself.

### Example:

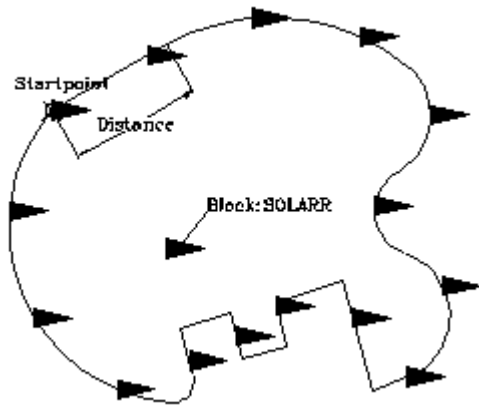
The following pictures show at the sample of a polyline, how the commands **MEASURE** and **DIVIDE** are working:



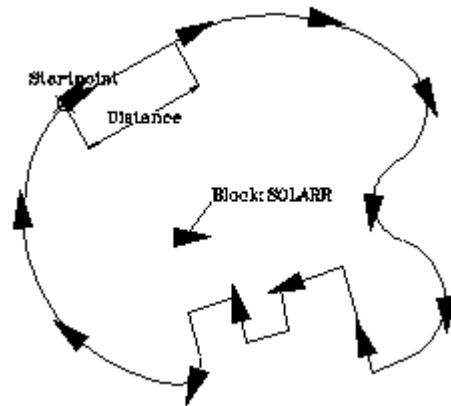
The initial state: The entity to measure is the polyline. Moreover the block SOLARR is already defined in the drawing.



After measuring the polyline using the distance, we receive points at each distance along the polyline, starting at the polyline start point



After dividing the polyline in ten pieces the block SOLARR is inserted ten times along the polyline, but it is not aligned.



In this case the polyline is divided into 12 pieces with the option block, and aligned with an angle of 0 degrees, this means the block is inserted locally tangential to the polyline.

#### Technical notes

The command Divide inserts number+1 entities along the object unless a circle is selected. So it may be necessary to delete the first or last inserted entity, if a closed polyline is selected. This is much easier, however, then creating an additional entity manually.

The command is not an internal command and so it available for the usage inside the (command...) function. To use it inside of a LISP-application use the related LISP-Function (MesDiv EntName Dist|Nr B|Name OffsAngle) instead that is defined in MesDiv.LSP in the PCAD-Applic directory.

---

#### See also:

POINT

## Desktop and Menu Editor

### Desktop and Menu Editor – Command DLGEDIT

Alias DM

Located:

1. Pull-down Menu: Tools – Resource Manager – Run Desktop and Menu Editor
2. Or Entered Directly at the Command Line

The Desktop and Menu Editor (DME) design and lay out your menus and palettes in a visual environment. Knowledge of programming is not required.

Please refer to the Command Button Chart for input information when modifying palettes.

For more information on the DME click here: DME Help 

## Distance Offset

**Command:** DOFFSET

Alias DOF

**Located:**

1. **Pull-down Menu: Modify – Offset Distance**
2. **Or Enter Command or Alias Directly at the Command Line**

The DOFFET command is similar to the command OFFSET, but not identical. Like OFFSET, the command DOFFSET creates lines and curves parallel to an existing entity. DOFFSET has been added for convenience of those users who are familiar with other CAD programs.

The main difference is that the command DOFFSET assumes that you want to input an offset distance first before you offset selected entities:

```
> DOFFSET
Distance or select option ThruPoint <1.00>: 0.30
Select entity (line, circle, arc, 2D-polyline):
Side to offset ?
Select entity (line, circle, arc, 2D-polyline):
Side to offset ?
Select entity (line, circle, arc, 2D-polyline): <RETURN>
```

If you want to indicate a through point (such as a significant point in the given geometry), select the option *ThruPoint* (or type, for short, T):

```
> DOFFSET
Distance or select option ThruPoint <0.30>: T
Select entity (line, circle, arc, 2D-polyline):
Offset through point:
Select entity (line, circle, arc, 2D-polyline):
Offset through point:
Select entity (line, circle, arc, 2D-polyline): <RETURN>
```

You can offset an entity to both sides by the specified distance by selecting the option *BothSides* when you are asked to specify the side to offset:

```
> DOFFSET
Distance or select option ThruPoint < 0.50 > :
Next point:
Select entity (line, circle, arc, 2D-polyline):
Side to offset ? BothSides
Select entity (line, circle, arc, 2D-polyline): <RETURN>
Side to offset ?
```

---

**See also:**

OFFSET

## Command DPAN



Command: 'DPAN'

### Alias DP

**Located:**

1. **Pull-down Menu: View – Dynamic Pan**
2. **Left Sidebar**
3. **Palette: 3D View**
4. **Or Enter Command or Alias at the Command Line**

As described previously, panning means to move the display of the drawing in its current window without changing the magnification.

With the command **DPAN** you can simply grab the drawing and move it in any direction you want, any number of times, until you have the drawing view that you want. At this point, you terminate the command by pressing the ESC key or Enter (right mouse click) and then select the Exit option.

To pan the drawing in this manner, choose *Dynamic Pan* from the menu *View*. The cursor changes to an icon symbolizing a hand. Place the cursor anywhere in the current drawing, then left-click. Now drag in the direction you want the drawing to move, releasing the mouse button to release the "grip" on the drawing.

If you reach the edge of the drawing window, release the mouse button, move the cursor away from the edge, and left-click and drag again.

To exit the Dynamic Pan mode, press **ESC** or **ENTER**. You can also right-click to open the context menu that contains the *Exit* option. Alternatively, you can start a command via the pull-down menus or palettes to exit the dynamic pan (this works, as long as you do not call a transparent command from the menu or palette).

Beneath leaving the dynamic pan operation, the cursor menu (that you can open by clicking the right mouse button) also allows you to switch to dynamic zoom mode and to execute a Zoom Extents.

Dynamic Pan can be used transparently (in the middle of another command).

---

**See also:**

PAN command

DZOOM command

## Drawing Modes...



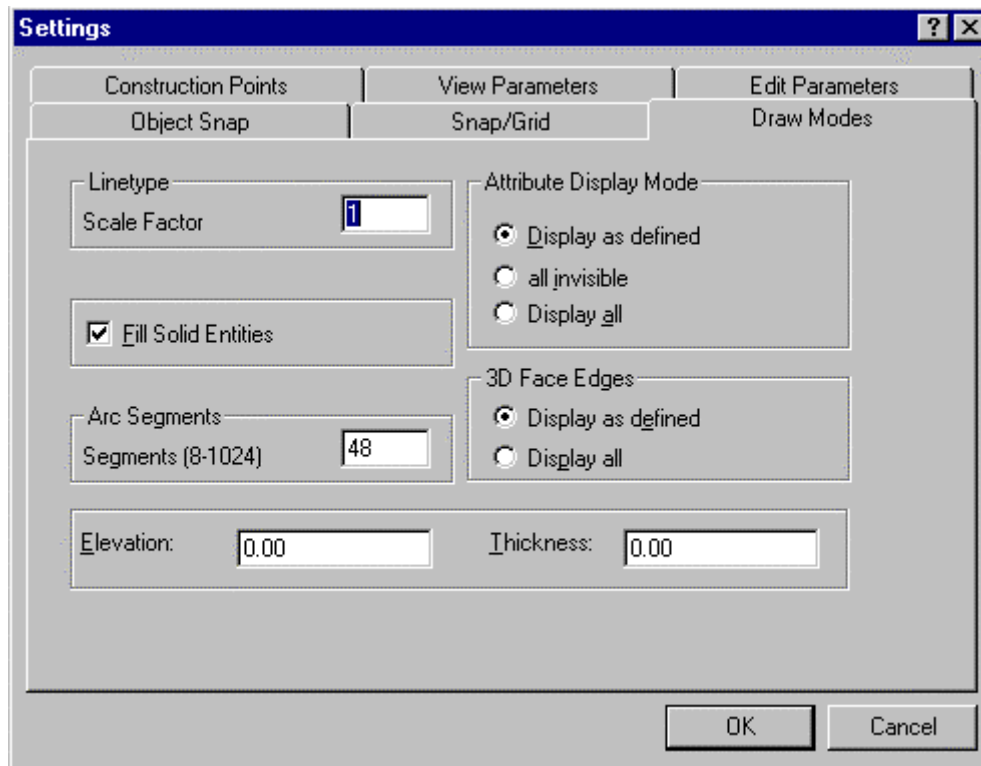
**Command: DRAWMODE**

**Alias DR**

### Located:

1. **Pull-down Menu: Draw – Drawing Modes**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

The Drawing Modes tab (on the SETTINGS command dialog) allows you to specify default settings for subsequent drawing commands. Parameters may be changed, activated, or inactivated.



## Dialog Box Items

### Linetype Scaling Factor

*Linetype Scaling Factor* determines a scaling factor for the line pattern selected. The line patterns are shown extended (long dash lines, for instance) or contracted (short dash lines, for instance) according to this scale. The default value is 1.00. To change the scale, enter a new factor and verify it.

Remember that, after a regeneration of the drawing, the scales of line patterns that have been drawn already will be adjusted to the new value. Results of *Ltscale* changes will therefore not be apparent until after a REGEN.

### Filling Solid Entities

*Fill Solid Entities* specifies whether or not to "fill" a polygon. Polygons are closed 2-D planes, or polylines with a width greater than zero.

A cross in the box indicates whether or not the "fill" function is activated.

Remember that activating the function is required in order to draw filled planes. If the option *Fill Solid Entities* is deactivated, then the option *Fill Polygons* will not be available in the Dialog Box *Print/ Plot*.

### Attribute Display Modes

This option allows the user to choose the desired method of display of attributes. The options are:

<b>Show as Defined</b>	All attributes are displayed according to their definition.
<b>Show Nothing</b>	All attributes are made invisible, e.g. their status is defined as invisible.
<b>Show All</b>	All attributes are displayed, independently of how their attributes were defined.

Choose a display option by clicking its field.

### 3D Frame Visibility

If, in the display of 3-D views, intersections or lines that are "behind" other drawing segments are to be

hidden, then the user should select the option *normal*. If, on the other hand, all intersections are to be displayed, then the selection should be *show all*.

### Arc Segments

Arcs and circles, understandably, are always displayed as a series of short straight-line segments.

In this area the user selects the number of segments that will appear in the definition of any arc or circle that is drawn. The number selected may be any whole number between 8 and 1024. A smaller number will make the arcs or circles appear "jagged"; a larger number will tend to smooth them out. The default number used is 48.

Remember that a large number of line segments in an arc or circle will tend to slow the performance, whereas a small number of segments will increase working speed.

### Point Representation

This dialog box area allows you to choose between 20 different display forms for construction points. The point forms will be updated when the drawing is regenerated.

### Point Size

The user chooses the size of the construction point in this area.

<b>Absolute</b>	The size of the construction points is determined in the absolute scale of drawing unit .
<b>Relative</b>	The display of the construction points is determined relative to the size of the visible drawing plane.

This parameter defaults to *Absolute*, and uses 5.00% of the drawing unit size as the size of the construction point.

## Command DRAWORDER

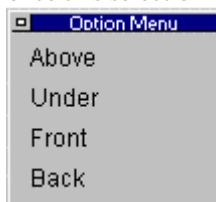
### Command: DRAWORDER

#### Alias DO

#### Located:

1. **Pull-down Menu: Edit – Change Object Display Order**
2. **Or Enter Command or Alias at the Command Line**

The **Command\_DRAWORDER** command changes the drawing and plotting order of any object in the AllenCad drawing database. In addition to moving objects to the "front" or "back" of the sort order, you can order objects relative to another object (that is, above or below a selected object). After issuing the command, you are asked to pick entities to re-order. You are then presented with the following dialog, once this selection is complete:



You simply select the option that you want and then select the entities that you want the original entities placed in relation to.

AllenCad differs from AutoCAD, in that object database order is changed directly, using the **Command\_DRAWORDER** command. Different object sort methods are therefore not supported, and a **SORTENTS** system variable is not required in AllenCad.

See Also HIDE, Fhide

## Command DTEXT

**Command:** DTEXT



Alias DT

### Located:

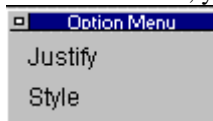
1. **Pull-down Menu:** Detail – Single Line Text
2. **Palette:** Detail
3. **Or Enter Command or Alias at the Command Line**

Unlike the other text input commands, DTEXT places your text on the drawing as you enter it.

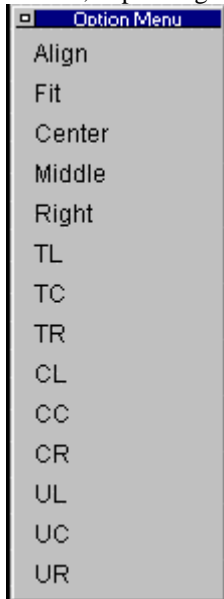
AllenCad's DTEXT command operates essentially the same way that AutoCAD's DTEXT command does. When you issue the command, you are prompted at the command prompt to pick the start point for your text on the drawing.

```
> *** Cancel ***  
> dtext  
Specify start point or select option (Justify, Style):
```

At the same time, you are offered two options:



These will either be offered at the command line as well as in the option menu as shown, or else on the toolbar, depending upon your desktop preferences. Choosing Justify will raise another dialog:



On this dialog you can select the justification you want the text to have.

Choosing Style will raise an input prompt at the command line requesting the required text style name.

Next, you will be prompted to enter the required text height, if other than the current default.



```
> dtext  
Specify start point or select option (Justify, Style):  
Text height <0.20>: |
```

Note that as with most dimension prompts, you can also respond by picking two points on the drawing that will define the height to apply to the text.

Next, you are asked to input the angle (orientation) at which the text is to run. Again, picking two points on the drawing will establish the text orientation.

```
Specify start point or select option (Justify, Style):  
Text height <0.20>:  
Insertion angle <0>: |
```

Lastly, you will type in your text. Note that it appears immediately on your drawing as you type. Pressing the Return key will move you to a position one line below, on which to type a second or subsequent lines of text. Pressing Return key twice in a row will terminate text entry and pressing the Esc key will cancel the current command.

## Command DVIEW

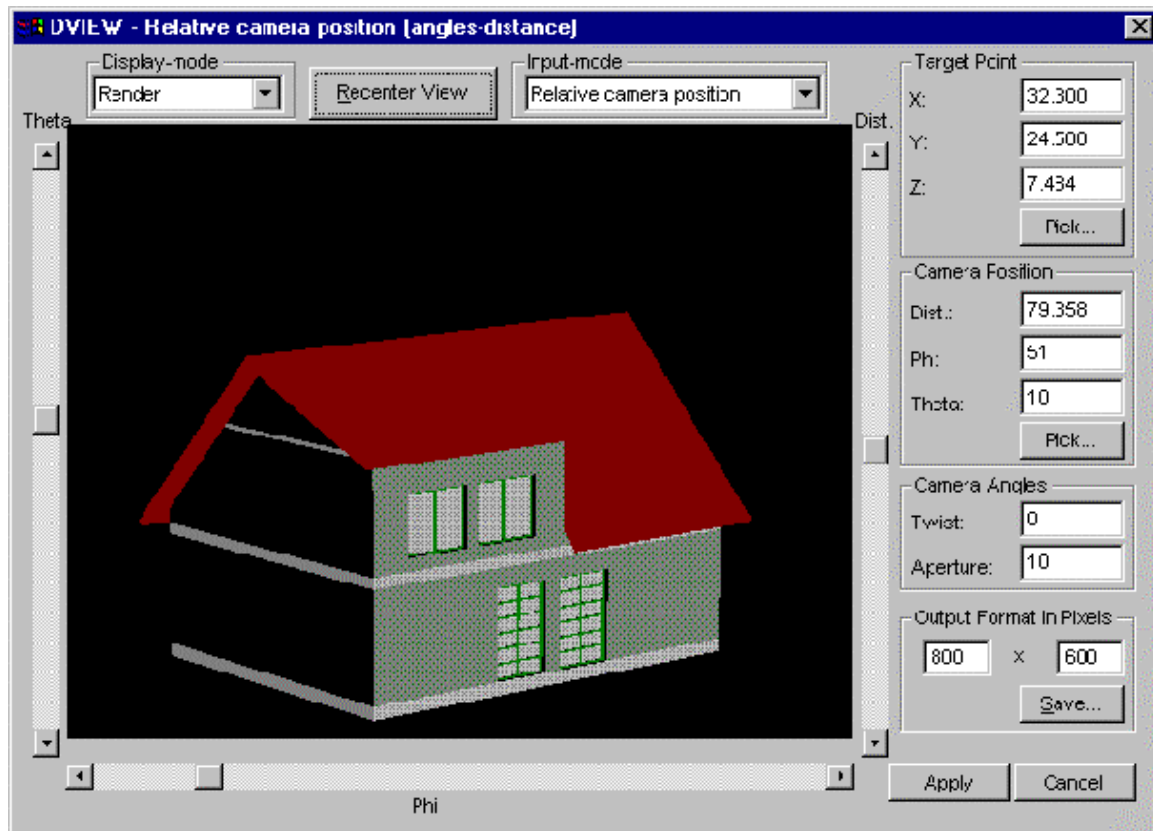
**Command: DVIEW**

**Alias DV**

### Located:

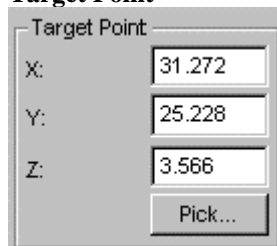
1. **Pull-down Menu: View – Dynamic View**
2. **Or Enter Command or Alias at the Command Line**

The **DVIEW** command creates a dynamic 3D-perspective view using different display modes. The command shows a dialog with a preview window. It is possible to save this view as a separate bitmap (BMP) file as well as transferring this view to the drawing.



The functionality of DVIEW in detail:

#### Target Point



This is the point the camera is aimed at. The x,y,z coordinates can be set either by entering values in the appropriate edit boxes or by picking (**pick...** button) a point in the drawing using the standard snap functions.

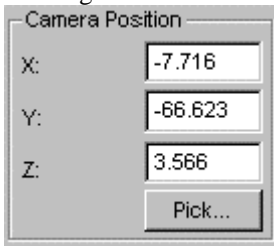
#### Camera Position



This is the position of the camera relative to the target point, in other words, the position the drawing is viewed from. The distance (**Dist.**) as well as longitudinal (**Theta**) and latitudinal (**Phi**) settings can be set in this section.

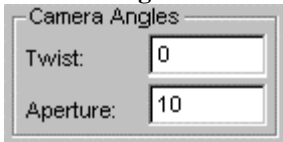
The **Theta** and **Phi** values correspond to a sphere of radius **Dist.** centered around the target point.

Again, these values can be adjusted by entering the values in the edit boxes or by picking a point in the drawing.



Alternatively, the absolute camera position can be set by selecting the **Absolute camera position** entry in the **Input** drop-down box pictured above.

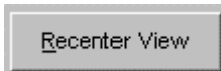
### Camera Angles



**Twist:** rotates the camera while facing the target point.

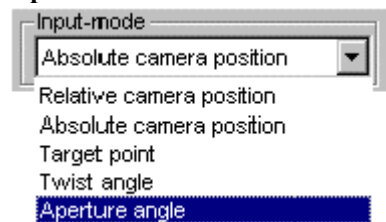
**Aperture:** changes the aperture of the camera.

### Re-center view



This button re-centers the view. Pressing this button shows a good starting point for the rotation. Note: when you first issue the DVIEW command, you may need to re-center the view to bring the drawing into view.

### Input Mode



The input mode changes the function of the sliders on the preview window. The different modes are:

**Relative camera position:** The sliders set the relative camera position values using Phi/Theta/Dist.

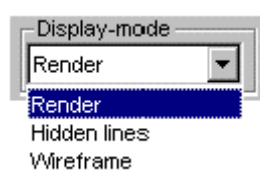
**Absolute camera position:** The sliders set the absolute camera position.

**Target point:** The sliders set the absolute target position.

**Twist angle:** A single slider changes the twist of the camera.

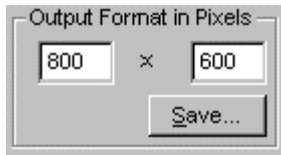
**Aperture angle:** A single slider changes the aperture of the camera.

### Display Mode



Three display modes are supported: Render, hidden line and wire frame.

### Output Format in Pixels



The 3D-view displayed can be exported as a bitmap file at the resolution entered in the edit boxes.

**Save ...** opens a standard save file dialog.



**Apply** assigns the changes to the view to the drawing coordinate system.

Please note that panning of objects (pressing the mouse in the picture and then dragging the mouse to a new location) in the preview does not result in a panning of the object in the actual drawing as only changes in the perspective view are transferred to the drawing.

## Import DXF-File

**Command: DXFIN**

**Alias DXN**

**Located:**

### 1. Entered Directly at the Command Line

The command DXFIN reads a DXF file specified as argument to the command and opens the file as FLX drawing.

DXFIN is performed at command line and is provided especially for Lisp programming.

Specify the name of the file to be imported. If necessary, enter the exact path of the import file.

**Note:** DXF files can also be opened as a new drawing, using the OPEN command, or inserted as a part, using the INSERT command.

## Export DXF-File

**Command: DXFOUT**

**Alias DXO**

**Located:**

### 1. Entered Directly at the Command Line

The command DXFOUT allows you to export the current drawing as DXF file.

DXFOUT is performed at command line and is provided especially for Lisp programming.

Specify the name and the exact storing location for the export file.

## Dynamic Zoom



**Command: 'DZOOM**

**Alias DZ**

**Located:**

1. **Pull-down Menu: View – Dynamic Zoom**
2. **Left Sidebar**

### 3. Or Enter Command or Alias at the Command Line

With the command **DZOOM** you can zoom in or out to any extent within the current drawing window.

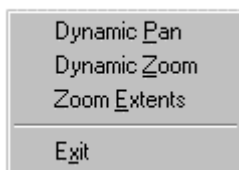
To zoom in this manner, choose *Dynamic Zoom* from the menu *View*. The cursor then changes to a magnifying glass icon with a plus sign and a minus sign. To start the dynamic zoom process, click the left mouse button and drag the mouse, either up or down. Whether you zoom in or out depends on the direction of movement of the cursor. To zoom in, drag up (in the direction of the plus sign). To zoom out, drag down (in the direction of the minus sign).

If you reach the edge of the drawing window, you can release the mouse button, move the cursor away from the edge, and left-click and drag again.

To exit the Dynamic Zoom mode, press **ESC** or **ENTER**. You can also right-click to open the context menu that contains the *Exit* option.

Alternatively, you can start another command via the pull-down menus toolbars or palettes to exit the dynamic zoom process (this works, as long as you do not call a transparent command from the menu or palette).

While executing the DZOOM command, you can also right click with the mouse and produce the following option dialog:



Beneath leaving the dynamic zoom operation, the cursor menu (that you can open by clicking the right mouse button) also allows you to switch to dynamic pan mode and to execute a Zoom Extents.

Dynamic Zoom can be used transparently (in the middle of another command).

---

#### See also:

ZOOM command

DPAN command

## "E"

Parameters for Modify Commands...



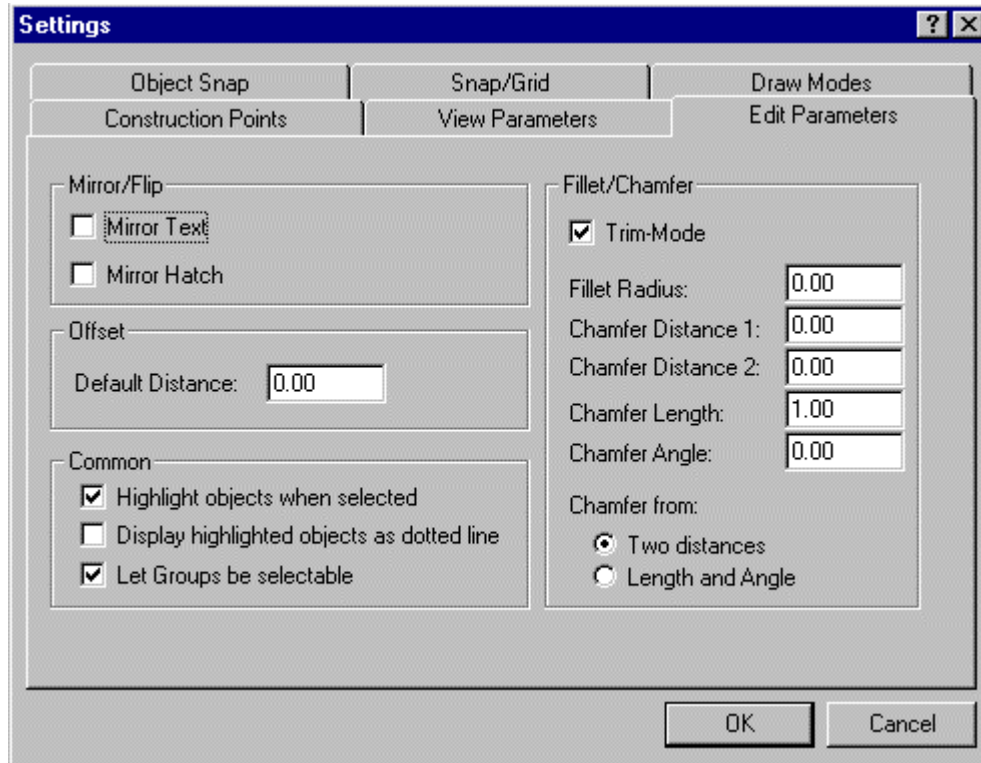
**Command: EDITPAR**  
**Alias EP**

#### Located:

1. Pull-down Menu: **Modify – Modify Parameters**
2. Palette: **Modify**
3. Or Enter Command or Alias Directly at the Command Line

The function *Modify parameters*, allows the user to define a series of parameters that are used as preset values for some editing functions. These parameters are mirroring of text entities, trimming and chamfering, and the radii of fillets.

*Modify parameters* is called by selecting that option from the pull-down menu *Modify* or typing in EDITPAR. A dialog box will appear. This dialog is the Edit Parameters tab dialog for the SETTINGS command. Using EDITPAR or the menu selection, is simply a more direct route to this dialog.



### **Mirror / Flip**

Mirror / Flip determines the position and orientation of text objects in the drawing.

Activate MIRROR / FLIP, by checking the box MIRROR TEXT in that position, from the dialog box *Edit Parameters*.

Activating the option will cause the text objects to be mirrored, with the original text object retained.

With the option de-activated, the text objects will be copied, but not mirrored, about the specified axis.

### **Offset Distance**

*Offset* determines the parameter value for the distance in which an object will be offset when the command OFFSET is used. This value defines the distance of each point of a copy from the original drawing entity. There is more detail on the command OFFSET in this section. Enter the desired value in the input area.

Note that, during a working session the value used last will always be displayed as the parameter value for the next call to the command.

### **Fillet / Chamfer**

This area decides whether the trim function is used by default when these commands are called, or whether a separate TRIM will be required.

### **Trim Mode**

Trim Mode, when activated, will cause the erasure of those parts of objects that lie between, inside, or outside of intersecting points. For more detail see the TRIM command.

### Fillet Radius

This field allows the user to specify a fillet radius as a default value. Enter the desired value in the input area. Please note thereby that during a working session the value last used will always be displayed as the parameter value for the next call to that command.

### Chamfer Distances

This values establish default values for the chamfer distance from the point of intersection. Each value may be specified independently of the other.

## Command EDITOR

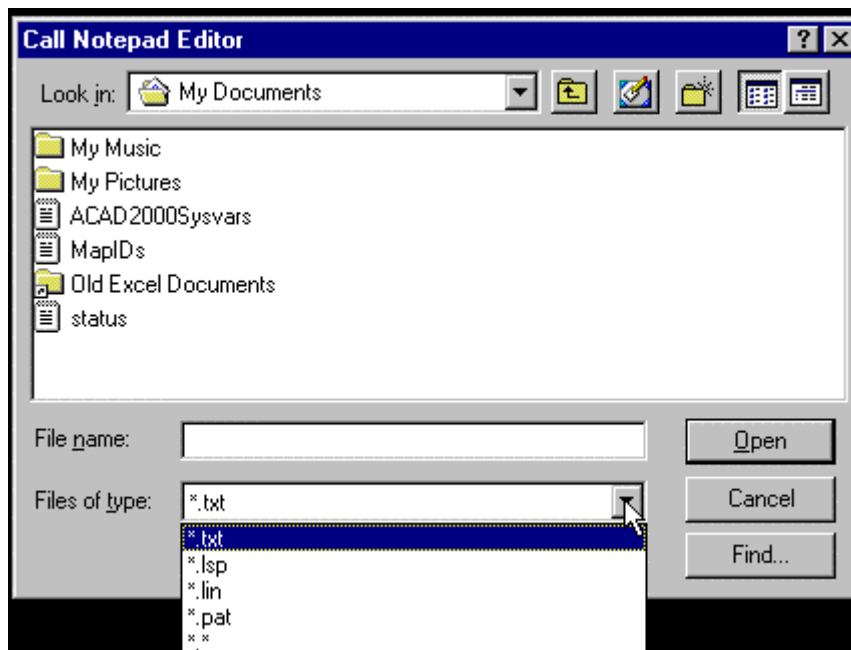
### Command: EDITOR

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Editor**
2. **Or Enter Command or Alias at the Command Line**

The EDITOR command is used to call a file into Window's Notepad program, to allow you to make file edits while working in AllenCad. This can be handy for such tasks as modifying LISP routines, writing Macros or editing a text file prior to importation into AllenCad. The command is located on the Tools pull-down menu under "Resource Manager – Editor" and can be also run at the command line.

Upon execution, you are presented with the following Dialog:



Notice that it allows you to select certain file types that are commonly used by the program to assist in file selection. You can also select \*.\* as an option, to display all files types. After selecting the file type, you can simply browse to locate the file you wish to edit. After selecting the file and pressing the Open button, the file is opened in Notepad for editing in a new window.

## Command EDGE



**Command:** EDGE

**Alias** EG

### **Located:**

#### **1. Entered Directly at the Command Line**

The EDGE command allows the user to select 3DFACE edges to be hidden and not displayed. The command has an option, "Display" that allows you to reverse the hidden status of selected edges and make them visible again.

See Also:      HIDE  
                 FHIDE  
                 DRAWORDER

## Entity Info...



**Command:** EINFO

**Alias** LI

### **Located:**

- 1. Pull-down Menu: Edit – Entity Info**
- 2. Left SideBar**
- 3. Palette: Edit**
- 4. Or Enter Command or Alias at the Command Line**

The EINFO command prompts you to select objects for identification. It then opens the text window and displays object information as in the following EINFO report on a selected line entity:

```
> EINFO
Select objects:
  1 selected.
Select objects:
*** 1 selected. ***
Type:      LINE
Layer:     0
Line type:  Layer's line type
Color:     Layer's color
Handle ID: "A"
Length:    0.44
Thickness: 0.00
Start Point (UCS):  5.09, 5.31, 0.00
End Point (UCS):   5.53, 5.35, 0.00
```



Delta (UCS):0.44, 0.04, 0.00  
 Start Point (WCS): 5.09, 5.31, 0.00  
 End Point (WCS): 5.53, 5.35, 0.00  
 Delta (WCS): 0.44, 0.04, 0.00  
 Projected length (in xy-plane): UCS: 0.44 WCS: 0.44  
 Angle in xy-plane: UCS: 6 WCS: 6  
 Angle to xy-plane: UCS: 0 WCS: 0

**Tip:** You may use this information in external reports, etc, by highlighting the text and using Windows' Copy (Ctrl C) and Paste (Ctrl V) key definitions to copy the text from the text window first to the Windows Clipboard (Ctrl C) and then to the external application (Ctrl V).

## Ellipse



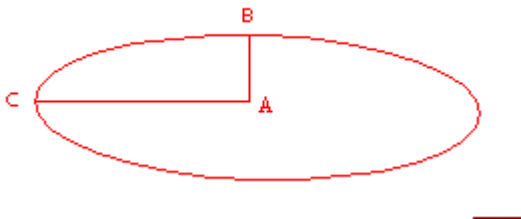
**Command: ELLIPSE**

**Alias ELL**

### Located:

1. **Pull-down Menu: Draw - Ellipse**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The command ELLIPSE allows you to draw a full ellipse.



The default is to identify *Center* point "A" of the ellipse first and then specify Point "B" (the first axis of the ellipse) Note that in this step, however, the orientation of the ellipse is established by either determining the location for "B" by picking it onscreen, or through distance/angle input as indicated below. Lastly, the secondary ellipse axis (A – C - at a 900 orientation from A - B) is input, either by picking point C, or entering the A-C dimension.

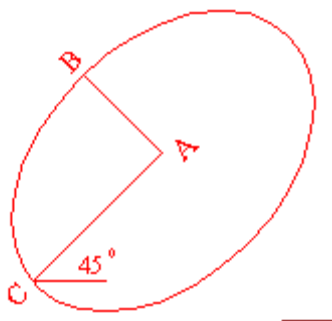
> ELLIPSE

Center point: Point A

Primary radius: Dimension A-B

Secondary radius: Dimension A-C

When you know the orientation of the A-B and A-C axes of the ellipse, you can specify this orientation when you input the primary A-B axis, by specifying the distance and angle of the A-B axis in relative terms, using the format (@distance<angle).



> ELLIPSE  
 Center point:  
 Primary radius: @3<45  
 Secondary radius: 2

In this example, pictured above, the A-B alignment (3 units) has been rotated 45 degrees to the left, as has the C-D alignment.

Note: The ellipse will be created as a closed polyline with straight-line segments.

## Command ERASE



### Command ERASE

Alias E

#### Located:

1. **Top Function Bar**
2. **Entered Directly at the Command Line**

Command ERASE is essentially a duplicate function to the DELETE command.

During the creation of drawings, or, later, the editing, changing or detailing of drawings it may be necessary to erase or delete certain parts of the drawing.

This command is called by entering ERASE in the command line area.

After calling the command the request to select the objects to be deleted will appear. The program offers (in the option bar) a series of methods for object selection.

For a single deleting operation, as many objects can be selected the user desires. When the object selection is complete the selected entities will be deleted without another confirmation, by simply pressing ENTER.

## Exiting the Program

**Command: EXIT**

Alias X

#### Located:

1. **Pull-down Menu: File – Exit**

## 2. Or Enter Command or Alias at the Command Line

It is very important to end every working session correctly (otherwise you might lose work that you have performed on a drawing file). The best way to terminate the session is by executing the command EXIT in the menu *File*.

The command EXIT is correctly performed in any of these fashions:

1. Select the command Exit in the pull-down menu File and confirm.
2. Press the keys Alt + F4
3. Double-clicking to the system menu area (see Windows manual)

If there are open drawings that have not been saved, the program will send a safety question. If you confirm, the dialog *Save File As* will be called.

After saving the drawings (see page 7) the current working session with the program will be finished.

---

### See also:

CLOSE

SAVE

SAVE AS...

Expand



**Command: EXPAND**

**Alias EXP**

### Located:

1. **Pull-down Menu: Modify - Expand**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Expand** allows the user to extend objects to a selected common boundary edge. In this way, it is almost identical to the EXTEND command.

During the execution of the command the drawing entities to be expanded are extended. They do not change direction. A line will remain a line, an arc will remain an arc. For chains and polylines, only the last segment will be extended to the selected boundary.

Select the entities that are to serve as boundary edges for the entities to be expanded. These boundary entities will remain unchanged. Object selection functions are presented in the Option Bar.

Next, select the entities to be expanded. In doing this, make sure that the selecting window of the cursor is on the part to be trimmed.

The expansion of selected entities is always executed from the endpoint that lies nearest to the cursor when it selected the entity.

If more than one boundary edge has been selected, the entity is expanded to the first boundary edge first. "Picking" a second time will result in the expansion being continued to the second boundary edge.

## Explode Complex Objects...



**Commands: EXPLODE, XPLODE**

## Alias EX. XPL

### Located:

1. **Pull-down Menu: Edit – Explode Complex Objects**
2. **Palettes: Edit and Parts**
3. **Or Enter Command or Alias at the Command Line**

Parts and other complex objects, like polylines, crosshatches, and dimensioning, inserted in the drawing constitute one single drawing object.

In this mode, it is not possible to edit any of the individual entities that make up the part. If, for any reason, there is a requirement to edit one of the entities comprising a part, then the part must be "exploded" (broken into its individual entities), the element(s) edited, and then the component parts reassembled again.

The explosion of complex objects is executed by the command EXPLODE, or by selecting and activating the option **Explode Complex Objects...** in the *Edit* menu.

When the object selection is complete, the objects (parts, polylines, hatches, or dimensions) will be exploded.

Now you may choose any of the entities for individual editing.

### Notes:

1. It is important to realize that, if there are attributes attached a part to be exploded, these attributes will be lost. Instead, the attributes defining names will appear.
2. Exploding a part does not change the part definition in the drawing. Another part or parts with the same name and original definition can be re-inserted into the drawing at any time.
3. To change the part definition in the drawing, you must modify the part components after exploding, then create a new internal or external part with the PARTEXP or PARTDEF commands, using the old part name. The program will ask you to confirm the re-definition when you do so.

## Command EXTEND

### Command: EXTEND

### Alias ET

### Located:

1. **Entered Directly at the Command Line**

### See Also: Command EXPAND

The EXTEND command is used to simply extend one linear entity until it reaches another selected entity. The intersection of the two becomes the new terminal point for entity #1. This command can be executed either from the command icon on the Modify palette, or else entered at the command line.

### Notes:

1. For this command to work, the intersection point must lie between the existing terminal points of the object selected as the border edge.
2. The "fence" object selection is very useful when extending multiple entities to a common boundary.

Sample input :

> EXTEND

Select entities for border edge (line, circle, arc, 2D polyline): (pick)

1 selected.  
Select entities for border edge (line, circle, arc, 2D polyline):  
\*\*\* 1 selected. \*\*\* (pick a second border if necessary) or simply press Return to advance  
Select entity to extend (line, arc, 2D polyline): (pick – extension to boundary occurs)  
Select entity to extend (line, arc, 2D polyline): (pick 2nd object – extension to boundary occurs)  
Etc.

See Also: Commands EXPAND and FILLET

## "F"

### Compiling a Source Shape File to a Binary Font File

**Command:** FCOMPILE

**Alias** FC

**Located:**

1. **Pull-down Menu: Detail – Compile Font**
2. **Or Enter Command or Alias Directly at the Command Line**

This program uses binary font files that permit defining text styles in the drawing (see command FONT).

Defined text styles can then be set and used with the commands TEXT, TEXTEDIT, ATTDEF, ATTDEDIT.

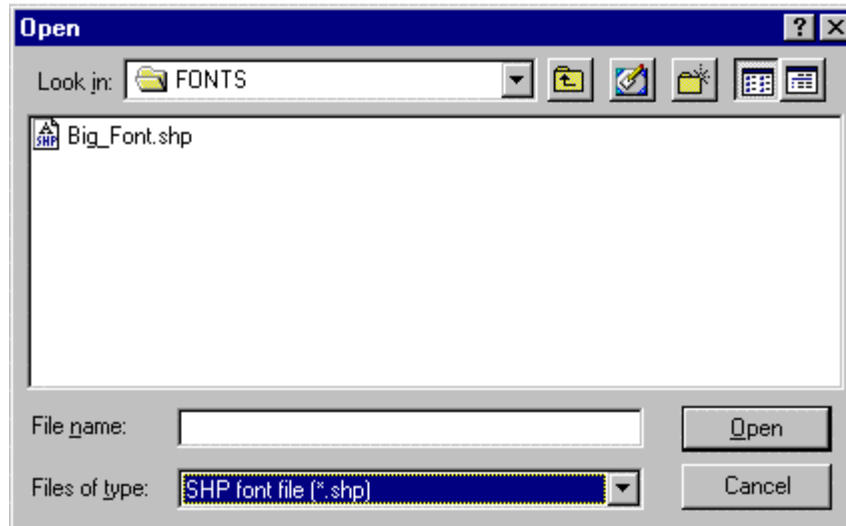
Binary font shape files (*.fsh*) provide less loading time and improved memory usage. A number of these FSH files are delivered with the program.

Application-specific or customized font shapes can be defined in plain text in **.shp** or **.fnt** shape files.

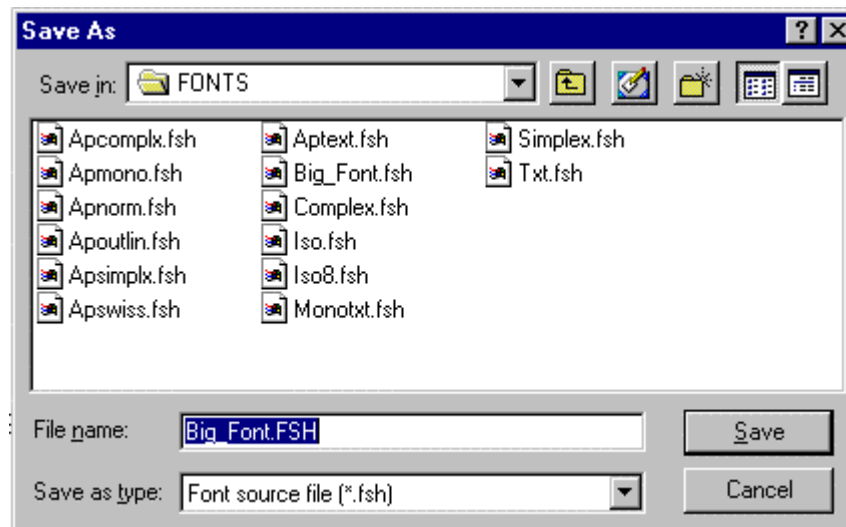
Before they can be used in a drawing, **.shp** or **.fnt** files must first be compiled.

**To convert a source shape file to a binary font file**

1. Call the command FCOMPILE



2. Specify the *.shp* or *.fnt* file name in the file dialog box



3. In the next dialog box, enter the name of the *.fsh* binary font file to be created for later use.

---

**See also:**

FONT  
QFONT

## Fast Hide



**Command: FHIDE**

**Alias FHD**

### Located:

1. **Pull-down Menu:** View – Hidden Lines – Fast Hide
2. **Palette:** View
3. **Or Enter Command or Alias at the Command Line**

In contrast to the HIDE command, which creates a new drawing, the command Fhide displays a hidden line view in the current drawing.

This representation is maintained until the drawing window is refreshed by a REDRAW, REGEN, ZOOM, or PAN command. This hide command offers you a visual impression of the 3D model from the current viewpoint.

Note that you can continue to draw in this display mode, even utilizing the object snap functions.

The command Fhide is based on the OpenGL system from Microsoft that allows high performance hidden line removal and rendering.

---

**See also:**

HIDE

Command DVIEW

DRAWORDER

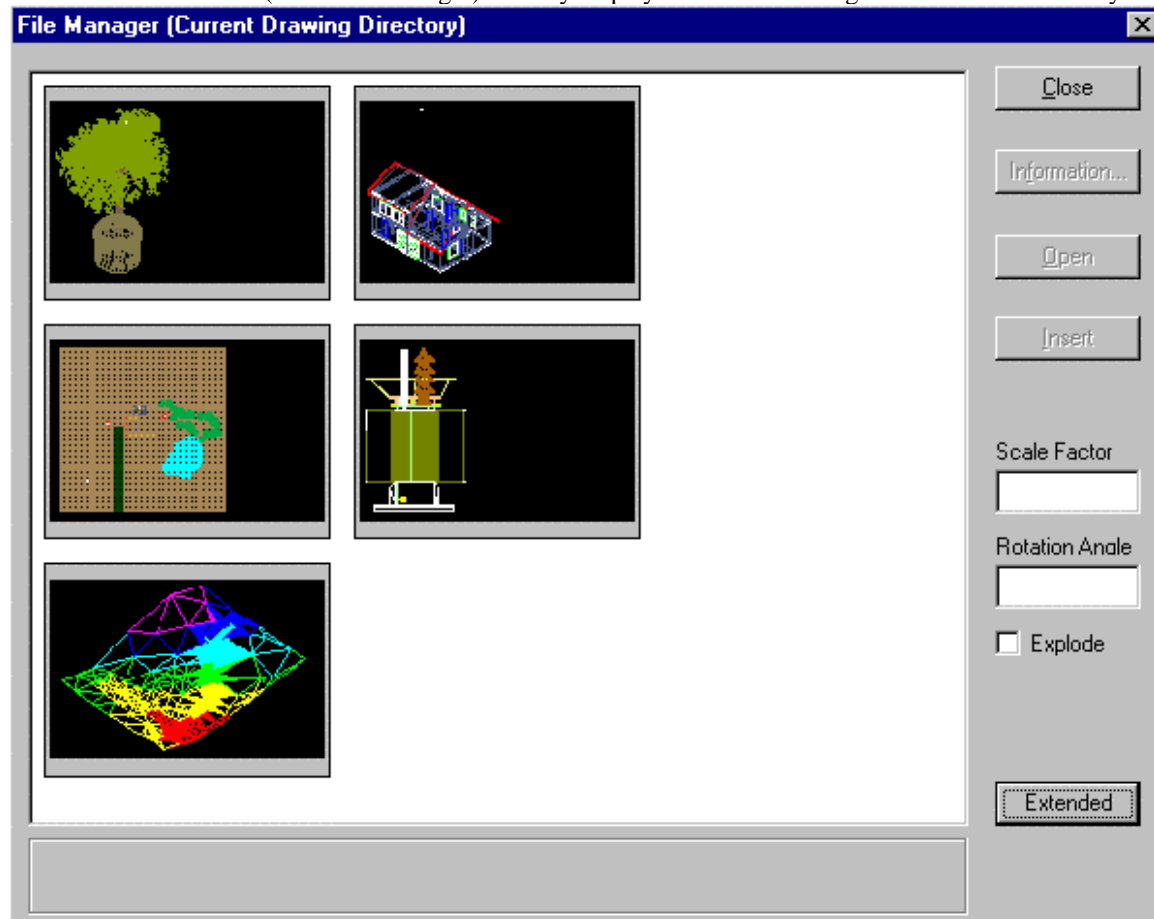
File Manager

**COMMAND: FILEMAN**

**Located:**

1. **Pull-down Menu: File – File Manager**
2. **Or Entered Directly at the Command Line**

The Fileman command (File>File Manager) visually displays all .FLX drawings in the current directory.



## Fillet



Command: **FILLET**

Alias **F**

### Located:

1. **Pull-down Menu: Modify – Fillet or  
Modify – Edit Polyline – Fillet Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Often two solid edges do not intersect exactly, in acute or right angles. With the Fillet command solid edges can be filleted quickly and precisely.

The first prompt will request the selection the two lines whose intersection is to be filleted. These can also be lines of an object such as a rectangle.

### Options:

The following options will be shown in the Options Bar.

#### Polyline

Instead of placing a radius between two separate linear entities as with the standard FILLET command, FILLET-POLYLINE will apply the radius at all vertices in the selected polyline. Note: this command acts upon a polyline only once. To change the radius you must UNDO the first fillet command or else reconstruct the polyline by exploding it, deleting the arcs and INTERSECTing the segments and then using POLYEDIT-JOIN to reconnect the segments to form the original polyline.

#### Radius

The option **Radius** will defined the radius value for the fillet. The program will always offer the value of the last radius used in the current command operation. A radius value of zero means no radius will be applied to the fillet and that entities will simply be trimmed off where they cross each other.

#### Trim (Off)

**Trim** functions as a switch. It decides whether the parts of lines that lie beyond the fillet are erased or whether they will be left as separate entities.

If the switch in the Options Bar displays the term **Trim**, you can **switch off** the function **Trim** by clicking the switch. It will then change to **Trim (Off)**.

If the switch in the Options Bar displays **Trim (off)**, by clicking the switch the function will change to **Trim**.

#### Flip



Command: **FLIP**

Alias **FL**

### Located:



1. **Pull-down Menu: Modify - Flip**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command **Flip** contrasts to the **Mirror** command, in that the original object does not remain in its original position; it is simply "flipped" about an axis specified by the user.

The selected objects have, after "flipping", the same relative position to the symmetry axis as they had prior to "flipping", but are now "flipped" in a new direction.

The reflection axis can be defined freely, or parallel to the X-axis or the Y-axis

The first prompt will be for object selection.

Next, select one of the options for choosing the axis from the option bar:

#### **Axis**

The option **Axis** allows the user to freely define the flip axis. It is the standard option for axis definition, and is used when no other option has been selected.

Define the position of the axis by the indicating two points or by the input of their coordinates. The axis can have any alignment toward either the objects or the axes of the coordinate system.

If the mirror axis should run through any of the points of existing geometry of the object(s), you should specify object snap functions to define the points precisely.

#### **Horizontal Flip**

To flip the marked object about a horizontal axis choose the option **Horizontal**. This option creates a flip axis that runs parallel to the Y-axis of the coordinate system. Only a single point is required to define this axis.

#### **Vertical Flip**

To flip the marked object about a vertical axis choose the option **Vertical**. This option creates a flip axis that runs parallel to the X-axis of the coordinate system. Only a single point is required to define this axis.

### **Command FLXRECOVER**

**Command: FLXRECOVER**

**Alias REC**

#### **Located:**

1. **Pull-down Menu: File - Recover**
2. **Or Enter Command or Alias at the Command Line**

Beginning with the version 4.01 (Service Pack2) release, the FLX drawing format modifications permit the partial recovery of damaged drawing files. This utility, called by the FLXRECOVER command, is therefore available to FLX files saved in this (or later) release format. For damaged drawing files saved in AutoCAD DWG or DXF format, applicable AutoCAD recovery utilities should be used.

Usually, the need to use this utility becomes apparent when you experience difficulties opening an existing drawing file. File damage may apply to only a portion of the drawing database and the recovery utility will help you to salvage undamaged drawing elements from the file.

To use the recovery routine, simply issue the FLXRECOVER command from the pull-down menu selection or at the command prompt and select the damaged filename in the dialog presented.

## Font... (Text Style Definition)



Command: FONT

Alias FT

### Located:

1. Pull-down Menu: Detail – Fonts
2. Palette: Detail
3. Or Enter Command or Alias Directly at the Command Line

The command FONT loads fonts from the font files and customizes them to the requirements of the drawing.

Fonts define the shapes of the text characters that make up each character set. In AllenCad, you can use:

1. AllenCad's own compiled shape (FSH) fonts,
2. AutoCAD's own compiled shape (SHX) fonts
3. True Type fonts.

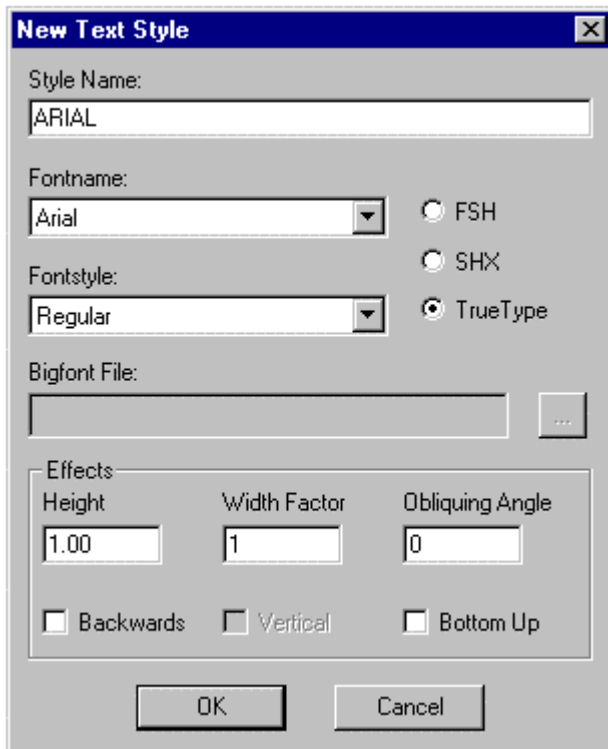
All installed True -Type fonts, provided with your Windows system, are listed by font family name.

SHX fonts and FSH fonts are listed by the name of the file in which the fonts are stored.

All fonts to be used in the program must first be assigned to a text style. When you give the FONT command, you are therefore presented with the following dialog:



Here, you create a new text style or choose an existing one to modify, by assigning a different font. With the required style name highlighted, press OK and you will be presented with the font dialog:



In this case, we have chosen to create a new style called Arial and to apply the Arial true type font to this style. To do this, we click on the True Type radio button and select the down arrow on the Fontname box to display the available choices. We scroll to Arial and select it.

Next, we may need to select a fontstyle. Again we click on the down arrow in the fontstyle window and choose regular text, as we do now wish it to be bold, italic or otherwise.

If we wish to use a Bigfont file, we press the browse button to the right of the window and select our bigfont file from the available choices.

Other text effects are available at the bottom of the dialog. We can assign a text height, width factor or Obliquing (Tilting) Angle. If we want the text to run right to left in a mirror image, we can select the Backwards option. The Bottom Up option will give us text that is upside down. .

**Note:** Some type fonts do not support all of these options. For instance, some fonts can be aligned vertically. On others, there is no change in aspect ratio permitted.

Confirm the settings of the properties by selecting OK. Fonts added or changed in this manner can now be used for creating or editing text objects when the modified text style is current.

**Note:** Defined text styles are saved with the drawing file only with a reference to the corresponding font file. They are available any time that the drawing is opened, if the corresponding font file is found in the programs directories.

---

#### See also:

Adding Additional Fonts

Command QFONT

Command TEXT

## Command FREEHAND

## Command FREEHAND

Alias FH

Functional in: AllenCad CE Pro **YES**

AllenCad CE **YES**

Located:

5. **Draw Menu > MarkUp > Freehand Sketch**
6. **Left Side Bar**
7. **Palette Draw**
8. **Enter Command or Alias Directly at the Command Line**

The FREEHAND command gives the operator the ability to simply draw 3D polylines that track cursor movement while the mouse button is depressed. In this way, it can be used for quick insertion of handwritten notes or linear polyline drawing. The resulting entities are drawn on the current layer in red to conform with redline markup convention. The resulting entities are LWPolylines with a specified display width equal to 1/100 of the screen size. The POLYEDIT command can be used to subsequently set a specified width and the PROPEdit command used to change color, as necessary. FREEHAND is similar to the SKETCH command, except that with SKETCH you are prompted for a segment length that can provide a measure of smoothness to the resulting polyline. The command is repeating, when you use the left mouse button to begin the line and the right button to terminate it. You can use the Esc key, Cancel button or issue a different command to terminate Freehand drawing.

## "G"

Grouping Entities



**Command: GROUP**

Alias G

Located:

1. **Pull-down Menu: Parts – Group**
2. **Palette: Parts**
3. **Palette Edit**
4. **Or Enter Command or Command Alias Directly at the Command Line**

Drawing entities can be combined in named Groups. Essentially, groups are a named selection set, and as such, the user is able to manipulate all entities within the Group in a single operation (Moving, Rotating, Mirroring, or Scaling, for example) just by specifying the group name. Single entities of a Group may still be *edited* individually (for example, intersecting or stretching). Entities may be removed or added to a Group at any time. An object may be a member of more than *one* Group. *Groups* may be contained in *other* Groups. Group definitions are stored in a distinct table of the drawing database. Groups are maintained in this fashion from one drawing session to the other.

### Differences Between ‘Parts’ and ‘Groups’

A Part (a block) has its own insertion point. A Group has none. A Part can occur multiple times within the drawing. If a block is redefined, all of its occurrences are updated. Groups are not treated like that. Groups can, however, be copied. A Group that originates from another Group that has been copied is a separate

union of entities. Each Group has a unique name.

**Note:** If an entity that is member of a Group is included in a part definition, the entity is deleted from the drawing and removed from the Group.

### The Command GROUP

To group entities, or to manage and manipulate Groups, use the command GROUP. The options within the command are:

<b>Create</b>	Brings entities into a newly named Group
<b>Explode</b>	Abolishes a Group definition
<b>Rename</b>	Alters the name of an existing Group, especially of anonymous groups
<b>Add</b>	Adds drawing objects to an existing Group
<b>Remove</b>	Removes drawing objects from a Group
<b>Selectable</b>	Determines whether a Group is selectable or not

### Creating Groups

To create a group, first determine a unique name. The name will be required to identify the group later for a number of command options. In addition, specify whether the group should be selectable or not (see below).

#### To define a new group,

1. Call the command GROUP with option Create,
2. Specify a unique group name (max. 31 characters),
3. If desired, enter a description (max. 64 characters),
4. Determine whether the group should be selectable or not (the default is Yes), and
5. Select the drawing objects that should belong to the named group.

To create a group using Command Line:

```
> GROUP
Select option: Create
Group name: G1
Description: Screw
Should the group be selectable (Yes/No)? Yes
Select objects: Select the entities that should belong to the group
```

### Exploding Groups

Group definitions may be abolished by using option **Explode** of the GROUP command. When a group definition is dissolved the corresponding entry in the group table is deleted in the drawing database.

The only way to abolish a group definition is by abolishing its name.

To explode a group,

1. Choose the command GROUP with its option *Explode*,
2. At the prompt "Group name: " enter the name of the group, to be exploded.

Because a single entity may belong to multiple groups and groups may be part of other groups, the **Explode** option does not allow the user to pick an entity within a group to explode the group.

### Renaming Groups

The name of a group may be changed by using the GROUP command **Rename**. It is useful to employ this method when "anonymous" groups that have been created by copy commands exist. The system will automatically apply anonymous names to copies group (\*\*A1, \*\*A2, etc.) that exist through the use of one of the commands like COPY, MIRROR, or ARRAY.

To rename a group,

1. Call the command GROUP with its option Rename;
2. At the prompt "Current group name: " enter the name of the group (for example "\*\*A0");
3. At the prompt "New group name: " enter the name of the specified group (for example "Screw\_Copy1")

### Add / Remove Entities

Two other options of the command GROUP are provided to allow the user to integrate additional entities into an existing group or to remove entities that are currently in the group.

After a group name is specified, the entities that belong to the group are highlighted.

► To add entities to an existing group, simply 'pick' the entities that are to be added (they will *not* be highlighted).

► To remove entities from an existing group, 'pick' the highlighted entities that are to be removed

**Note:** Should *all* entities be deleted from a group, remember that *the group will still exist as long as it has a name!*

### Working with Groups

Editing commands will treat all entities in a group as a single object. Commands like MOVE, ROTATE, or COPY will act on the entire group.

If the user should select an object for one of these commands at the prompt that selection will lead to the automatic selection of all group entities.

A typical prompt of a command like this would be:

Select objects:

The rule is: In commands that allow **multiple selection**, identifying a *single* entity of a group is a selection of *all* entities belonging to the group(s). When the user is *prompted* to select multiple objects, he or she may choose the object selection option *Group* from the options bar, and enter a group name to select all of the entities belonging to the group. Groups, therefore, can also be named selection sets.

Other MODIFY commands (Offset, Lengthen, Intersect, Trim, etc.) request a single entity in one or more steps (usually to modify the geometry of drawing entities). A typical prompt for a command like this would be:

Select entity (Line, Circle, Arc, 2D-Polyline):

In this case **single selection** is being used. Any time the user is prompted to select a single entity, the group does not matter. These commands modify the geometry of single selected entities even though those entities belong to a group. The modified entities will remain members of the group. This feature makes allows great flexibility in working with GROUPS and provides significant advantages for the user compared to using BLOCKS.

The following table summarizes the editing commands and describes how entities that are member of a group will be processed when using each command.

#### DELETE

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. When a single entity of a group is deleted, the entity is removed from the group definition.

## MOVE, ROTATE, SCALE, FLIP

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used.

## MIRROR, COPY, ARRAY

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. The system will apply anonymous names to copied groups ("\*A1", "\*A2", etc.) that are created by the use of these commands.

## STRETCH

Those entities belonging to a group that have been selected using the object selection modes *Crossing* or *Cpolygon* will be stretched.

## TRIM, EXPAND

The prompt to "Select cutting edges" or "...boundary edges" will include all entities of a group, unless the Object Selection Mode *Single* has been used.

## OFFSET, FILLET, CHAMFER, LENGTHEN, INTERSECT

These commands always prompt the user to select a *single* entity to be modified. That an entity belongs to a group does not matter in these operations. Single entities altered by these commands will remain in the group. If *new* entities *originate from these commands* they will *not* be automatically included in the group.

## DELPARTIAL, REJOIN

When an entity (line, circle, arc or 2D-Polyline) is broken into two or more entities by partial deletion, only one of the entities will remain in the group. When an entity that has been broken into two or more pieces is rejoined, the newly created entity is no longer member of any group, even if one or more of the pieces was part of a group.

## PARTDEF, HATCH, HATCHEDIT

Selecting a single entity will select all entities of the group, unless the Object Selection Mode *Single* has been used. This also applies to the command HATCHEDIT when the user is prompted to select additional objects: to be included to the set of entities for associative hatch editing.

## TEXTEDIT

The text entity retains its association to its group if the command TEXTEDIT is used to modified text.

# Command GRAPHSCR

## Alias GS

### Located:

#### 1. Enter Command or Command Alias Directly at the Command Line

This command is provided primarily for use by programmers. It explicitly calls the program's graphics screen into view. Users will normally use the `tscreen (toggle screen display) or more simply press the F2 key to toggle between text and graphics screens.

# "H"

Hatch...



Command: HATCH

Alias H

**Located:**

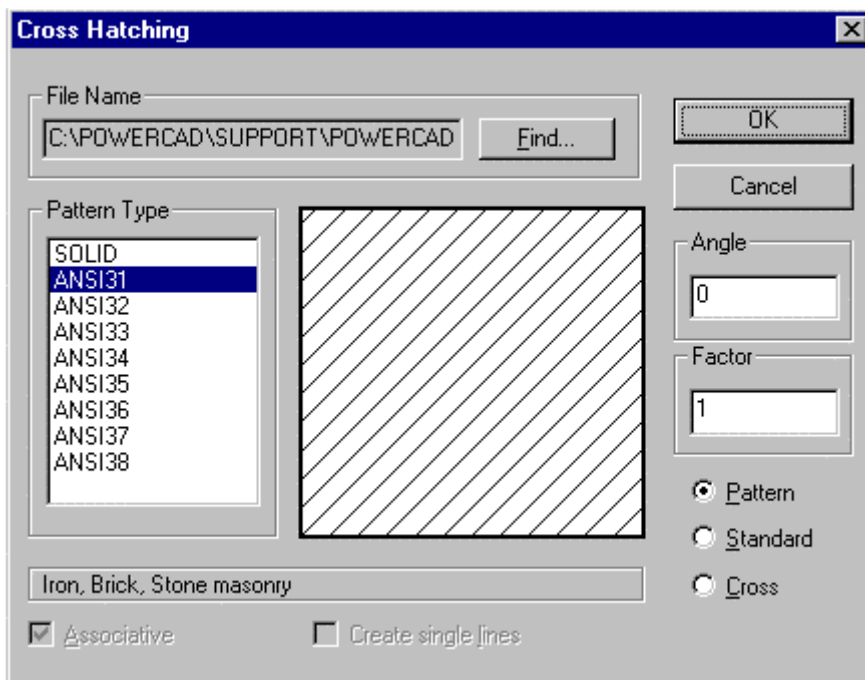
1. Pull-down Menu: Detail – Hatch
2. Left Side Bar
3. Palette: Detail
4. Or Enter Command or Alias Directly at the Command Line

Hatching (or cross-hatching) a drawing adds meaning to the drawing, and helps to differentiate the materials, the areas, or any of a number of other unique attributes. Both the HATCH and the BHATCH commands enable you to fill in areas of your drawing with either a solid color or a distinctive pattern.

**Command Execution**

To execute the command, you are first prompted for the boundary this is typically some closed entity, such as a rectangle, circle or closed polyline. **Note:** If only part of these selected entities forms the boundary, you should instead be using the BHATCH command.

Having selected the boundary, you are next presented with the dialog in which you select the pattern to apply, as well as pattern parameters:



Hatch patterns are stored together in **.pat** files. The file name input at the top of this dialog allows you to select a pattern from any of a number of .pat files that you may have available. Having selected the file, all patterns included in that file are listed in the Pattern Type window on the left. Clicking on any of these causes a representation of that pattern to appear in the central window, making selection easier.

The selections to the right on this dialog modify the pattern, in which:



- Angle – Will cause a counter-clockwise rotation of the pattern by the angle entered;
- Factor – Will cause the pattern to become more condensed or expanded, depending upon the entered factor value;
- Pattern will display a non-uniform pattern (if that is available in the particular selection);
- Standard – Will modify the pattern to a uniform spacing and
- Cross – Will create a "criss-cross" pattern effect.

The hatch will be applied immediately after the OK button is pressed.

### Hatch Color

Hatch color is, as with other drawn entities, determined by current layer color setting, or else by a set color that over-rides bylayer color settings. Color fills are achieved by making the selection of "Solid" in the pattern name window.

Further information on hatch pattern definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

---

#### See also:

Boundary Hatching  
 Modify Hatching  
 Polyline Boundary

## Command HELP



### Command HELP

Alias ?

#### Located:

1. Pull-down Menu: Help – ?
2. Left SideBar
3. F1 Key (for Contextual Help)
4. Or Enter Command or Alias at the Command Line

The command HELP is used to call Online Help whenever assistance with commands and concepts used in the program may be needed. Online Help can also be called by entering a question mark "?" at the command line, or by pressing Function key #1. The F1 choice will provide "context-sensitive" Help, in that the program senses what you were doing before the F1 key was pressed and it attempts to provide you with specific explanations to those functions from the Help file. This can save considerable time, especially for new users.

### Fast Hide



Command: FHIDE

Alias FHD

#### Located:

1. **Pull-down Menu: View – Hidden Lines – Fast Hide**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**

In contrast to the HIDE command, which creates a new drawing, the command Fhide displays a hidden line view in the current drawing.

This representation is maintained until the drawing window is refreshed by a REDRAW, REGEN, ZOOM, or PAN command. This hide command offers you a visual impression of the 3D model from the current viewpoint.

Note that you can continue to draw in this display mode, even utilizing the object snap functions.

The command Fhide is based on the OpenGL system from Microsoft that allows high performance hidden line removal and rendering.

---

**See also:**

HIDE

Command DVIEW

DRAWORDER

**Modify Hatching...**



**Command: HPEDIT**

**Alias HED**

**Located:**

1. **Pull-down Menu: Detail – Modify Hatching**
2. **Left SideBar**
3. **Palette: Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

You can exchange an existing associative hatch using this command.

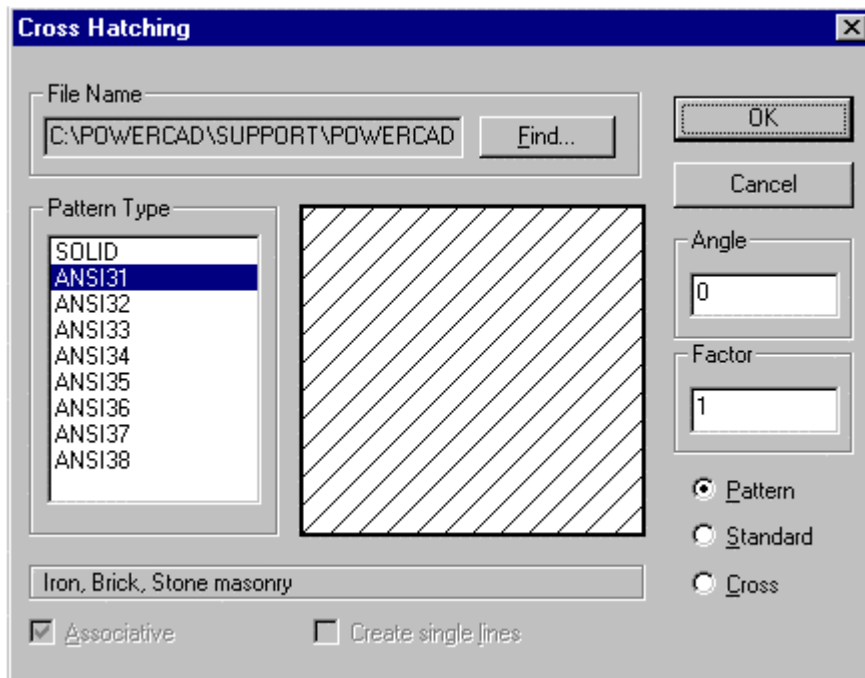
It also allows you to add additional objects to the set of entities to be hatched, as well as change the hatch pattern.

> HPEDIT

Select an associative hatch:

Select additional elements? <N>:

The routine then displays the Hatch Pattern dialog box to select the type of hatching desired.




---

**See also:**

Command QHPEDIT

QHPEDIT is the command line version of the HPEDIT command provided for utilization in Lisp routines, macros, etc.

## Command HPUPDATE

**Alias HPU**

**Located:**

1. **Enter Command or Command Alias Directly at the Command Line**

This command allows users to apply hatch pattern changes (made with command HPEDIT) to existing drawing hatches.

## "I-K"

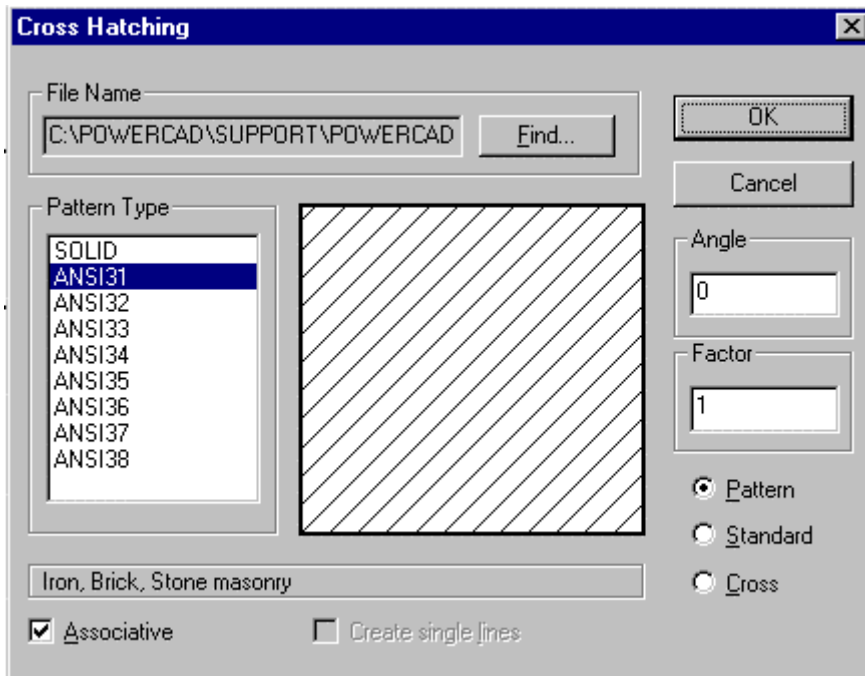
Icon Menus

**Command ICONMENU**

**Located:**

1. **Pull-down Menu: Tools – Resource Manager - Icon Menu...**

The Icon menu command is a programming tool that presents the user with selections that are supported by images. Icon menus use groups of images as graphic labels. Icon menus also have a list box containing text labels associated to each image. This combination allows the user to select options



Icon menus are a valuable visual tool, as in the example above, when selections of hatch patterns or text styles should be customized. Also, icon menus can be useful in creating symbol library selection menus.

With this command, a dialog presenting an assortment of graphical options can be created. A slider bar is added automatically to the dialog if the number of images exceeds display limits.

You can customize the number of columns and rows to be displayed for each icon menu.

## Image Library Tool

### Command IMAGLIB

#### Located:

1. Pull-down Menu: Tools – Resource Manager – Run Image Library Tool
2. Or Entered Directly at the Command Line

The Image Library Tool assembles and organizes images that can be used in several ways. Icons and other screen grabs incorporated in image libraries can be displayed in icon menus, toolbars, palettes, and application dialog boxes. Image libraries created and maintained with the ImageLib program are stored in files with extension *.ilb*.

AllenCad's image libraries are similar in nature to AutoCAD's slide libraries. You can import entire slide libraries (*.slb*) from AutoCAD into a AllenCad image library. But AllenCad's Image Library Tool provides features not available in AutoCAD's slide libraries:

- Beneath slides, you can store bitmaps (*.bmp*) and Windows Metafiles (*.wmf*) in image libraries.
- You can use icons and images from your libraries in icon menus, dialog boxes, palettes, and toolboxes.
- The ImageLib program is a Windows program with easy-to-use import and export of image files.

To start the ImageLib program, select File > Resource Manager > Run Image Library Tool from the AllenCad menu bar.

For more help with the Image Library click here: [IMAGELIB HELP](#) >>

## Insert Part ...



Command: INSERT

Alias I

### Located:

1. Pull-down Menu: Parts – Insert Part
2. Palettes: Draw and Parts
3. Or Enter Command or Alias Directly at the Command Line

Command: QINSERT

Alias QI

### Located:

1. Pull-down Menu: Parts – Quick Insert Part
2. Or Enter Command or Alias Directly at the Command Line

The command INSERT offers broad support for the user in its Dialog Box. The command QINSERT is the command line version of the command. The command macro QINSERT ? allows fast insertion of internal parts (parts already defined in the drawing).

The command INSERT provides selection areas for:

- Selection of external parts;
- Definition of the insertion point, scale factors, and rotation angle, with future choices of using coordinate/ numeric input or cursor selection;
- Parameter values may be reset; and
- The Part may be "exploded" into its components.

### Selecting a Part

To select the Part to be inserted, type the name of the Part (or select an existing Part) in the field Part Name. Use the buttons External and Internal to select the correct Part listing.

### Defining the Insertion Point

Defining of the Insertion Point either by entering the proper coordinate values, or by selecting Interactive, and "picking" the point with the cursor.

### Scale Factor

The Scale Factor input areas will offer 1.0 as the default XYZ scale factor. This will insert a part at its original size. Remember that a scale factor greater than 1.0 will increase the size of the part, and a factor less than 1.0 will reduce the size of the part to the percentage of the original size specified by the Factor. A factor of 2 will result in the part being twice it's original size, while a factor of 0.5 will be half original size.

By default, AllenCad prompts you for x and y scale factors only. If you need to specify a z scale factor, simply respond to the X scale factor prompt with an xyz response. You will then be prompted for all three scale factors in turn.

The option *Interactive* allows the user to define the size of the Part during the insertion as well. To

insert the Part using the Interactive option, simply define by opposite corners the rectangle into which the Part is to fit.

Since dragmode is active during this operation, the user will have the benefit of seeing the selection results before confirming the selection.

### **Rotation Angle**

The Rotation Angle can be set from the Dialog Box. Enter a value for the Rotation Angle in the proper input window. For counter-clockwise rotation enter a positive value, for clockwise rotation enter a negative value between 0 and 360 degrees

### **Default**

Default allows the user to reset the parameters for the Insertion Point, the Scaling Factor and the Rotation Angle back to the original parameter values.

### **Exploded**

Exploded allows the user to "explode" the part into its component entities. Exploding will cause the loss of the characteristics that were retained as a part.

### **Troubleshooting**

If you find that the blocks you are inserting are not placed in the specified location, check the settings for INSBASE and ELEVATION in both drawing files. These should be set to 0,0,0 and 0 respectively, if you do not want them to affect your block insertion.

## **Command INSERTQ**

### **Command: INSERTQ**

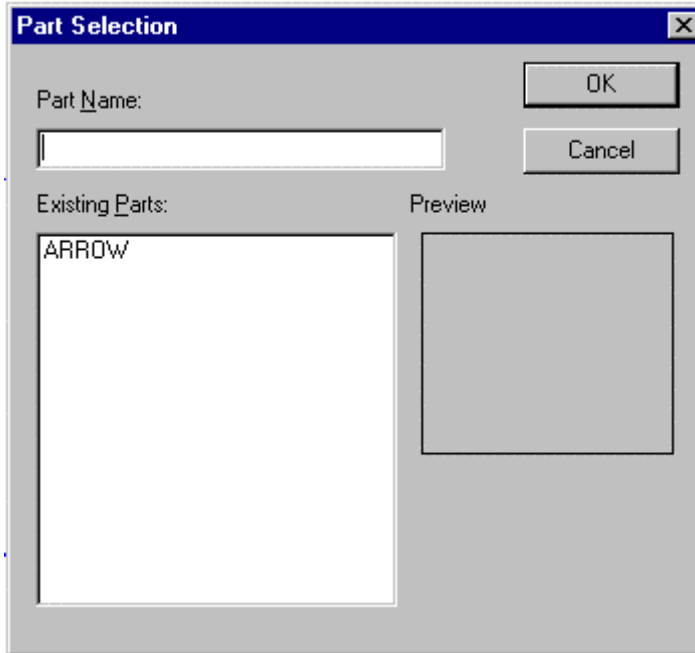
#### **Located:**

- 1. Pull-down Menu: Parts – Paste Internal Part 1:1**
- 2. Or Enter Command or Alias at the Command Line**

The INSERTQ command is used to insert a part into the drawing that has already been defined in the drawing database. In other words, it has been inserted or defined beforehand and is a known entity.

The command can be issued from the "Parts – Paste Internal Part 1:1..." pull-down menu, or entered at the command line, as you prefer.

When executed, the command brings up the following dialog:



Here, you will see in the lower left window, a listing of all predefined parts available for insertion. Clicking on one of them produces an image of the part in the right window, for confirmation. When the part to insert has been correctly identified in the Part Name window, pressing the OK button will begin the insertion routine. You are asked to identify the insertion point, however you are also given a dialog in which you can specify a scale factor or rotation angle other than the defaults as recorded in the Part definition. Unless these options are selected, the insertion point terminates the command with the part being inserted at the default 1:1 scale and in the original orientation.

### Troubleshooting

If you find that the blocks you are inserting are not placed in the specified location, check the settings for INSBASE and ELEVATION in both drawing files. These should be set to 0,0,0 and 0 respectively, if you do not want them to affect your block insertion.

## Command INSBASE

Applies a constant offset to the base (insertion) point for parts.

Notes:

1. A value for INSBASE other than 0,0,0 will apply the designated coordinate adjustments to the insertion base point specified when inserting a part.
2. The command PARTEXP does not request a base point when creating a drawing file representing a part when the entity selection method is used.

### Properties

<i>Type</i>	3D point
<i>Default</i>	0.00,0.00,0.00
<i>Status</i>	Read / Write

### Command

PARTEXP. See also PARTDEF and ELEVATION

## Intersect



**Command: INTERSECT**

**Alias IN**

### Located:

1. **Pull-down Menu: Modify - Intersect**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Intersect** is a special case of trimming that causes intersecting lines to be trimmed following the intersection. Essentially this command is the same as issuing the FILLET command with a specified radius of zero.

Enter the command by selecting the option **Intersect** from the menu *Modify* or by typing the command INTERSECT.

Select the two lines to be trimmed. Make the selection by picking, and be sure to make the pick on the segment of each line that is to remain.

---

### See also:

TRIM

FILLETCommand FILLET

## Command 'ID



**Command 'ID**

**Alias ID**

### Located:

1. **Pull-down Menu: Edit – Identify Coordinate**
2. **Left SideBar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

The ID command is used to display the X, Y and Z coordinates of points selected on the drawing. The coordinates are listed at the command line.

Object snaps should generally be used with this command to ensure correct point selection.

## "L"

Command LAYCONF

**Command: LAYCONF**

**Alias LC**



### Located:

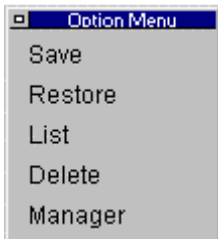
1. **Pull-down Menu: Tools – Layer Configurations**
2. **Or Enter Command or Alias at the Command Line**

The LAYCONF command is used to save and restore layer settings in and from a named status file.

As you work in the program, you will probably add new layers and change layer colors or other parameters using the Layer Manager. The LAYCONF command allows you to save layer settings at any time, in a named setting configuration file, and also lets you return to those saved settings at any later point by specifying the saved configuration file.

Note that the status of newer layers, that are not specified in the named configuration file are not changed in any way by a return to a previous layer status – only layer parameters for layers specified in the saved configuration are changed.

The command LAYCONF is typed in at the command line, at which time you see the option dialog:



You also see a corresponding prompt at the command line:

```
> LAYCONF  
Select option: Save/Restore/List/Delete/Manager:
```

At this point, you can make your selection by picking the option on the dialog with the mouse, or by typing in the first letter of the option you want at the command line.

The **Save** option will save current layer status in the filename that you specify. The default is "LC2" for the first saved file, "LC3" for the second, and so on. You can accept these default filenames by pressing the Enter key, or else you can change the name to another and confirm your choice by pressing Enter.

The **Restore** option is used when you want to revert to a previously saved configuration. You will be prompted for the filename, after which the layer status is changed back. Remember that if you think you will want to return at any time to the status you have prior to the reversion, that you should save that status first in a different configuration file.

The **List** option simply returns a list of all stored configuration files. This can be useful if you have forgotten the filename that you specified earlier when you saved.

The **Delete** option is used to delete a configuration file that you no longer need.

The **Manager** option simply calls the Layer Manager, which allows you to change current layer settings and add new layers.

### Layer Manager



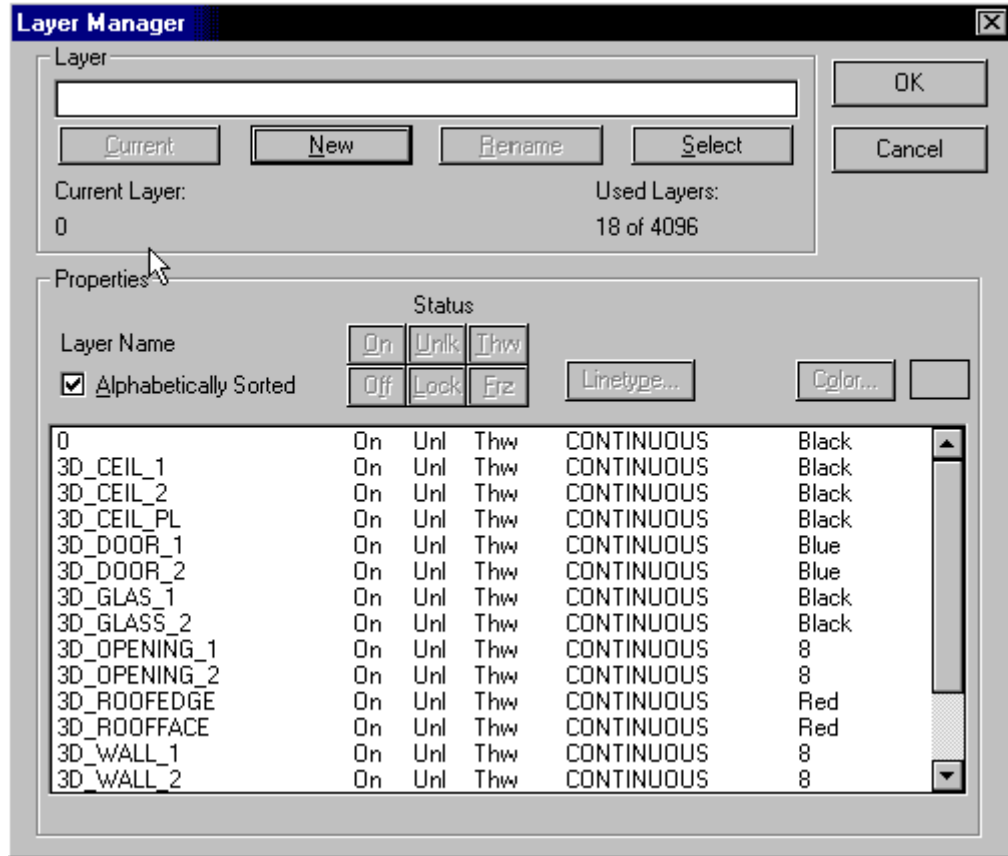
**Command: LAYER**

**Alias LA**

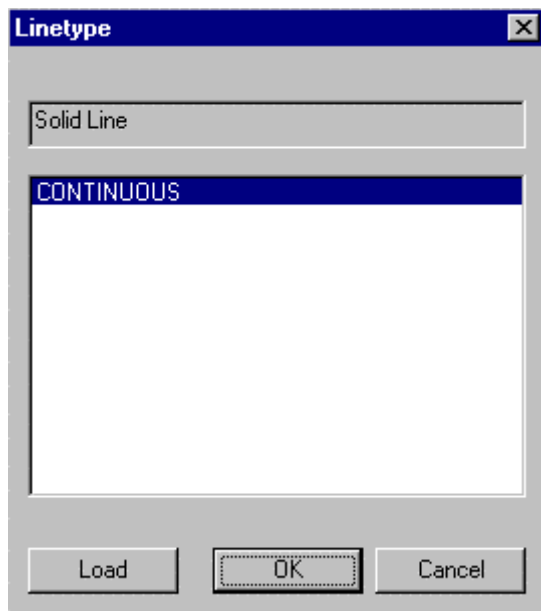
**Located:**

1. Pull-down Menu: Tools – Layer
2. Properties Toolbar
3. Or Enter Command or Alias at the Command Line

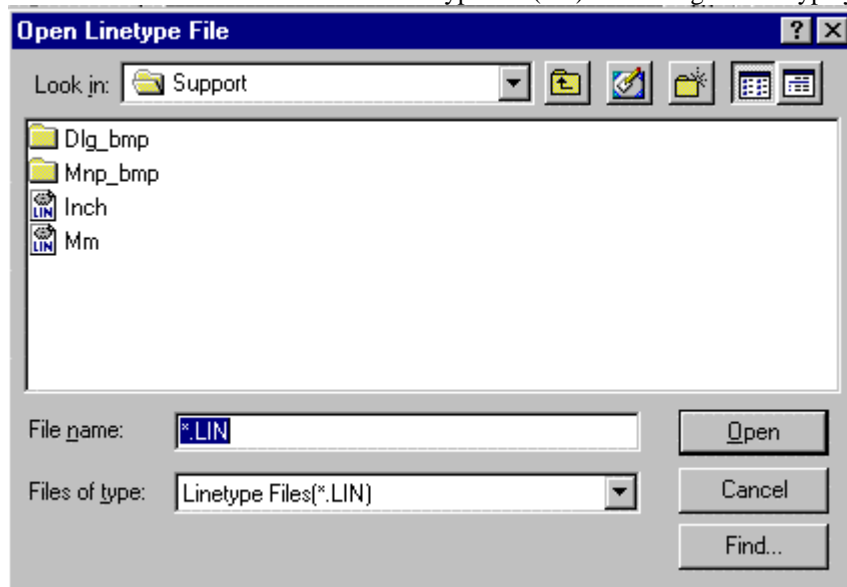
Creating new layers, choosing an active layer and controlling all previously described layer characteristics are accomplished through a dialog box. This dialog box is called *Layer Manager*.



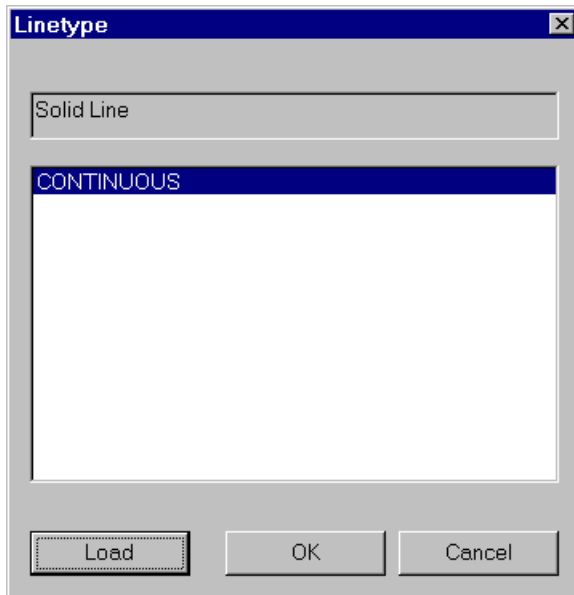
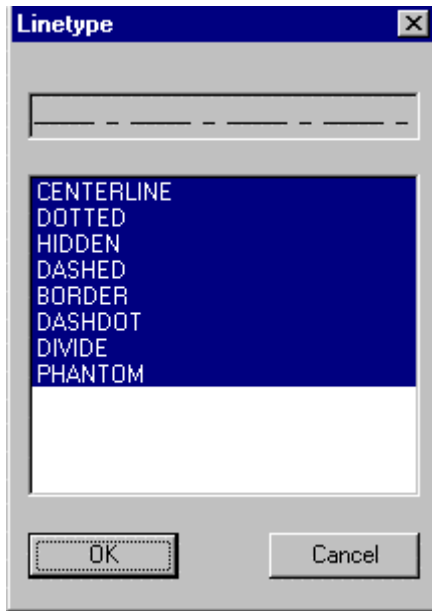
Line types may be loaded directly from the linetype dialog, in the Layer Manager dialog, by pressing the **Load** button



You will then be asked to select the linetype file (.lin) containing the linetype you want to open:



At this point, you are presented with the dialog below, showing all of the linetypes contained in the linetype file that you selected. As you select each linetype in turn, a representation of its appearance is presented in the window at the top of the dialog. Choose the linetype that you want loaded and press OK.



In the **Layer Manager** the default button is the **New** button. When you type in a layer name and press the return key, a new layer will be created and you remain in the layer manager.

#### **Current Layer and Used Layers**

In the upper rows of the *Layer Manager* Dialog Box is a field for the name of the active (current) layer (left window) and a field for the number of defined layers in this drawing.

#### **Layer**

Later rows contain combined view and entry fields for layers that still have to be worked on, and a selection box that allows the user to choose between creation of new layers, renaming layers and choosing an active layer.

#### **Properties**

There are fields for the layer state and for choosing line types and layer color.

Directly beneath that, in the dialog box, all the existing layers of the drawing are shown. Any of these may be selected and activated with the cursor. The layer color and linetype are also shown there.

The meaning of the abbreviations used are shown [here](#).

Abbreviation	Meaning
On	Visibility on - The layer becomes visible
Off	Visibility off - The layer becomes invisible
Lock	Status free, drawing elements can be edited
Unlk	Status closed, drawing elements cannot be edited
Thw	Layer thawed, visible and can be edited
Frz	Layer frozen, invisible and cannot be edited, will not be regenerated

## Set Layer by Reference

**Command: LAYREF**

**Alias LRF**

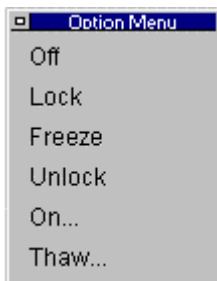
### Located:

1. **Pull-down Menu: Tools – Layer By Reference**
2. **Or Enter Command or Alias at the Command Line**

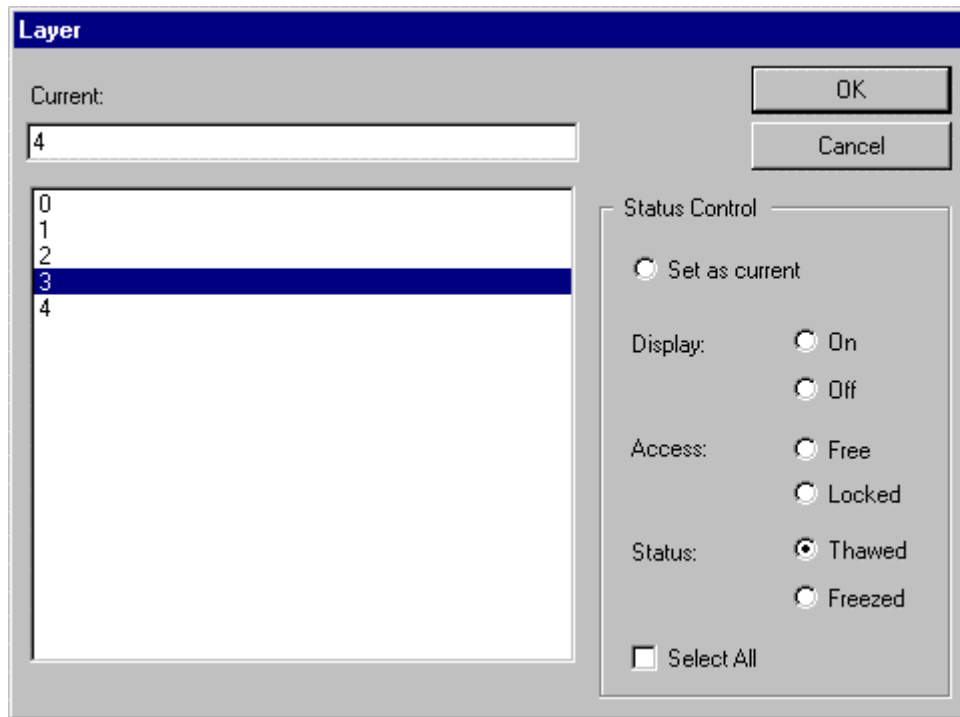
The LAYREF utility provides an alternate means of layer control that is, in some situations, more appropriate and direct. The user can set, turn off, freeze, lock, or unlock a layer by picking a reference entity.

Select the function **Set Layer by Reference** from the Tools pull-down menu or call it by typing LAYREF.

If you have Option Menus or Toolbar set, you will be presented with the following command options:



You simply select the option that you want to perform and then select an object onscreen to indicate the layer that you wish the action to be taken upon. The last two options (On and Thaw) however, would have no visible entities onscreen, so instead, you are presented with the following dialog:



Here, you simply select the layer you want to change the status of (Layer 3 is shown selected in the window), and you simply select the action you want to perform on that layer. As you can see, you can set the layer as current and also switch display, access and status settings.

**Notes:**

1. This latter dialog can be called with any of the original options, simply by pressing enter or right-clicking with the mouse, instead of picking an entity onscreen.
2. The option SET is understood, if you do not choose an option and simply pick an entity after giving this command. The current layer will then become that of the entity selected.

## Lengthen



**Command: LENGTHEN**

**Alias LEN**

**Located:**

1. **Pull-down Menu: Modify - Lengthen**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Lengthen** provides the ability to extend the length of existing lines or arcs.

Unlike the command EXPAND, **Lengthen** extends an entity to a specific value or in a particular ratio (percentage value) of the original size.

Entities will be only lengthened in their existing direction. All entities will always be lengthened on the end that lies closest to the point where the cursor was positioned during the selection.

## Options:

### Dynamic

Lengthen normally works in the **Dynamic** mode. This mode is active after the selection of a drawing entity and needn't be selected from the Option Bar. After any lengthening operation the mode will return to Dynamic, no matter which of the options has been used.

In the Dynamic mode the line or the arc will be lengthened to a point or to an angle indicated the cursor. A preview will show, during this operation, the result of the command.

In addition to the standard mode Dynamic, the following modes are available for lengthening lines and arcs from the Option Bar:

1. **Total** allows the entity to be lengthened to a dimension that is defined. The dimension will apply to either the length of a line or to the circumference of an arc. To assist the user, the current length of the selected entity is displayed in parentheses. If the option **Angle** been selected the included angle of the arc is displayed instead of the length, and the input should also be the additional angle required.
2. **Percent** will permit the entity to be lengthened by a defined ratio of the original size. This percentage value relates to either line length or the circumference of an arc. For instance, the input of the percent value 200 doubles the length of the line or the arc, the value 50 diminishes it to the half of the original value. If the option **Angle** has been selected, the input of the percentage value will apply to the arcs original angle.
3. **Relative** lengthens a line or an arc by a length that is defined by indicating as distance between two points. This is an advantage particularly if the entity is to be lengthened by a straight line.

## Part Library Manager

### Command LIBRARY

#### Alias LB

##### Located:

1. **Pull-down Menu: Parts – Part / Symbol Library**
2. **Palette: Parts**
3. **Or Entered Directly at the Command Line**

The Part Library Manager easily manage and organize parts, blocks, and symbols, as well as project drawing files and your company's template drawings.

A drawing library is set up quickly with the Part Library Manager with visual control. This tool collects all or selected files that are located in a folder (or subdirectory). Descriptive information on the parts or drawings is stored within the drawing files.

You access the Part Library Manager with the Library command.

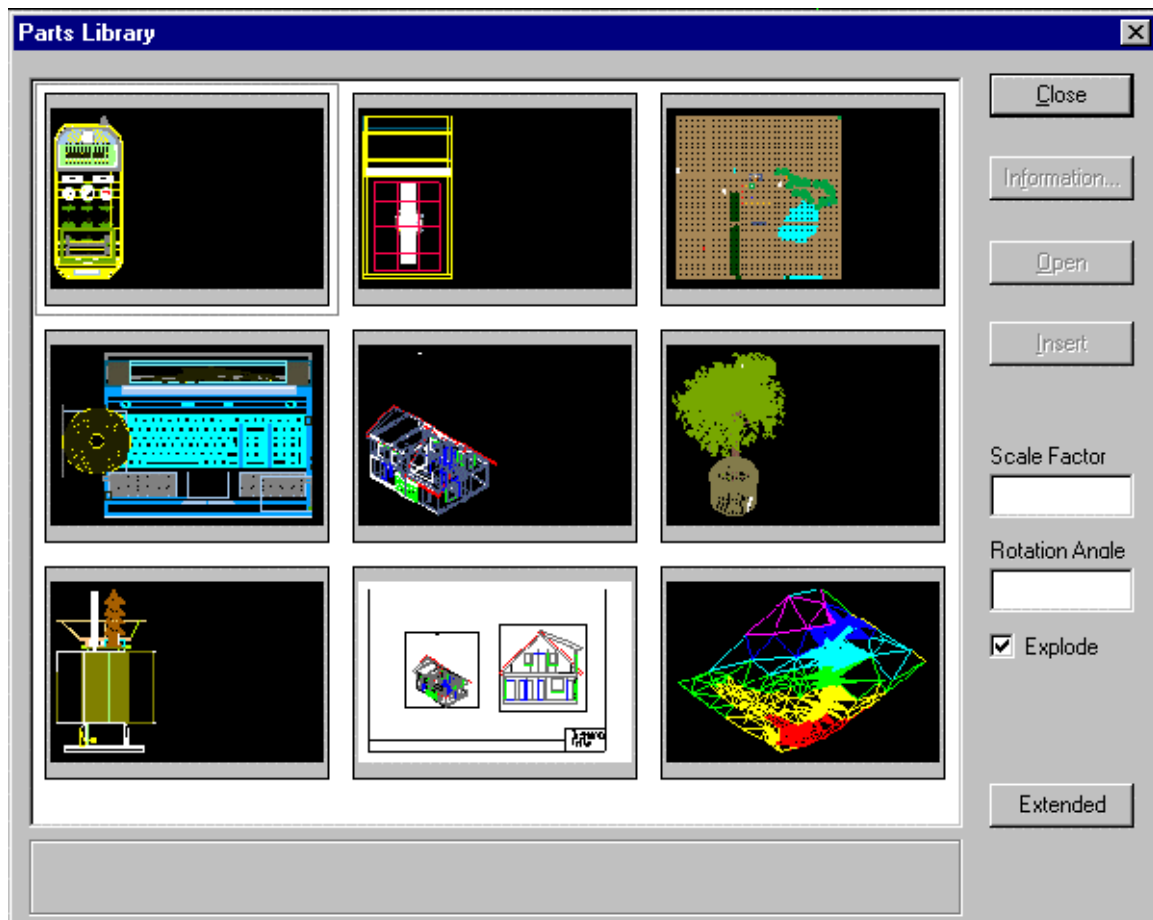


Figure i.7: The Parts Library Manager organizes parts, blocks, and symbols, as well as project and template drawings.

For more help with the Image Library click here: [IMAGELIB HELP](#) >>

The SETLIB command is used to set the default or current part library.

## Render Lights: Inserting Light Sources



**Command: LIGHT**

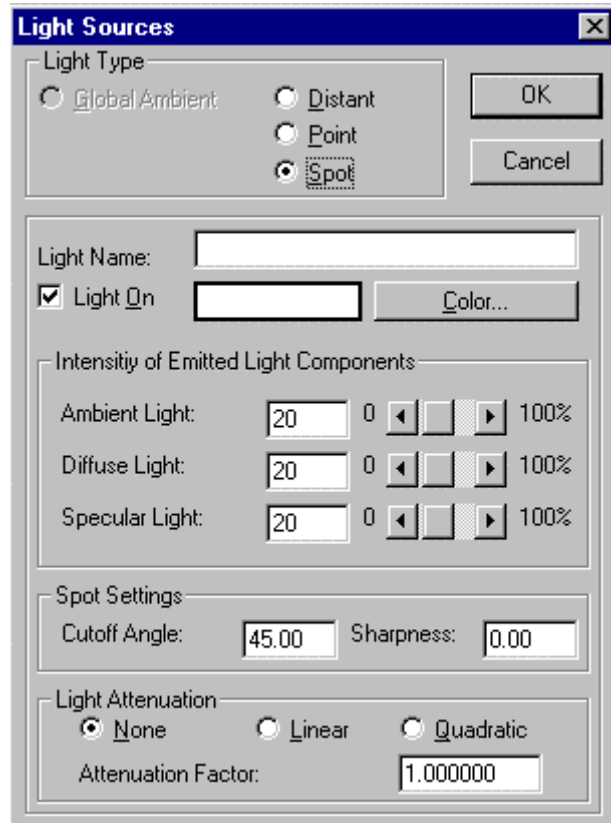
**Alias LIT**

### Located:

1. **Pull-down Menu: View – Render – Insert Light**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**



When you give the LIGHT command to create a new light source for your rendering, you will be presented with the following dialog:



For any new light source, the first step is to indicate its type. Selecting the Distant, Point or Spot button will signal that choice and also activate the light edit features on this dialog.

Next, we give the light a name, so that it can be called for later editing, as needed. You may need to switch the light on or off during the course of your work and there has been a selection button provided here for that purpose. By pressing on the Color button, you can select the color of light that you want. This color appears in the box to the left of the color button.

Below, you will find further advanced light editing controls. Unless you are an expert and familiar with all of these features, you would be best advised to leave these at default values.

**Notes:**

1. Ambient light is given any time, but linear, point and spotlight sources must explicitly be placed in the 3D model.
2. Note that although you can insert as many light sources you can have **only eight light sources turned on**.
3. Light sources are represented in the drawing as complex objects (part/block objects)
4. A light insertion will always be placed on a layer with the name RENDER\_LIGHTS.

---

**See also:**

3DVIEW

LIGHTEDIT

## Render Lights: Editing Light Sources



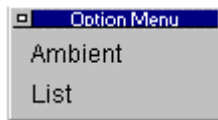
Command: **LIGHTEDIT**

Alias **LE**

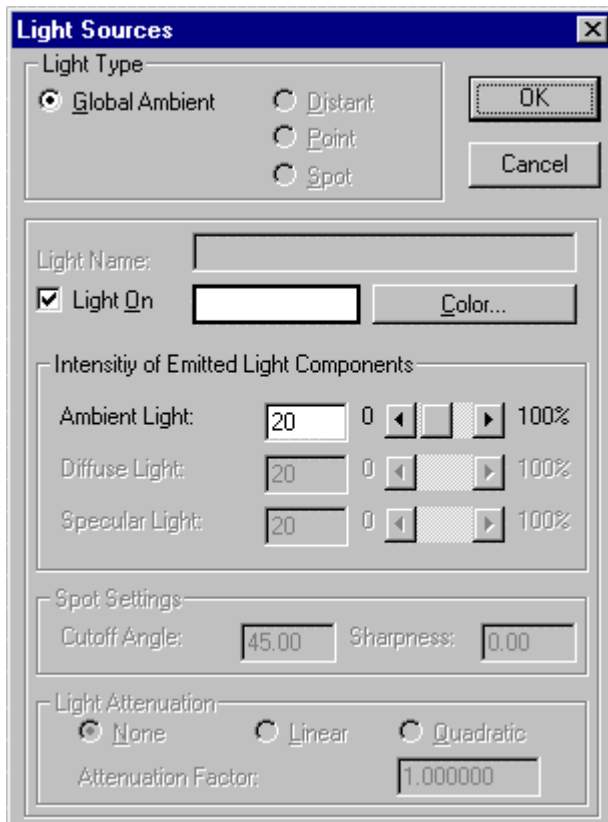
### Located:

1. Pull-down Menu: **View – Render – Edit Light Parameters**
2. Pull-down Menu: **View – Render – Ambient Light (LIGHTEDIT-A)**
3. Palette: **View**
4. Or Enter Command or Alias at the Command Line

When you issue the command, you are presented with the following dialog. Here, you specify whether you wish to edit ambient light settings, or else a previously defined (other) light source.



Below, we show the edit dialog, where we see that we can select/deselect global ambient light, where we can specify light color and intensity. In this example, we are simply editing ambient (natural) light characteristics. With other specifically set light sources, many different lighting effects become possible, which can be configured in this dialog.



---

**See also:**

3DVIEW

## Command LIMITS

**Command LIMITS**

**Alias LM**

**Located:**

**Entered Directly at the Command Line**

Although not strictly necessary, it can sometimes be helpful when plotting, to have the drawing area defined through the use of the LIMITS command. For greater convenience, it is wise to set your drawing area to a size that can be directly scaled to match the standard drawing sheet that you will be using. Afterwards, when plotting, you need only to specify plotting by limits to produce a plot of your full drawing, no matter what you presently are showing in your drawing edit window.

Command sequence is:

> LIMITS

Loading d:\AllenCad\SUPPORT\flx\_util.lsp... OK.

Reset the model space limits.

On/Off/<Lower-left corner><0.00,0.00>: [Accept default or enter new coordinates]

<Upper-right corner><12.00,9.00>: [Accept default or enter new coordinates]

Line



**Command: LINE**

**Alias L**

**Located:**

1. **Pull-down Menu: Draw - Line**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The **Line** command is used to draw lines or line segments or to connect lines to existing lines or arcs. Inputs to the command are two points; the starting point and the ending point of the line.

### Options:

#### Continuous

**Continuous** is the standard option when drawing lines, and will automatically be used when no other option is selected. It allows the user to draw line segments in sequence until the command is terminated.

The command will request a start-point for the line, and will continue to request points until it is terminated. Each subsequent point will be connected to the previous point by a line.

From Point:

To Point:

To Point:

...

**Undo**

The option **Undo** can be used repeatedly to erase the last drawn line segment of a single **Continuous** line command in succession.

### **Close**

The option **Close** connects the last drawn point P<sub>n</sub> with the start point P<sub>1</sub> in the **Continuous** line command.

To terminate the command, press RETURN or ESC.

### **Line Segments**

The option **Segments** allows the creation of single line segments, as opposed to continuous lines.

**Line Segments** are useful when the line that is being drawn for future drawing construction purposes (IE a temporary line that defines a parameter to be used in drawing subsequent objects).

As with continuous lines, you determine the start- and end-point of the first segment. The program will continue to ask for start- and end-points.

In this mode the end of one line segment is not automatically used as the start point of the next segment.

To end the drawing of line segments, press RETURN.

### **Append**

The option **Append** allows the connection of a line to either the start or end-point of another line or arc.

Choose the line or arc that you wish to connect to. Note that the end of that object closest to the pick point will be used as the start point for the line segment.

#### **Line to line**

When appending a line to a line, select the line to which the appended line will connect. The point (start or end) of that line closest to the selected point on the line will be the start point for the new line.

#### **Line to arc**

To append a line to an arc, select the arc to which the line is to be appended. The prompt will ask for the length of the line. Either keyboard the length, or "pick" the length with the cursor. For precision, it is usually better to keyboard the length.

The appended line will be connected to the end point of the arc as a tangent.

To end the command press RETURN or ESC.

Load Linetype(s)...

### **Command: LINETYPE**

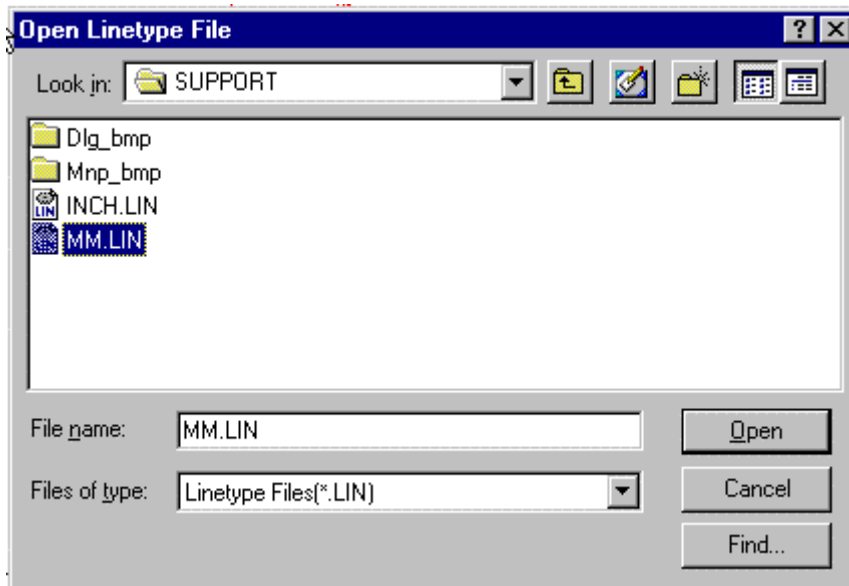
#### **Alias LT**

#### **Located:**

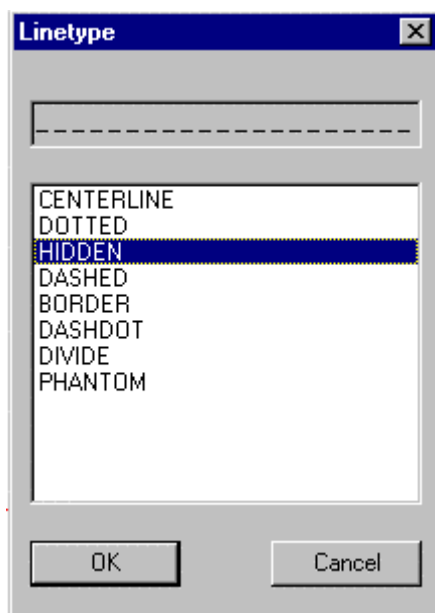
1. **Pull-down Menu: Tools – Load Linetypes**
2. **Or Enter Command or Alias Directly at the Command Line**

The LINETYPE command allows the user to load additional linetypes for use in the drawing. These linetypes will join those originally loaded with the drawing when it was opened, including those opened in previous drawing sessions.

Linetypes are stored in .lin files and these files are selected for linetype extraction when using the command, as in the commands first dialog (below):



Once you select and confirm the .LIN file to use, the following dialog opens, and displays the linetypes that are available in that file.



As you click on each one, a representation of the linetype appears in the window at the top of the dialog. This aids in linetype selection. Multiple linetypes can be loaded at once through the use of the Shift or Ctrl key while picking individual linetypes on the list.

Linetypes that are loaded, will be saved along with the drawing so as to be available at a later time.

Once loaded, linetypes can be assigned as default to layers (Command LAYER), can be assigned to existing linear entities (Command PROPEdit), or set as default linetype for subsequent dline drawing (Command SETLINETYPE).

Further information on linetype definition and file syntax can be found in Chapter 5 of the **Customization and Programmer's Guide**

**See Also: Command LOADLTYPE**

**Command LISTFILE**

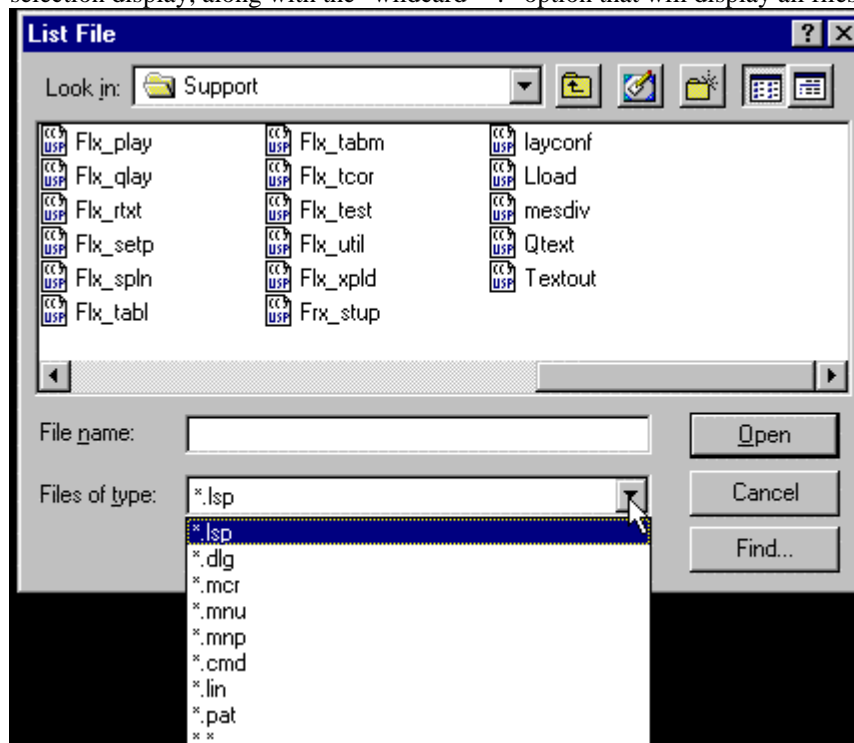
**Command: LISTFILE**

**Alias LF**

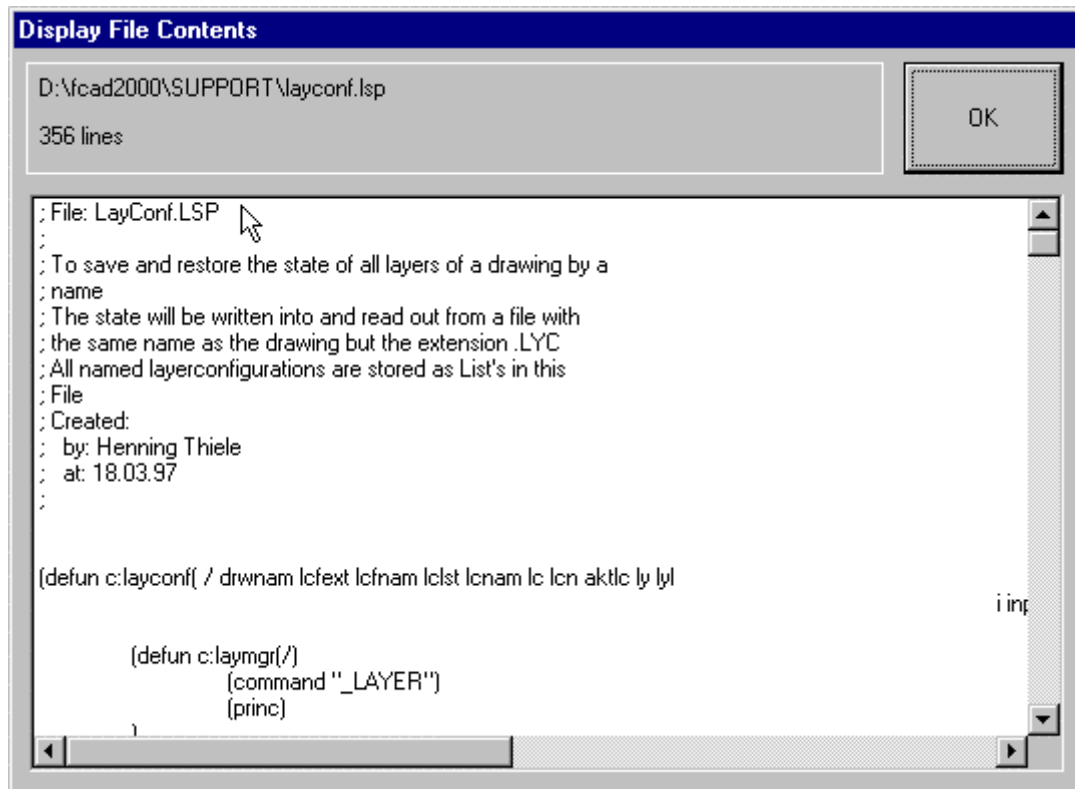
**Located:**

- 1. Pull-down Menu: Tools – Resource Manager – List File**
- 2. Or Enter Command or Alias at the Command Line**

The LISTFILE command, located on the Tools pull-down menu as "Resource Manager – List File..." is used to open and inspect file contents. The command can also be entered at the command line, if preferred. This can be useful to confirm contents before the file is used in the program, for instance. As the dialog below indicates, the most common file types that are used by AllenCad are available for selection display, along with the "wildcard" \*.\* option that will display all files.



Below, you see the window that opens showing the file contents after selection has been made. Click on OK to close the window when you are finished and wish to return to the drawing editor.



## Command LOADLISP

### Command: LOADLISP

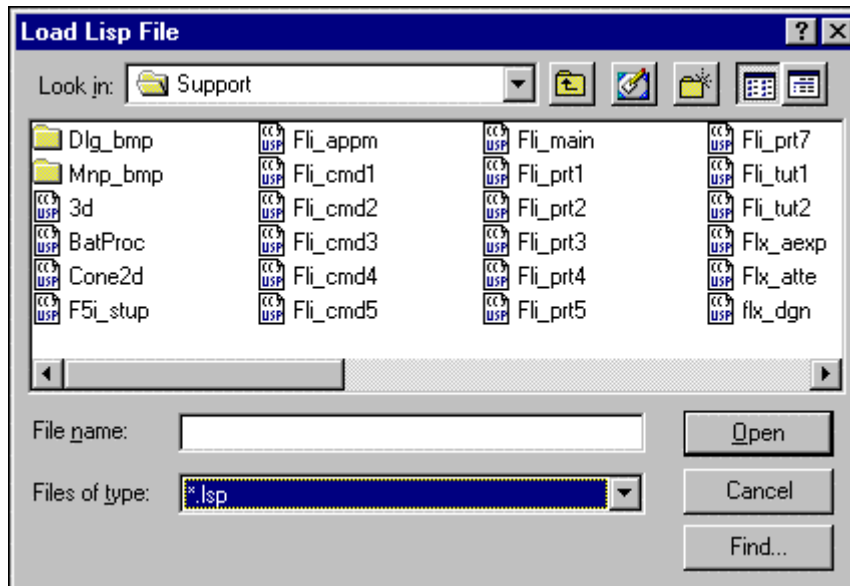
#### Alias LL

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Load Lisp File**
2. **Or Enter Command or Alias at the Command Line**

The **LOADLISP** command is used to locate and load LISP programs for use in the program. The command is issued from either the "Resource Manager - Load Lisp file" selection on the Tools pull-down menu, or else it can be entered at the command line.

The following dialog offers an opportunity to browse and select your file from the dialog display.



**Note:** Be sure to check the command line as the LISP file is loaded. There you will see the command name to enter whenever you want to run the routine during the current edit session. The example below indicates the command name to use when the "Textout.lsp" file is loaded.

```

Loading D:\FCAD2000\SUPPORT\TEXTOUT.LSP...Command TEXTOUT loaded.
Done:nil
> |

```

## Command LOADLTYPE

**Command:** LOADLTYPE

**Alias** LLT

**Located:**

### 1. Entered Directly at the Command Line

The LOADLTYPE command is a command line version of the LINETYPE command, achieving the same result, but without the use of dialogs. When the command is issued, you will be prompted to supply the name of the .LIN file that is located in the program's Support subdirectory. Finally, you will be asked to name the linetypes you want loaded.

## Command Command\_LOGFILEOFF

**Command:** LOGFILEOFF

**Alias** LOF

**Located:**

### 1. Entered Directly at the Command Line

Command logging is turned off using the Command\_LOGFILEOFF command. When this command is given, the log file will be saved with the default filename and path that were previously set using the LOGFILENAME system variable. The value for LOGFILEMODE is also changed to zero when this command is issued, thereby disabling file logging.



## Command LOGFILEON

**Command:** LOGFILEON

**Alias** LON

**Located:**

### 1. Entered Directly at the Command Line

Recording of issued commands and program responses in a log file is turned on using the LOGFILEON command. This command, along with a related command Command\_LOGFILEOFF control the value setting for the system variable LOGFILEMODE. The system variable LOGFILENAME should also be set with a recorded default log file name and path before logging is used.

## "M"

Macros

**Command:** MACRO

**Alias** MA

**Located:**

### 1. Entered Directly at the Command Line

Macros (sometimes referred to as script files) are designed to automate the execution of a series of specific instructions or program commands one step at a time. This eliminates the need to enter the commands or entries each time they are required in the program.

#### Creating Macros

Macros can be written using any text editor or word processing program that is able to store pure (ASCII) text format i.e. without the addition of any formatting information.

Enter the desired commands and instructions and their corresponding input values into this file in the same order in which the instructions are to be executed. A description of the individual commands and their necessary inputs can be found in the corresponding chapters of the *User's Guide*.

Command execution takes place line-by-line or instruction-by-instruction when separated by a blank space. Either an end of line or a space character is interpreted as confirmation of the previous entry.

This means that either of the following macros could be used to draw a circle with a mid point at 0,0 and a radius of 10.

**Sample**

#### Using the Clipboard

A simple and effective method of creating a macro file is made available by using the text window History Function and the Windows clipboard utility.

The text window contains the complete list of program commands and data entries that have been made during the course of a work sitting. This enables the user to reference and reconstruct every procedure carried out so far. By either using the scroll bar located on the right hand side of the text window, or alternatively pressing the upward / downward directional keys, the user can "scroll" through the listed entries. The Windows clipboard utility can then be used to copy entries into an editor program where they

can be stored as a macro file.

Use the following procedure:

1. Carry out the command steps that you wish to store.
2. Open the text window and use the mouse cursor to mark the command sequence.
3. Copy the highlighted section onto the clipboard using the CTRL + C key combination.
4. Call up your editor program, for example Windows Notepad.
5. Insert the contents of the clipboard into the editor using CTRL + V
6. Store the file in the support directory using any file name followed by the **.mcr** file extension identifier.

### Utilizing Macros (Command MACRO)

The macro utility can be called by entering the MACRO command. The program will then request the file name. It utilizes the pre-defined **.mcr** file if not specified otherwise.

Further information on the use of Macros can be found in Chapter 3 of the **Customization and Programmer's Guide**.

## Commands Measure and Divide

**Command: MEASURE, DIVIDE**

**Alias MR, DI**

### Located:

1. **Pull-down Menu: Draw – Measure / Divide**
2. **Or Enter Command or Alias Directly at the Command Line**

The commands MEASURE and DIVIDE order points or multiple blocks (parts) along a 2D-Object. This object can be an arc, a line, a circle or a 2D-Polyline. Before a block may be used with these commands, **it must exist in the drawing database before the command is given**. This may be accomplished, either through insertion or part definition. When the command is given, the block can be inserted aligned, or not aligned to the object. Both commands work in a similar fashion. While the command MEASURE asks for a distance, DIVIDE requests for the number of points with the distance between points or parts dependent upon the length of the linear entity selected.

### Command sequence for MEASURE and DIVIDE

After typing in "measure" or "divide" (or selecting it from the menu DRAW), the following request appears:

/A> measure

Select a line/circle/2d-polyline or an arc to measure:

After selecting the entity it is highlighted and the following question appears:

Block or distance:

Here you can input either a distance directly, (by putting in a real value or selecting two points), or you

can input "**Block**" (or "**B**". "**Bl**" etc), to use a block. In the first case the points at each distance from the start to the endpoint of the object are inserted and so the command is completed. If you select the option "**Block**" you will be asked for the block name:

Name of block to insert:

Now type in the name of an **existing** block. You will be asked for the offset angle for the alignment of the block along the object.

Offset angle for alignment the block along the object (or ENTER for no alignment):

Here you can press return to avoid the alignment, or to put in any offset angle that is added to the tangential angle of the object at the point of insertion. Lastly, you will be asked for the distance again.

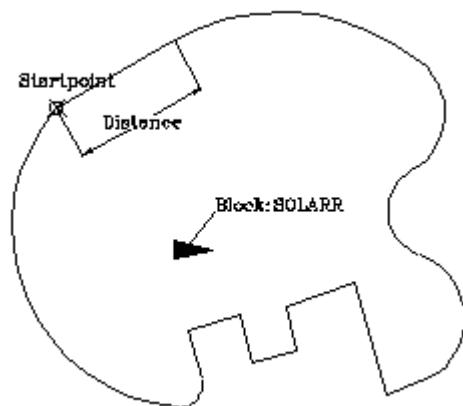
Distance for dividing the object:

You have to put in the distance here, either by giving a real value or by selecting two points. Now the Block at each position is inserted and the command will terminate.

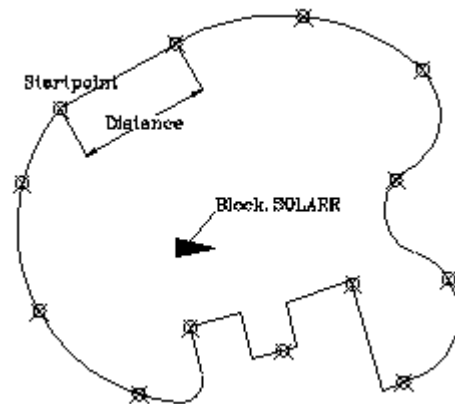
The sequence of the command DIVIDE is the same as above, except that you are asked for the number of points or blocks to insert, instead of a distance. In this case the command will compute the length between the insertion points itself.

### Example:

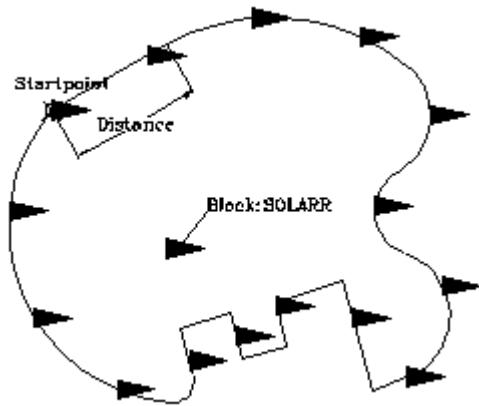
The following pictures show at the sample of a polyline, how the commands MEASURE and DIVIDE are working:



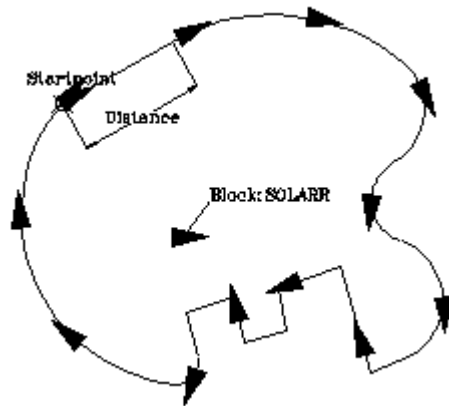
The initial state: The entity to measure is the polyline. Moreover the block SOLARR is already defined in the drawing.



After measuring the polyline using the distance, we receive points at each distance along the polyline, starting at the polyline start point



After dividing the polyline in ten pieces the block SOLARR is inserted ten times along the polyline, but it is not aligned.



In this case the polyline is divided into 12 pieces with the option block, and aligned with an angle of 0 degrees, this means the block is inserted locally tangential to the polyline.

#### Technical notes

The command Divide inserts number+1 entities along the object unless a circle is selected. So it may be necessary to delete the first or last inserted entity, if a closed polyline is selected. This is much easier, however, then creating an additional entity manually.

The command is not an internal command and so it available for the usage inside the (command...) function. To use it inside of a LISP-application use the related LISP-Function (MesDiv EntName Dist|Nr BldName OffsAngle) instead that is defined in MesDiv.LSP in the PCAD-Applic directory.

---

#### See also:

POINT

## Load Menu



### Command: 'MENU

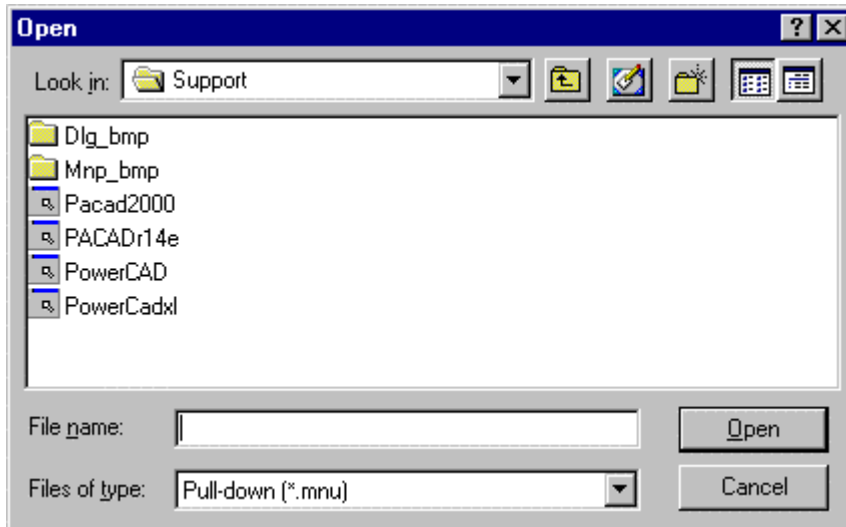
Alias MU

#### Located:

1. **Pull-down Menu: Tools – Resource Manager – Pull-down Menu**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The program is designed so that it is possible to work with different pull-down menus. AllenCad comes complete with both it's own AllenCad menu, as well as AutoCAD and FelixCAD emulation menus. Although the AllenCad menu offers the greatest functionality, users more familiar with other CAD programs may prefer to use the emulation menus provided.

The pull-down menus are stored in menu files with the extension **.mnu**. To switch to using another stored menu file, simply execute the MENU command and select the file from the listing.



## Command MERGE

**Command: MERGE**

**Alias MG**

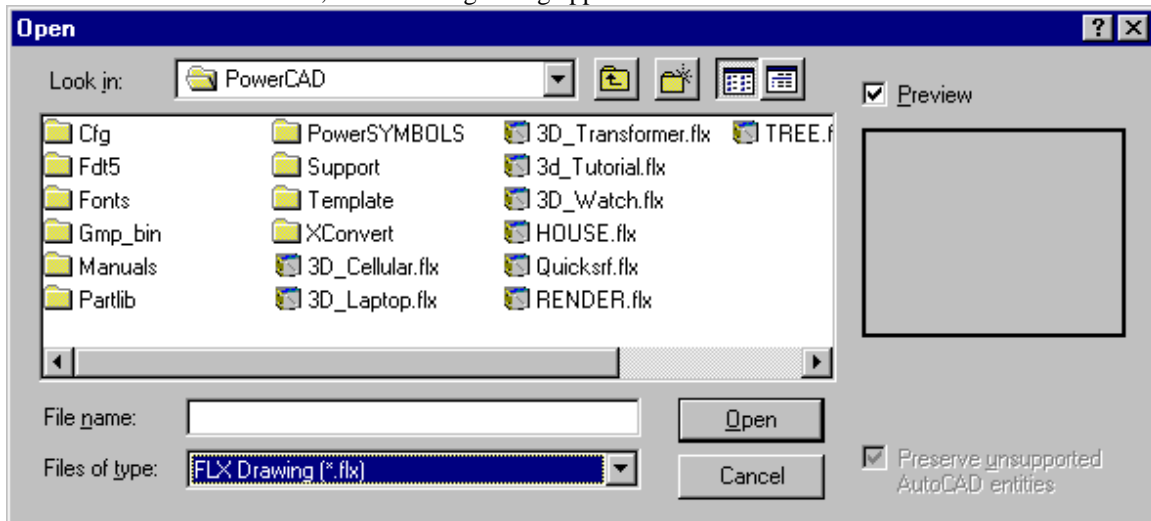
**Located:**

1. **Pull-down Menu: Parts – Merge External Part 1:1**
2. **Or Enter Command or Alias at the Command Line**

The MERGE command is used to import an external .flx drawing file, introducing it as a new Part in the drawing.

The command is located on the Parts pull-down menu as the "Merge External Part 1:1 " selection. It can alternately be entered at the command line, if you choose to.

When the command is issued, the following dialog appears:



With this, you can browse to the location and select the file you want. Note the "Find" button that can be used when you know the name but not the file location. Other buttons are also provided to help with

search mobility and file identification.

## Mirror



**Command: MIRROR**

**Alias Mi**

### Located:

1. **Pull-down Menu: Modify - Mirror**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Mirroring drawing objects produces identical copies of selected entities that are placed in a "mirrored" position about an axis specified by the user.

The selected objects have, after "mirroring", the same relative position to the symmetry axis as they had prior to " mirroring".

The mirror axis can be defined freely, or parallel to the X-axis or the Y-axis

In order to use the command the user must select an object to be mirrored. Select the object or objects, remembering that the selection of multiple objects will result in the treatment of the objects as one.

The next prompt will ask for the definition of the mirroring axis.

### Options:

#### Mirroring With a Free Axis

The Option **Axis** allows the user to freely define the mirror axis. It is the standard option for mirroring objects and is used when no other option has been selected.

Select this option, then define the position of the mirror axis. This can be made by indicating two points or by inputting their coordinates. The mirror axis can have any alignment with regard to the objects to be mirrored as well as with regard to the axes of the coordinate system.

If the mirror axis should run through points of an existing geometry (for instance, along a solid edge), the user should employ the object snap options for precise definition.

#### Mirroring on a Horizontal Axis

To choose a horizontal-mirroring axis, select the option **Horizontal**. This option creates a mirror axis that runs parallel to the Y-axis of the coordinate system. Its exact position is indicated by a single point.

#### Mirroring on a Vertical Axis

To choose a vertical-mirroring axis, select the option **Vertical**. This option creates a mirror axis that runs parallel to the X-axis of the coordinate system. Its exact position is indicated by a single point.

#### Note: Mirroring Text

If there is text associated with the selected object or objects, it is likely that the user does not want to mirror that text. In order to mirror the text it is necessary to activate an option called **Text Mirroring**. Activating this option will result in the text also being mirrored.

## Move



## **Command: MOVE**

**Alias M**

### **Located:**

1. **Pull-down Menu: Modify - Move**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The command is used to move objects within the coordinate system. An object can be moved in any combination then, of the x, y, or z directions.

The first prompt will be to select the object to be moved. Multiple objects may be selected, and they will be moved together in exactly the same relationship as they are currently placed. Select the object or objects, and confirm the selection by pressing RETURN.

### **Options**

#### **Basepoint**

It is the standard reference point for moving objects. It denotes the point that the user is choosing as the base from that to begin the command.

#### **Rel. Point**

The prompt will then ask for distances in the X, Y, and Z direction. Enter these values (if there is no Z value, enter only the X and Y values). The move will occur when the values are verified.

#### **Quick Move**

This method of accomplishing the move is to simply pick up the object(s) with the cursor, and "rubberband" them to the new desired location. This works well if no particular precision is required.

## **Creating and Viewing Slides**

### **Commands: MSLIDE**

#### **MSLIDE is Located:**

1. **Enter Command or Alias at the Command Line**

### **VSLIDE (Alias VS),**

#### **VSLIDE is Located:**

2. **Pull-down Menu: View – View Slide**
3. **Or Enter Command or Alias at the Command Line**

### **SLDOUT (Alias SO)**

#### **SLDOUT is Located:**

1. **Pull-down Menu: View – Take Snapshot (Slide)**
2. **Or Enter Command or Alias at the Command Line**

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.

You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

### To create a slide file

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.
3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

### To view a slide file

Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

## Command MTEXT



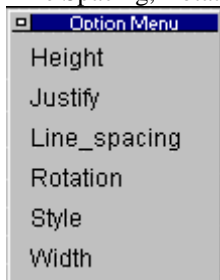
### Command MTEXT

#### Alias MT

#### Located:

1. **Pull-down Menu:** Detail – Multiline Text
2. **Palette:** Detail
3. **Or Enter Command or Alias at the Command Line**

The MTEXT command provides AllenCad with full multiline text functionality. After issuing the command you are asked to specify the location of the new text by picking an upper left and lower right corner on the drawing. This window roughly locates the placement for the text but is essentially used to determine the line length. Text will be wrapped into the next line if the available width is insufficient. While specifying the two window points, you are presented with options to change Height, Justification, Line Spacing, Rotation, Style and Width.

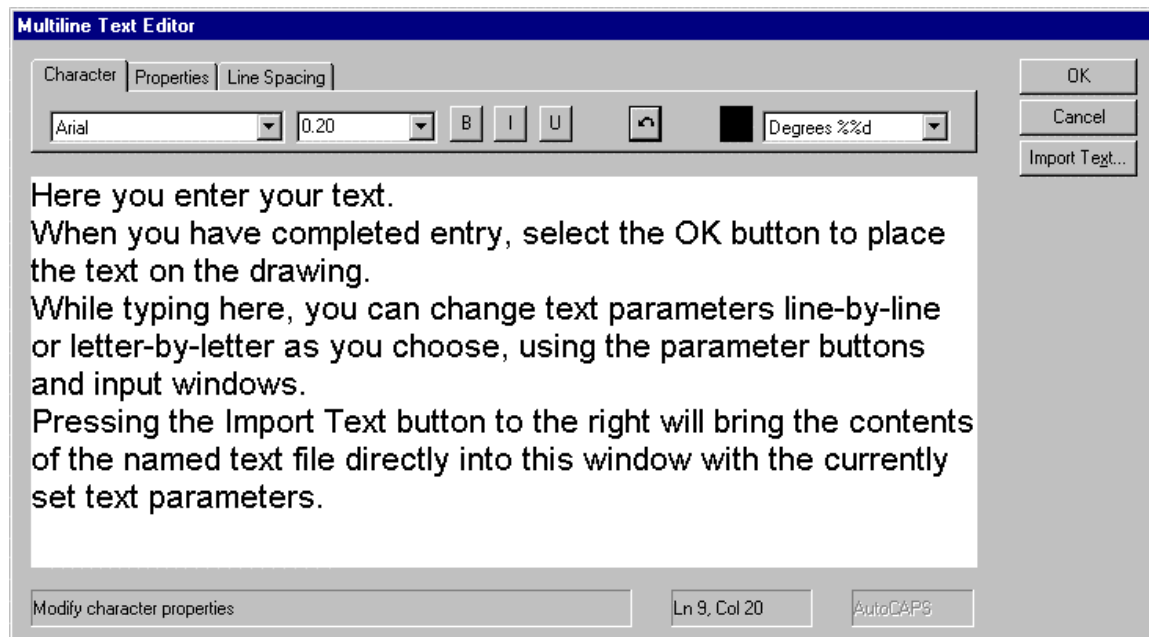


You may choose to set these values here, however these options are also available in the dialog that follows.



Next, you are presented with a windowed dialog box in which you enter your text. Three tabs allow you to exercise control over all text parameters and a **Text File Import** function enables you to import an external text file into the window. These duplicate and add to the earlier options that were presented and you can set your preferences in either manner.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's MTEXT command dialog window, either through picking on the window and using Windows' "Ctrl V" function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry



#### MTEXT Dialog showing Character Options.

**Note:** to allow the text to fill the indicated space in your drawing and to line wrap as needed within that designated width, be sure to continue the line entry beyond the visual extents of the entry window. Pressing the Enter button as we have done in the above example will allow you to see all the text without scrolling left-right, however it may not give the desired effect when this text is transferred to the drawing.

Most of these windows and buttons you are probably fairly familiar with if you do much word processing. Viewed left to right, these windows and buttons:

1. Specifies the required font
2. Toggles between bold and regular type
3. Italicizes text
4. Underlines Text
5. Removes the latest entry or reverses the latest action
6. Indicates the color to be applied to text. Click on this to change the setting
7. The last button allows you to call up special symbols (diameter, degree, plus/minus)



#### MTEXT Dialog showing Property Options.

Here, you can specify a named text Style, set Justification, override text Width originally set using the picked window points, and finally, Rotation, where you can specify an angle or bearing at which to place the entered text.



MTEXT Dialog showing Line Spacing options.

Here you can specify line spacing distance -either a minimum (At Least) or (Exactly)

See Also MTEXTEDIT , QMTEXT

## Command MTEXTEDIT

**Command: MTEXTEDIT**



Alias MTE

**Located:**

1. **Pull-down Menu: Detail – Edit Multiline Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

This command predictably is used for editing multi-line text. When the command is issued, you are prompted to select the multi-line text that requires editing. This command will only work on text that has been entered with the MTEXT command. You must use TCORRECT or TEXTEDIT on other text entities.

When you select an MTEXT entity, the full multi-line entity is brought into the MTEXT dialog window, where you can edit any of the parameters that were originally available when the text was first created.

# "N"

## Start a New Drawing



**Command: NEW**

Alias N

**Located:**

1. **Pull-down Menu: File – New**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

A new file has to be opened in order to create a new drawing. The command NEW in the menu *File* is used for this purpose. Alternatively, a new file can be created by clicking the system button NEW on the Function Bar. Invoking the command will cause the file dialogue box to be displayed.

The program will name the drawing `noname_n.flx`. The user may now name the drawing with the name of her or his choice.

To determine the storage location of the file select a drive, a directory, and, if required, a subdirectory in the middle part of the dialogue box. If the file name and the storage location are correct, confirm the settings. A new drawing (a new file) has now been created.

If a drawing file already exists with the same name the program will warn the user and require a confirmation so that the existing file can be replaced.

### Using Template Drawings

A template drawing may be called in order to start a new drawing. A template drawing is a model that can be used as basis for a final drawing or for the establishment of a drawing environment.

To start a new drawing using a template drawing, activate the check box *Load Template* in the dialog box *New file* and then double-click on the button *Template Drawing*.

The dialog box *Template Drawing* will be displayed. Select the drawing to be to used as a template drawing and confirm the choice.

---

#### See also:

OPEN  
SAVE  
SAVE AS..  
SAVE ALL  
CLOSE

## Polygon (N-gon)

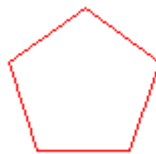


**Command: NGON**

**Alias NG**

#### Located:

1. **Pull-down Menu: Draw - Ngon**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**



The command **N-gon** is used to draw regular polygons. It is possible to draw polygons with up to 1024 sides, a value that should lie above any practical need.

When using this command, three options, based on different principles of construction, are available:

1. Definition of the length of sides of the polygon;
2. Definition of a center point and a radius of a circle outside ; or
3. Definition of a center point and a radius of a circle inside.

Select the command for drawing a polygon and choose one of the options from the options panel:

**Sides**  
**Outside**  
**Inside**

### Options:

#### Length of the sides

**Sides** is the standard option for drawing regular polygons. It will be used if no other option is chosen.

The regular polygon is defined by the length and placement of one side.

To draw a regular polygon, enter the number of its sides first. To do that, enter a value between 3 and 1024 and verify this entry by pressing RETURN.

If there have been polygons drawn previously, then a number will appear in the prompt. That number is the number of sides of the polygon drawn last. The user may elect to use that number by simply pressing RETURN.

After specifying the number of sides, determine the starting point and the end point of the first side either by entering the coordinates or by picking it with the cursor. Remember that the polygon will be drawn counterclockwise, starting from the first side.

```
> NGON
Number of sides <4>: 6
First corner point of the side: P1
Distance or second point of the side: P2
```

#### Circumscribing the Polygon

##### Outside

The option **Outside** defines a polygon by determining a center point and the distance to its corner points. This distance is the radius of a circle that contains the polygon, a point on whose circumference all corner points of the polygon lie.

After entering the number of sides, choose the option **Outside** from the Options Panel.

Determine the center point of the polygon by entering its coordinates or picking it. Next, determine the distance to its corner points. The polygon will be drawn counterclockwise, starting from the first corner point.

```
> NGON
Number of sides <4>: 6
Center point of the polygon: P1
Distance to its corners: P2
```

##### Inside

**Inside** allows the polygon to be defined by the circle whose circumference touches each side of the polygon at its mid-point.

Choose the option **Inside**, then enter the number of sides. Next determine the center point of the polygon by entering its coordinates or "picking" it; then determine the radius of the circle. The polygon will be drawn counterclockwise, starting from the thus defined side.

```
> NGON
Number of sides <4>: 6
Center point of the polygon: P1
Distance to its corner points: P2
```

**"O"**

Offset



## Command: OFFSET

Alias O

### Located:

1. **Pull-down Menu: Modify – Offset Through Point**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

**Offset** creates congruent pictures of lines, circles, arcs and 2D-polylines. This command enables a copy of a selected entity to be created and placed a distance from the original entity. The original entity(s) stays in position.

The operating mode of the command depends on the type of drawing entity:

- For lines and polylines identical copies are created and moved in the direction of the X or Y-axis.
- For circles and arcs copies are also created, but, in addition, they will be scaled by a selected factor. As a base point for scaling the center of the original circle or arc is used. In this way, concentric circles or arcs are created.

### Options

#### Offset for Lines/ 2D-Polylines

After entering the command, select the line(s) or polyline(s) that are to be offset by "picking" with the cursor. Then you either select the distance for the offset by picking, or select **Distance** from the Option Bar, and enter the offset distance. Then answer the prompt "Side to Offset?" with a point on the desired side.

#### Offset for Concentric Circles/ Arcs

Concentric circles or arcs can be created with a radius or diameter smaller or larger than the original entity.

After having selected the objects to be offset, the position of the offset is determined as follows:

- By indicating, or picking, a point;
- By input of coordinates of a point on the circle or arc; or
- By using the option **Distance** from the Option Bar (see below).

#### Distance (Command DOFFSET)

### Located:

1. **Pull-down Menu: Modify – Offset Distance**
2. **Or Enter Command or Alias Directly at the Command Line**

The option **Distance**, selected from the Option Bar, allows the offset to be created a specific distance from the original entity.

Next, determine the distance between the original entity and the offset. Do by either a numeric value input (in drawing units) or by cursor selection.

Then, point (and click) with the cursor to the side of the original entity on that the offset is to occur.

---

### See also:

DOFFSET

## Command OOPS

### Alias OOPS

#### Located:

1. **Enter Command or Command Alias Directly at the Command Line**

This command allows users to quickly undelete objects accidentally deleted in the previous operation. This function can also be accomplished using the UNDO command. OOPS is provided primarily for users who are familiar with other CAD software.

### Open an Existing Drawing...



**Command: OPEN**

#### Alias OP

#### Located:

1. **Pull-down Menu: File - Open**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

In order to continue working on an existing drawing in a later working session, the user must open the drawing. The term "Load" a drawing will also refer to "opening" a drawing.

The command **Open...** in the menu *File*, is used to do this. This command can also be activated by keyboarding OPEN or by clicking the standard symbol *Open* in the Function Bar.

After the entering the command, the user will see the selecting window, and may type in or select a drive, a directory, a subdirectory if it is required, and the File Name.

The file-type box offers the user the option to display AutoCAD .dwg, .dxf, AllenCad .flx drawing files, or all three.

#### Read-Only Option

The user may activate the option Read-Only in the dialog box used to open files. To do that, click on the control box beside Read-Only. A cross inside of that box means that this function is active.

The function Read-Only allows the file to be opened in order to be read, but it may not be modified. This is a protective mechanism against inadvertently (or, perhaps, by an unauthorized user, purposefully) modifying or changing the file. After opening in Read-Only, the file can be modified in any way, but the modifications will not be saved.

If the user should try to save a Read-only file, the program will send an error message and ask, at the same time, if the file should be saved under another name. If the changes are to be saved, then the user should answer <Yes> and type another file name in the dialog box that appears.

#### Find...

If the user does not know the exact name or the storing location of a drawing or file, she or he can look for it by using the option **Find**. The dialog box *Find File* will be opened. Select the drives to be checked. Enter the file name of the drawing to be found. The user may use the known wild cards <\*> (for any group of characters) and <?> (for any single character).

After the entries are confirmed the program will search for corresponding file names in the indicated

drives. All files matching the search criteria will be listed in a result window. The size, construction date and time as well as the location will also be indicated. The user should select the desired file from this list and confirm his or her choice.

---

**See also:**

NEW  
SAVE  
SAVE AS..  
SAVE ALL  
CLOSE  
Command OSNAP

**Command: OSNAP**

**Alias OS**

**Located:**

- 1. Left SideBar (individual icons)**
- 2. Entered Directly at the Command Line**

For precise drafting, many commands require that you select an exact position in the drawing. You can pick the precise location by typing the coordinates, or by picking a point in conjunction with an object snap. Object snap allows you to pick geometrically significant points on existing entities.

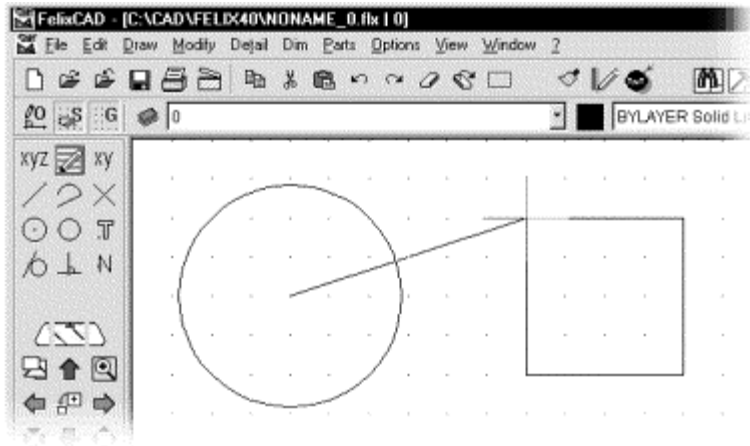
Object snap mode can be activated permanently or called for a single pick. The OSNAP command lets you turn on object snap modes; transparent commands - such as mid and qua -- temporarily activate an object snap mode.

The procedure for using the object snaps is always the same:

1. Start a command that requires a point, such as "From point:"
2. When the point is requested, reply with an object snap mode; the command prompts for the point with "of".
3. Select the object with the cursor.

Here is an example of using object snap. A line is drawn from the center (cen) of the circle to the corner (int) of the rectangle:

```
> line
From point: cen
of [pick]
To point: int
of [pick]
To point: [Enter]
```



### Drawing a line using object snap modes

Some object snap modes make no sense. For example, you cannot define the end point of a circle (use the qua mode instead), nor can you pick the center of a line segment (use the mid mode instead).

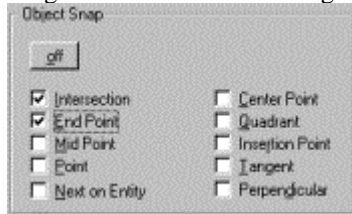
On the other hand, you can use several osnap modes on entities. For example, a line can use the end mode (twice, one for each end), mid mode, and per (perpendicular) mode. An arc can use end mode (again, twice), mid, cen, per, qua (quadrant points) and tan (tangent) modes.

The end object snap mode does not differentiate between starting or ending point of an entity; you must be careful with your pick point. The object snap selects the nearest of the two possible points.

The illustration below shows the connection between the point selected and the point snapped:

### Running Object Snap

Using the Precision Aids dialog box, object snap modes are turned on. This is called running mode.



Click the checkboxes for the desired mode(s). A check mark indicates the mode is active. Clicking the off button deactivates all running snap modes.

When several object snap modes are on, you differentiate between them by the way you select an entity. For example, End Point and Intersection are on; pick a point nearest an end point, then the end is selected. If you pick a point near the intersection of two entities, then the intersection is selected. Properly configured, this feature can speed the drawing of entities significantly.

### Clearing Object Snaps

Object snaps can all be switched off by either pressing the "OFF" button on the Osnap dialog, or clicking on the "No" icon on the Snap Palette. Individual Osnaps can be deselected on the Osnap dialog.

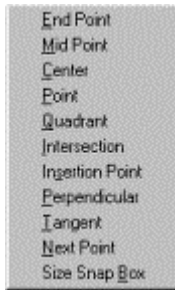
### Temporary Object Snap

You can choose between several options for a one-time activation of an object snap mode.

You can use the Snap Palette to activate an object snap mode just one time during a command.

An effective method of calling an object snap mode is to bring up the cursor menu. Press the middle mouse button (or hold down the Ctrl key and press the left mouse button on a two-button mouse). The following menu is displayed, from which you select an object snap mode:





Use the keyboard to type a command abbreviation. The object snap modes and their abbreviations are shown in this table:

Object Snap Mode	Abbreviation
End point	end
Mid point	mid
Center point	<u>cen</u>
Quadrant	qua
Intersection	<u>int</u>
Implied Intersection	Imp
Perpendicular to	Per
Tangent to	Tan
Next point on entity	<u>Nxt</u>
Insertion point	Ins
Point entity	<u>Pnt</u>

### Object Snap Modes in Detail

There is a difference between object selection and object snap modes. Object selection treats a group of entities, such as a rectangle, chain, and polyline, as a single object. In contrast, object snap modes treat these as individual entities. For example, you can snap to each segment and vertex of a polyline. This is also true for polygons and 2D and 3D faces.; the edges are interpreted as by object snap as individual lines. In this way, the object snap modes can be used for the edges of planes as well as for segments of chains and polylines.

For simplicity, when this text refers, for example, to the "end point of a line" the term also refers to polyline segments and the edge of a plane.

#### ENDpoint

The End object snap snaps to the endpoint of a line or of an arc. The endpoints of planes and faces can be accessed.

#### MIDpoint

The Mid mode snaps to the center of a line or an arc. The edges of faces may also be snapped.

#### CENterpoint

The Cen mode snaps to the center point of a circle, arc, and polyarc.

#### QUAdrant

The Qua mode snaps to the nearest quadrant of a circle or an arc. Quadrant refers to the points situated on the circumference at 0, 90, 180 and 270 degrees. Only the next visible quadrant of an arc can be snapped.

### **INTersection**



The Int mode snaps to the intersections of a pair of lines, arcs, and other entities -- or any other combination of them. This includes dashed and centerlines where the intersection point may not have line segments actually touching.

### **IMPlied**

The Int object snap mode only returns an intersection point of entities that have a real intersection. In contrast, the Imp mode snaps -- in addition -- to an implied (intersection point. This is the point where the extension of two entities would intersect.

### **PERpendicular**



The Per mode snaps to the perpendicular to an entity. This mode requires that a starting point already been selected. For example, when a perpendicular is desired, the ending point for the perpendicular is picked. The result is a perpendicular line from the ending point to the chosen entity.

### **TANgent**



The Tan mode snaps to the tangent point of a curve. This mode requires that a starting point has already been selected. For example, when a tangent to a circle is desired, the ending point for the tangent line is picked. The result is a tangent line from the ending point to the chosen entity.

### **NeXT**



The Nxt mode snaps to the next situated point of an object when at least one point of this object lies within the borders of the selecting cursor box. This ensures that entities that are designed or modified by use of the object snap mode share a point with the selected object.

(If you select a point in 3D space without using this object snap mode, it is possible that the selected point is not on the desired object. The two objects would not have a common point, or a closure.)

### **INSertion**



The Ins mode snaps to the insertion point of a part, an attribute, or a text object.

### **PoiNT**



The Pnt mode snaps to a point entity, which was created by the Point comm

## **"P"**

### **Close Palettes**

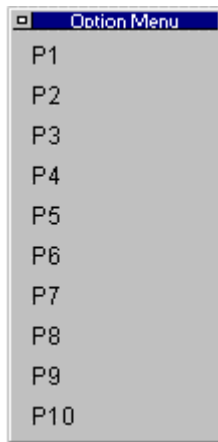
**Command:** 'PALCLOSE

**Alias** PLC

#### **Located:**

##### **1. Entered Directly at the Command Line**

To remove a palette from the desktop, use the PALCLOSE command or click on the close button in the palette header. You will be presented with the dialog:



Each open palette has its palette number designation in the header next to the palette name. Simply click on the corresponding number in the dialog to close the palette.

**Note:** Palettes can also be closed by clicking on the close button, located on the left side of the palette header.

---

**See also:**

Exchange Palettes

Close Palettes

Load Palettes...



**Command: 'PALETTE'**

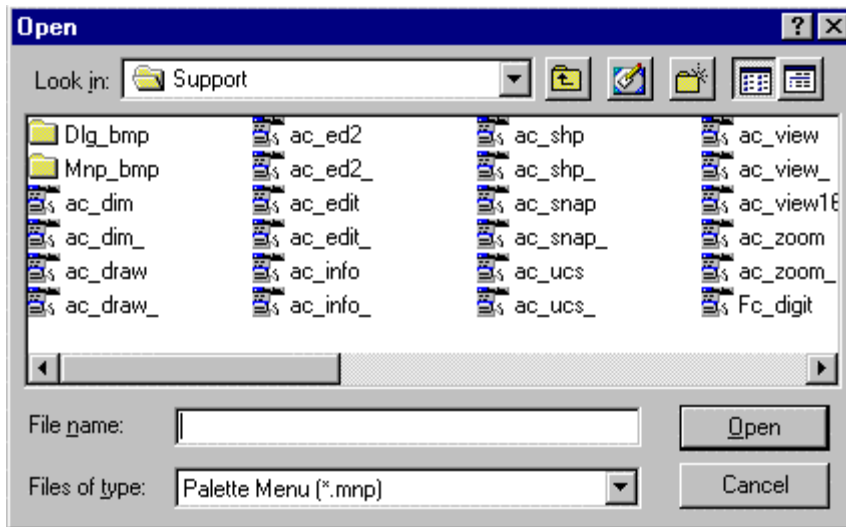
**Alias PLT**

**Located:**

- 1. Pull-down Menu: Tools – Resource Manager – Palette...**
- 2. Or Enter Command or Alias at the Command Line**

*Load Palette...* serves to open and place a Palette onto the program desktop. From the Open File dialog box that is opened the user can select the desired palette file to be loaded.

When you issue the command, a dialog box is presented, allowing you to select the palette file (\*.mnp) that corresponds to the palette you want to load. Simply select the file, and the palette will be placed on the desktop.



Every palette opens in its own window. In the title bar of the palette (toolbox) the user will find the name of the palette as well as an ID-number.

Palettes may also be opened through individual selections located on the pull-down menus, or using the palette icons located on the toolbar, as pictured in the image below:



Further information on creating and editing palettes can be found in Chapter 7 of the **Customization and Programmer's Guide**

---

#### See also:

Exchange Palettes

Close Palettes

### Palette Manager

**Command** PALMAN

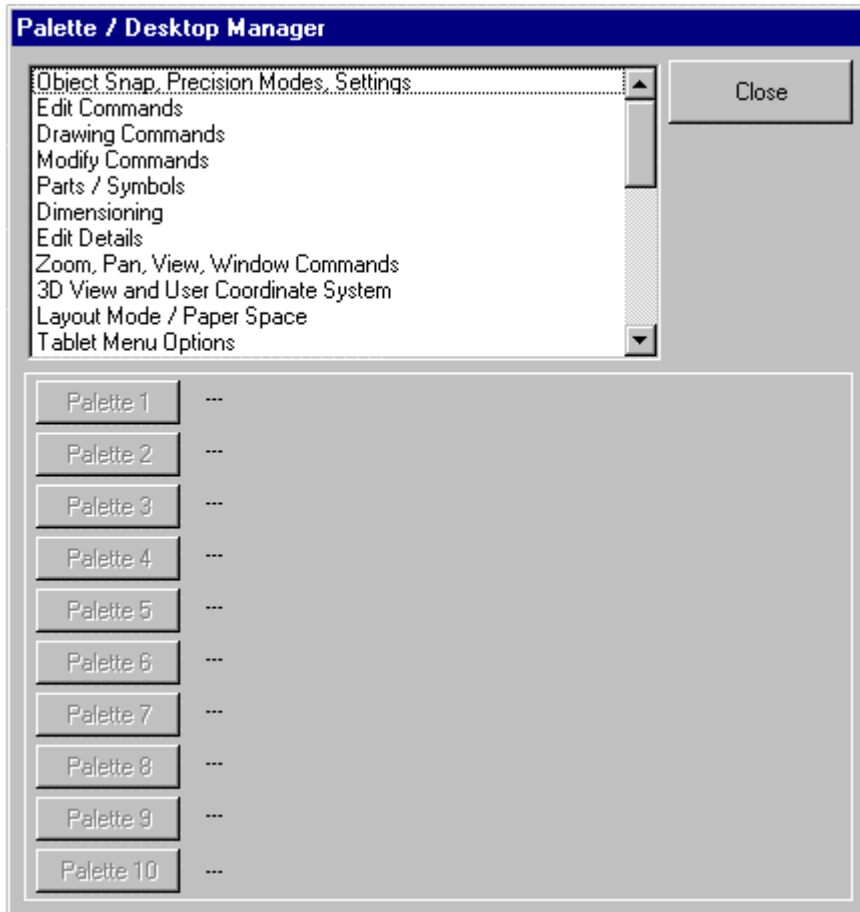
**Alias** PM

**Located:**

1. **Pull-down Menu:** Tools – Palette Manager
2. **Or Entered Directly at the Command Line**

Most users will elect to use the palette icons on the Toolbar to activate the various palettes, however others may prefer to select *Palette Manager* from the menu *File* and choose the action that they want in a more interactive fashion.

In the selection area of the Dialog Box, all of the palettes that are currently available are shown:



Select the desired palette. Underneath the selection area are three switches with the designations "Palette 1", "Palette 2", and so forth, to "Palette 10". By selecting one of the ten switches, the user can assign an identification number (for example, Palette 1) to the palette. If there is another palette with the same ID-number already opened, this palette will be closed and replaced by the new palette having that ID number. When the palette is opened, the selecting window will close. If no palette is to be opened or replaced the user may close the window by selecting *Cancel*.

If the user wants to include customized palettes in the Palette Manager she/ he should proceed as follows: Edit the file <Menu name>.pal that is situated in the \SUPPORT directory. For instance, if the user menu is called *Pcad.mnu*, the user must edit the file *Pcad.pal*. The structure of the file is:

Palette description 1

palette1.mnp

Palette description 2

palette2.mnp

and so forth.

The palette description will be found in the list shown in the Palette Manager Dialog Box, and the line following will always name the palette file. The user can complete or delete the entries as he or she wishes.

## Exchange Palettes

**Command:** 'PALOPEN

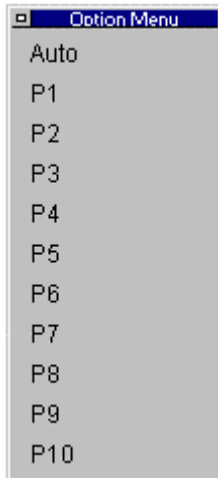
**Alias** PLO

**Located:**

1. **Pull-down Menus:** (Different pull-downs for different palettes)

2. **Toolbar icons for individual palettes**
3. **Or Enter Command or Alias at the Command Line**

The PALOPEN command will open a new Palette. If necessary, it replaces another palette.



Command Sequence:

> PALOPEN

Caption or ID of palette to replace: P1 (select from above dialog)

Upper left corner: Right click on the corner

File name: ac\_ucs (enter the filename of the palette to load).

The new palette is then placed on the desktop. Any existing palette loaded under the P1 heading, would have been replaced in this operation.

---

#### See also:

Load Palettes

Close Palettes

Pan



**Command: 'PAN**

**Alias P**

#### Located:

1. **Pull-down Menu: View – Pan**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias Directly at the Command Line**

The command PAN moves the visible portion of the drawing to the right, to the left, upward or downwards. It is now easy to look at objects that were previously not visible because they were off the viewing surface. In contrast to the ZOOM command, the drawing scale does not change.

The PAN commands are transparent ones that may be executed while other commands are being executed. It is possible, for instance, to execute the command PANDOWN (move downward) while entering points for drawing an element.

The user should remember that, when moving the drawing, it appears that the visible objects are moving in the direction of the command. The reason for this is that the PAN commands acts as does a photographer, to move the "camera", not the picture

'PANUP	Portion up
'PANDOWN	Portion down
'PANLEFT	Portion to the left
'PANRIGHT	Portion to the right

After selecting one of these commands, the visible drawing portion will immediately be moved in the opposite direction, allowing you to see more of the drawing in the direction you have indicated with PAN. The amount of movement is determined by a factor that is saved on the SETTINGS – View Parameters tab

---

**See also:**

DPAN command

ZOOM command

**Pan Down**



**Command: PANDOWN**

**Alias PD**

**Located:**

- 1. Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
- 2. Left SideBar**
- 3. Palette: Zoom**
- 4. Left Sidebar**
- 5. Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved down.

The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

**See also:**

PAN command

DZOOM command

Pan to the left



**Command: PANLEFT**

**Alias PL**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Left**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved to the left. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

**See also:**

PAN command

DZOOM command

**Pan to the Right**



**Command: PANRIGHT**

**Alias PR**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately be moved to the right. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR)..

---

**See also:**

PAN command

DZOOM command

**Pan Up**



**Command: PANUP**

**Alias PU**



**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Up**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

After selecting this command, the visible drawing portion will immediately shift upwards on the drawing. The amount of movement is determined by a factor that is saved as a system variable that can be entered through the configuration for the screen (Command VIEWPAR).

---

**See also:**

PAN command

DZOOM command

Define Parts...



**Command: PARTDEF**

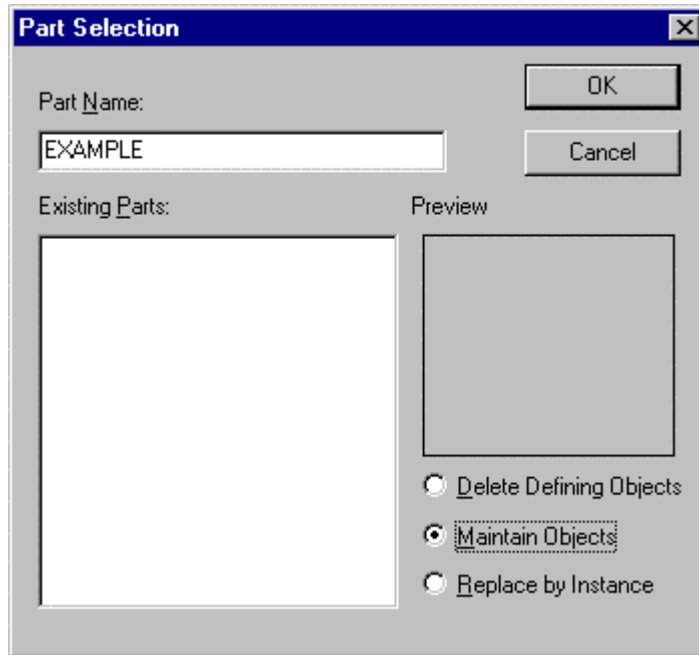
**Alias PDF**

**Located:**

1. **Pull-down Menu: Parts – Define Part**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

The PARTDEF command defines a new internal part. Select the function **Define Part...** from the menu *Part*, or enter the command PARTDEF via the keyboard. Once defined, the internal part definition will allow for multiple part insertions *within the current drawing*.

When issuing the command, you are presented with the following dialog:



### Part Name

Names of parts can contain up to 31 characters. In addition to letters and numbers the special characters \$, -, and \_ are allowed. The box is not case-sensitive.

Enter the name of the part in the input field of the Dialog Box. Existing Parts are listed in the selection window below. If one of those parts is required, select the part from the Selection Window using the cursor.

If there are parts of the same name already in the selection box, a message will appear so stating. The message will give the user the option of choosing a new name, or of re-defining the existing part.

The user may also, in this Dialog Box, elect to delete the part in the original drawing. The appropriate box should be checked to exercise this function.

After this choice has been made, verify the selections of the Dialog Box by OK. The Box will close, and the part will have been created.

### Existing Parts

If there are any existing part definitions in the drawing, they will appear in the window on the left side of the dialog. They can be selected here for inclusion in the new part definition, if required. The **Preview Window** to the right will show a thumbnail picture of the part selected in the window on the left.

### Other Dialog Options

The remaining three options determine what action the program will take with the objects being included in the part definition.

- **Delete Defining Objects** – deletes the objects after including them in the part definition
- **Maintain Objects** – Leaves them in the drawing as individual objects as well as including them in the definition.
- **Replace by Instance** – Substitutes the new Part in place of the individual defining objects.

This completes dialog input, where pressing OK moves you on to the final command line entries that complete the PARTDEF command.

You are next asked to define: **Insertion point**

Each part must have a defined insertion point.

The **insertion point** is a reference point that enables the user to position the part properly during insertion. The insertion point of a part is also the base point for changing the scale, or for use as a rotation point during the insertion.

Any point on - or even off of - the part may be selected as the insertion point. Typically, users will select the left lower edge, the center of an object, or some other discreet point of the geometry as the insertion point. It is a good idea to use the Object Snap options for this selection, as it insures precision.

### Object selection

Next, the user must select the object or objects that will make up this part. These will join any existing part definitions selected in the dialog, in the new definition. To do this, the Option Bar or Dialog furnishes the standard object selection options. After selection is complete, confirm by pressing Enter. At this point, definition is made and the command terminates. The part is now ready for insertion as required.

## Write Part File ...



**Command: PARTEXP**

### Alias PX

#### Located:

1. **Pull-down Menu: Parts – Write Part File**
2. **Palette: Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

**Write Part File...** works identically to the function **Part Definition...** (Command PARTDEF) but is used to export parts as external drawing files.

In practice, it is used to create new drawing files (.flx, .dwg or .dxf) consisting of only selected drawing entities, regardless of their intended use. It is similar to the SAVEAS command, except that Saveas will include all current drawing elements in the new file.

Both PARTDEF and PARTEXP functions create part definitions within the current drawing. The proposed part name, the part insertion point, an internal part definition and/or selected drawing entities are input during the export process.

Part / Symbol Library...



**Command: PARTLIB**

### Alias LIB

#### Located:

1. **Enter Command or Alias Directly at the Command Line**

The command PARTLIB in the menu *Parts* allows the user to manage drawing files such as project drawings, template drawings, part files, or symbol files.

A library is defined by a parameter file (of file type **.plb**) and a set of drawing files, assembled in a directory where the library parameter file is located.

The command PARTLIB called with the name of a parameter file (for example, c:\AllenCad\drawing.plb) will display a dialog that allows you to select a drawing to be inserted. The advantage of the command is that it displays all drawings with a preview bitmap, with verbal information in a list box.

Within the dialog box the user will see the preview of the drawings and information about the drawing.

There are two modes for the list box:

1. **Info Mode** the drawings are displayed together with the verbal information.
2. In **Quick Mode** the drawings are displayed only with their preview bitmap.

The other buttons of the dialog have the following functions:

- **Close:** Closes a dialog without further action
- **File Info:** Opens a dialog to display and to change the file information.
- **Open:** This opens the selected file as separate drawing.
- **Insert:** This button lets the user insert the drawing into your current drawing.

---

**See also:**

Setup and Manage Part Libraries

This section describes how to create PLB files to customize the PARTLIB command to personal needs.

### Paste Selection from Clipboard



#### Command: PASTE

Alias PST

#### Located:

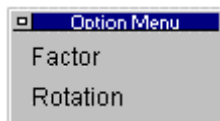
1. **Pull-down Menu: Edit – Paste Selection From Clipboard**
2. **Top Function Bar**
3. **Palette: Edit and Modify**
4. **Or Enter Command or Alias at the Command Line**

The PASTE command copies objects from the **AllenCad-specific** clipboard to point you select in the current drawing. These objects have typically placed on the clipboard by a preceding COPYSELECT or CUTSELECT command.

Use the command to copy graphic data from other drawings into your current drawing.

#### To paste the objects from the clipboard into the current drawing:

1. Choose the PASTE command.



2. You are presented with the option dialog. If you plan to scale or rotate your objects during insertion, select your option now. You will be offered the opportunity to scale and rotate as a conclusion to the command later if you do not choose at this point, however you will not be able to visualize the scaled or rotated objects in advance, if you wait until later.
3. Specify the insertion point by entering coordinates of the insertion point, OR
4. Drag the highlighted objects into position on the drawing. The objects of the clipboard contents appear highlighted (as well as scaled and/or rotated - if you selected this in the last step), and attached to the cursor during the exercise. This feature allows you to visualize object placement prior to selecting the insertion point with a mouse click.
5. Specify the scale factor for insertion – you can specify a scale factor at this point (if you have not

done so already) and have the objects inserted at a larger or smaller scale than they were originally. Note that the relative position of these objects to each other remains the same (is scaled too) during the operation).

6. Specify the rotation angle (if you have not done so already) – you can choose to rotate the object by any input angle during the exercise.

**Example:**

```
> PASTE
Insertion point:
Scale factor <1>:
Rotation angle <0>:
```

When inserting the objects from the clipboard you can also specify the scale factor and/or rotation angle before you indicate the insertion point by choosing the appropriate option from the context bar when prompted for the insertion point:

```
> PASTE
Insertion point: Factor
Scale factor <1>: .5
Insertion point: Rotation
Rotation angle <0>: 90
Insertion point:
```

**Note:**

The objects selected with COPYSELECT remain on the clipboard, ready to be placed in another drawing, until you place new objects on it with the COPYSELECT command.

To copy objects within the current drawing it is recommended that you use the COPY or ARRAY command found in the *Modify* menu.

---

**See also**

Command COPYSELECT

Command CUTSELECT

**Polyline Contour**



**Command: PCONTOUR**

**Alias PC**

**Located:**

1. **Left Side Bar**
2. **Entered Directly at the Command Line**

This command allows creating a closed polyline on top of (overlying) objects that make up a closed boundary. Such a polyline contour can serve for hatching regions of the drawing with the HATCH command or to determine the area or circumference of a region with the AREA command (applying the option *Object*). The BOUNDARY command is identical in operation to this command and is provided for users who may be more familiar with the term.

From the menu *Draw*, choose **Polyline Contour**. At the command prompt, enter PCONTOUR.

It is recommended that you set a specific layer before invoking the command (like a hatch or contour layer), because the polyline contour will overlap entirely or partially existing drawing entities, making selection otherwise difficult.

After giving the command, you are prompted to select objects to be examined to find the contour.

**Important Note:** At this point, it is not required to select only those entities that should make up the contour. You might enter the option *All* as object selection mode, however, the supplying a smaller selection set can improve the command efficiency, as fewer objects are evaluated in boundary establishment.

After the selection set to be examined has been created, the following **Ray Casting** methods for contour finding are available to find the next boundary object: Normally, the default works well, however there may be instances where you may wish to select a specific option:

1. **Nearest** sends radial rays from the pick point you specify (see below) to the nearest object.
2. **+X** sends a linear ray in the positive X direction from the pick point you specify to find an entity.
3. **-X** sends a linear ray in the negative X direction from the pick point you specify to find an entity.
4. **+Y** sends a linear ray in the positive Y direction from the pick point you specify to find an entity.
5. **-Y** sends a linear ray in the negative Y direction from the pick point you specify to find an entity.

Next you are prompted to pick the start point from which the program begins assessing the intended boundary..

Once a boundary object has been found in the direction of the specified pick point, the command traces the boundary in a counterclockwise direction.

After analyzing the specified selection set successfully, a closed polyline will be produced on the current layer, covering the boundaries of overlying drawing entities.

## Command PCURVE

### Command PCURVE

#### Located:

1. **Pull-down Menu: Modify – Edit Polyline – 2D polyline to Curved Polyline and Decurve Curved Polyline**
2. **Or Enter Command or Alias at the Command Line**

The PCURVE command is used to transform the straight-line segments and vertices of 2D polylines into arc-like shapes. This gives the appearance of transforming the polyline from being made up of straight lines to being made up entirely of spiral segments (Catmull-Rom splines). The program does this by breaking the line segments into shorter length segments and re-orienting them along the path of these spirals.

A polyline that has been transformed in this way can at time be changed back into its original straight-line format by re-issuing the PCURVE command and selecting the Decurve option that appears in an option dialog whenever the command is issued.

User input determines the length of the segments in the spline and therefore the smoothness of the resulting curves. In the following examples, the original polyline is shown in blue and the splined polyline resulting from the noted inputs is shown in red.

Inputs for this command are inexact and are essentially a relative number within a range of possible values (please note that zero is an invalid input). If you are experimenting, remember that the UNDO command can be used to easily back up from a PCURVE command that does not produce the required effect. The varying effects of different inputs is demonstrated below:

1. First, we will input a high number of interpolation points and a high tension setting:

```
> PCURVE
```

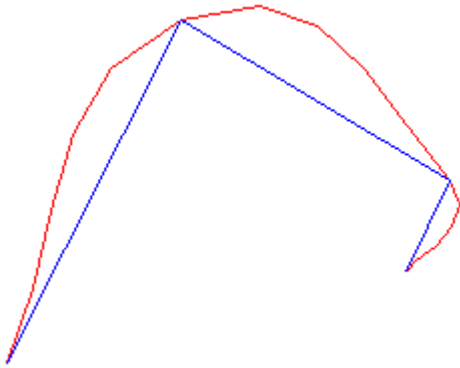
```
Loading C:\ALLENCAD\SUPPORT\flx_spln.lsp... OK.
```

```
Select 2D-polyline to alter to curved polyline:
```

```
Number of steps for interpolation points (0...6) <3>: 5
```

```
Tension (0.0 ... 1.0) <0.50>: 1
```

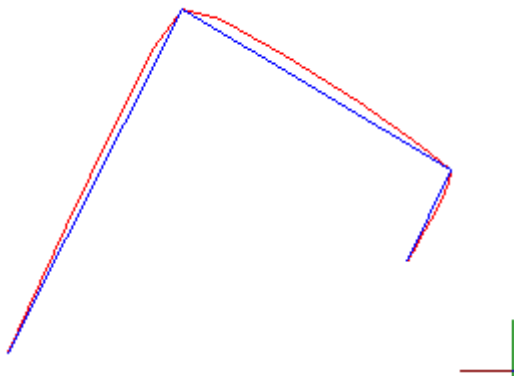
This gives us a rough bulging spline:



2. Next, we will input a high number of interpolation points and a low tension setting:

```
> PCURVE
Select 2D-polyline to alter to curved polyline:
Number of steps for interpolation points (0...6) <5>:
Tension (0.0 ... 1.0) <0.10>: .2
```

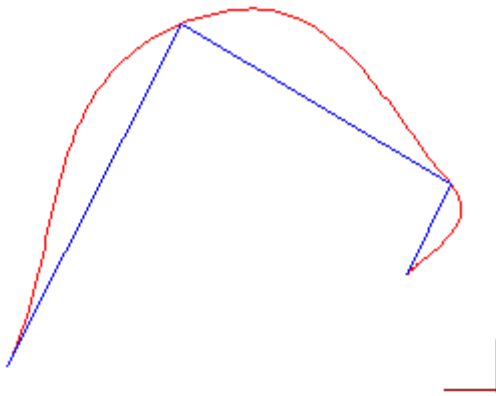
This gives us a rough tight spline:



3. Next, we will input a low number of interpolation points and a high tension setting:

```
> pcurve
Select 2D-polyline to alter to curved polyline:
Number of steps for interpolation points (0...6) <5>: 1
Tension (0.0 ... 1.0) <0.20>: 1
```

This gives us a smooth bulging spline:



3. Finally, we will input a low number of interpolation points and a low tension setting:

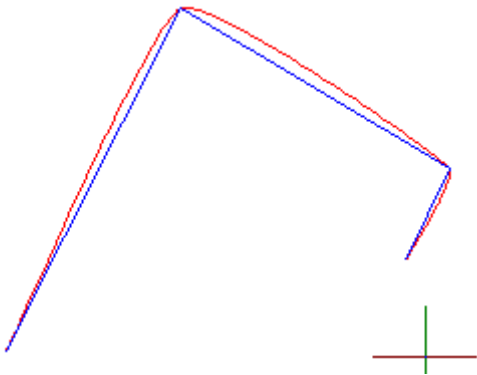
> PCURVE

Select 2D-polyline to alter to curved polyline:

Number of steps for interpolation points (0...6) <1>: 1

Tension (0.0 ... 1.0) <1.00>: .2

This gives us a smooth tight spline:



As shown in the examples, the two user inputs for this command are:

- **Number of steps for interpolation points (0...6)** – The number entered here (between 0 and 6) determines the relative length of the line segments that are used to simulate the spline. A **low** number of interpolation points within the range, produces Shorter segments and hence a smoother alignment.
- **Tension (0.0 ... 1.0)** – This input controls the "tightness" or "bulging" of the splines used in this transformation. A higher value (approaching) 1 will produce a spline that bulges out more from the original polyline alignment than a lower value as can be seen above.

## Polygon Face

**Command:** PFACE

**Alias** PF

### Located:

1. **Pull-down Menu:** Draw – 3D – Polygon Mesh
2. **Or Enter Command or Alias Directly at the Command Line**



The PFACE command creates three-dimensional polygon meshes of arbitrary topology and surface properties.

#### **To create a polyface mesh**

Specify the control points (vertices) first.

Then specify that vertex is on that face of the polyface mesh.

#### **Note:**

The PFACE command is rarely used as an explicit command in drawing practice. However, the command provides an easy way to generate meshes by Lisp routines based on 3D point lists (like surveyor's measurement points). The command is primarily designed for use within applications (either based on Lisp or on other programming languages) to construct general polyface meshes.

Do not confuse polyface meshes created by PFACE with polygon meshes created by the 3DMESH command.

#### **Example**

```
> PFACE
Control point 1: Specify a point
Control point 2: Specify a point
Control point 3: Specify a point
Control point 4: Specify a point
Control point 5: Specify a point
Control point 6: Specify a point
Control point 7: Specify a point
Control point 8: RETURN
Face 1, Control point 1: 1
Face 1, Control point 2: 2
Face 1, Control point 3: 3
Face 1, Control point 4: 4
Face 1, Control point 5: RETURN
Face 2, Control point 1: 1
Face 2, Control point 2: 4
Face 2, Control point 3: 5
Face 2, Control point 4: RETURN
Face 3, Control point 1: 5
Face 3, Control point 2: 6
Face 3, Control point 3: 7
Face 3, Control point 4: RETURN
Face 4, Control point 1: 4
Face 4, Control point 2: 5
Face 4, Control point 3: 6
Face 4, Control point 4: RETURN
Face 5, Control point 1: 3
Face 5, Control point 2: 4
Face 5, Control point 3: RETURN
Face 6, Control point 1: RETURN
```

Another null response terminates the command.

## **Plan View**

#### **Command: PLAN**

#### **Alias PLN**

**Located:**

1. **Entered Directly at the Command Line**

The PLAN command allows you to set the plan view to the World Coordinate System (WCS), to the current User Coordinate System (UCS) or to any other named UCS.

> PLAN

Set plan view to named UCS, World coordinate system, or press <Enter> for current UCS:

---

**See also:**

PLANVIEW

UCS

3DVIEW

Set Plan View to Construction Plane



**Command: PLANVIEW**

**Alias PV**

**Located:**

1. **Pull-down Menu: Tools – Set Plan View to Construction Plane**
2. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Pan Down**
3. **Left SideBar**
4. **Palette: View**
5. **Or Enter Command or Alias at the Command Line**

The PLANVIEW command sets the view planar to the current User Coordinate System (UCS). Normally, this command is used to return from an isometric 3-dimensional view of the drawing, to the standard (from above) view of the drawing.

---

**See also:**

PLAN

3DVIEW

UCS

## Part Library Setup...



**Command: PLBSETUP**

**Alias PLB**

**Located:**

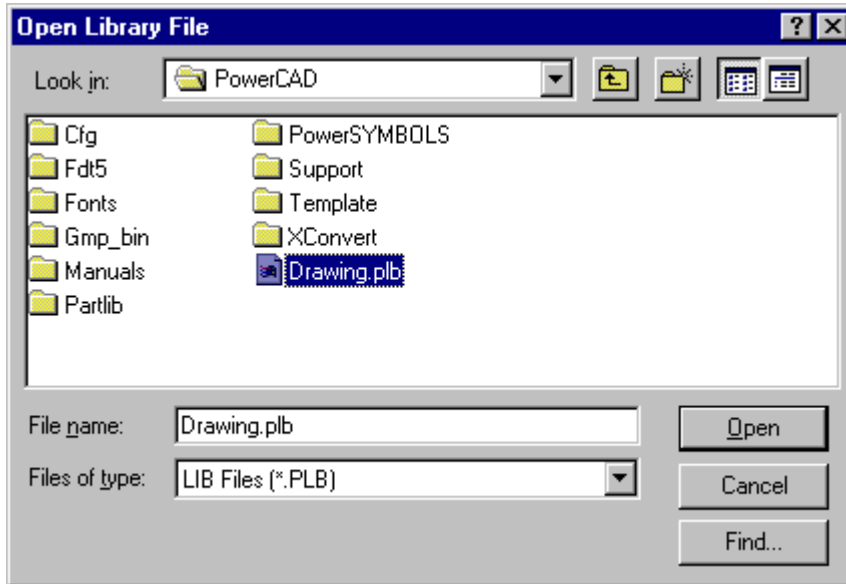
1. **Pull-down Menu: Tools – Resource Manager – Part Library Setup**  
**Also - Parts – Part Library Setup**
2. **Palette: Parts**

### 3. Or Enter Command or Alias Directly at the Command Line

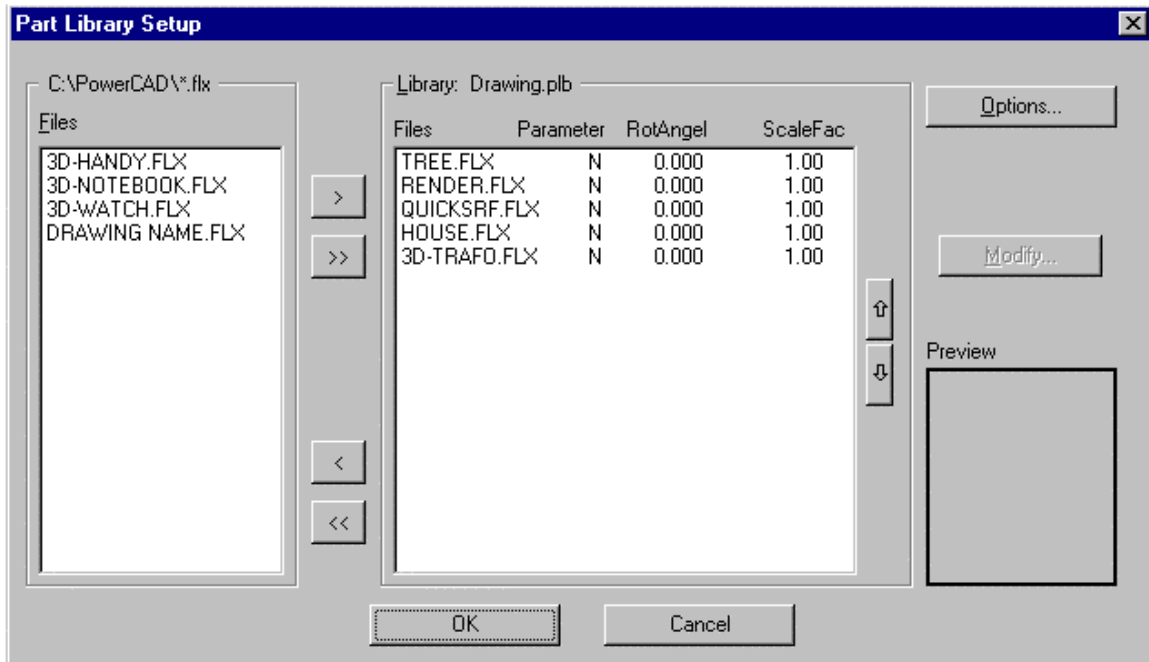
The command PLBSETUP (Part Library Setup) allows the user to setup up new part libraries and/or modify existing part libraries.

The command provides dialog boxes that:

- Select the drawing files to be included in a library;
- Set preferences for part or symbol insertions from the library; and
- Set the parameters that determine the layout and options of the part library Dialog Box.



PLBSETUP creates **.plb** files. These files store the parameters for a single part library. A part library is called by using the command PARTLIB, and then specifying the name of a .PLB file in the above dialog. No knowledge of structure or syntax of the .PLB format is required, when using the PLBSETUP command. Pressing the Open button opens up the next dialog:



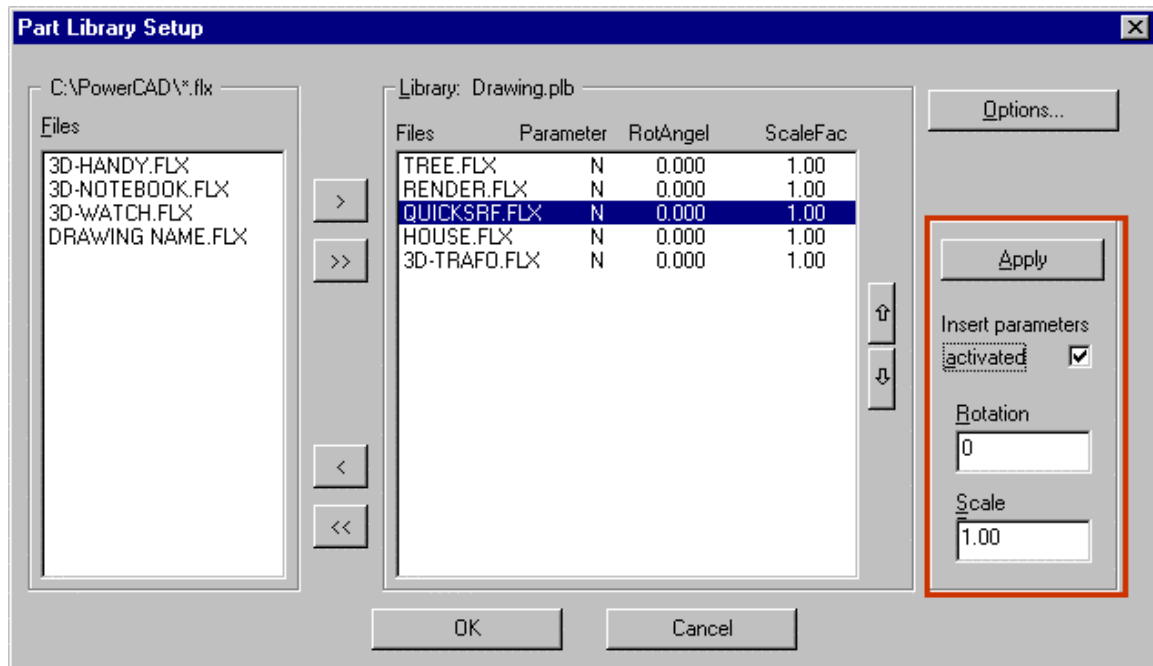
### Specifying the Drawing Files Belonging to a Part Library

The dialog box *Part Library Setup* allows the user to specify the drawing files to be incorporated into the individual library. He or she can **only choose drawing files from the programs Partlib subdirectory**. It is not necessary to include all of the drawings in a folder – simply select those that you want for the specific library that you are creating. Libraries allow the user to maintain all of the drawings used for part or symbol insertions within one directory, and at the same time permits the creation of multiple "thematically separated" libraries.

To add drawings to the current library, highlight the file names in the left side file list of the dialog box and then click to the button > to insert them to the file list making up the library (shown in the right frame). The buttons ( < and > ) permit both addition and removal of drawing files from the list of part library files. When only one specific file is selected from either of the list boxes, the preview image of the drawing will be displayed in the Preview Area. The buttons >> and << allow the user to easily move all file names from one list to the other.

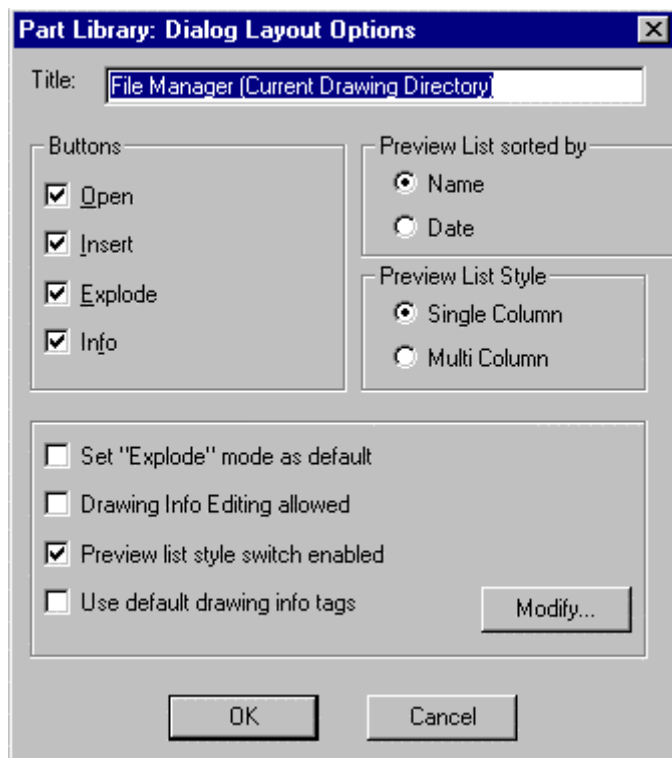
### Insertion Parameters

It is possible to pre-define insertion parameters for the files added to a library. Select one, more than one, or all entries from the list of the library files and then select **Modify**. Next, activate the option **Activate insertion parameters** and enter the default values of the *Scale Factor* and the *Rotation Angle* for insertion in the section outlined in red, below.



### Part Library Dialog Box Layout and Options

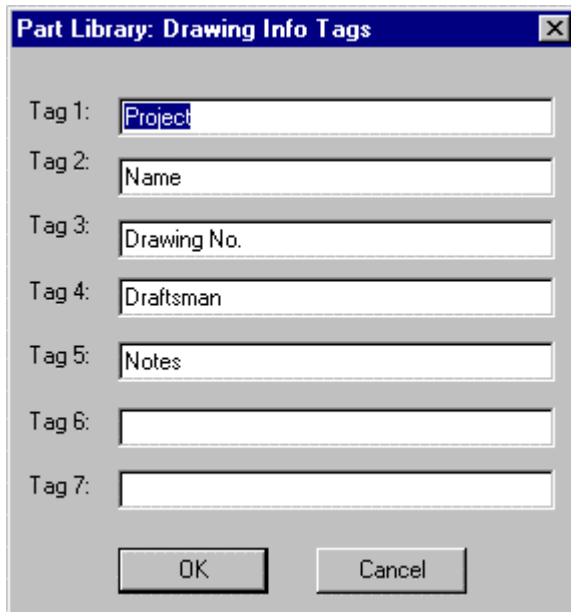
Using the Part Library **Options** dialog box, the user can determine the layout and the available buttons and options of the dialog box that comes up when calling this individual library with the PARTLIB command:



The edit box **Title** allows the user to enter a name for the dialog box of the part library.

### Preferences for the Preview List

- **Preview List Style** lets the user specify whether the part library Dialog Box should start up with a single column or multi column style of the list showing the drawing previews)
- **Sort by...** helps the user determine whether the drawing preview list should be sorted by name (usually determined for part libraries) or by date (usually determined for project libraries).
- **Buttons** assists the user in deciding options provided in the Dialog Box for a specific Part Library.
- **Open** allows the user to open a selected drawing for modification.
- **Insert** allows the user to insert a selected drawing into the current drawing as part.
- **Explode** allows the user to specify whether the part should be inserted as complex object (block), or exploded (broken down to single entities).
- **Info** allows the user to open the Drawing Information sub-dialog of the specific library.
- **Set "Explode" mode as default** sets the preference (activated/deactivated) for the mode of the button Explode.
- **Allow drawing info editing** specifies whether the user may alter drawing file information stored with the drawing database.
- **Enable preview list style switch** determines whether the library Dialog Box should contain the Quick Mode / Extended Mode button that, in turn, allows the user to toggle between single column and multi column style in the file preview list.
- **Use default info tags** determines whether default info tags (Project, Name, Drawing No., Draftsman, Notes) or user defined info tags are used when drawing information is displaced in the Drawing Information Dialog Box of the PARTLIB command.
- **Modify**, in the preferences area, allows the user to determine individual tags for the file info fields used in the specific library (for example; manufacturer, price, etc. in a library managing purchased parts). The dialog that appears with the Modify option is shown below:



---

See also:

## Construction Points



**Command: POINT**

**Alias POI**

### Located:

1. **Pull-down Menu: Draw – Construction Point**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**

It is possible to manually insert points of construction into a drawing using this command. Points can also be created in the drawing as the result of other commands and inputs, such as MEASURE and DIVIDE and the importation of survey data. The primary purpose of points is to serve as a reference in the construction of other drawing elements.

The form and size of the points is determined by specifying these parameters using the menu option *Drawing Parameters...* (See: Command DRAWMODE).

### Options:



**Point (Individual Points of Construction)**

This option is activated automatically when calling upon this command and thus does not have to be called up separately.

Determine the position of the points to be inserted either via keyboard or via pointing on the screen.

The command remains active after the insertion of the first point, and the user can repeat this procedure as often as is required. End this function by pressing RETURN.



**Measure (Using Construction Points to Measure)**

**Measure** allows the user to insert points of construction on 2-D lines, polylines, circles and arcs in a defined spacing.

This divides the entity into sections of a certain size. The drawing object is not cut into different pieces but remains one complete object.

Remember that all objects that are not an exact multiple of the length of the segment will have a remainder.

First choose a drawing element and determine the spacing of the segments (in drawing units), either by entering a numerical value or by cursor picks.

The insertion of the points into lines, polylines and arcs will be done beginning at the starting point of the segment and progressing to the end point. When inserting points of construction into a circle, the starting point lies at the intersection of the radius that has its starting point at the center point of the circle and is in the direction of the positive x-axis.

**Segments** makes it possible to separate lines, polylines, circles and arcs into segments of equal length by inserting points of construction. The size of the segment is determined by the length of the distance to be dissected and the number of construction points to be inserted. The DIVIDE command is the equivalent to Measure - Segments option.

In contrast to the option **Measure**, there will not be a remainder when using the **Segments** option.

The object being segmented is not cut into different pieces but remains one complete object.

First choose a linear object, and determine the number of segments into which the object is to be segmented. The number of segments should be a whole integer between 2 and 4096.

The insertion of points of construction into lines, polylines and arcs is done starting from the beginning of the segment and progressing to its end point, without inserting a construction point at the beginning or at the end. When inserting points of construction into a circle, the starting point lies at the intersection of a radius that begins at the center of the circle and continues along the positive x-axis. The beginning point is determined to be the end closest to the point at which the object was picked.

### Point Representation

Points can be shown onscreen in a number of different ways, apart from the common dot representation. The **SETTINGS** command's Construction Point tab has a number of alternate symbols that you may set as default. Note, however, that **all** construction points in the drawing will be shown with the same symbol, regardless of which you choose.

---

**See also:**

DRAWMODE

## 3D Polyline



**Command: POLY3D**

**Alias 3P**

### Located:

1. **Pull-down Menu: Draw – 3D – 3D Polyline**
2. **Or Enter Command or Alias Directly at the Command Line**

3D Polylines are drawing elements created from straight-line segments. It is possible to define 3D coordinates for each individual 3D Polyline control point (start and end points of the individual segments).

The user may include arc segments in 2D polylines, but this is not possible with 3D polylines. It should also be noted that each individual element is assigned a standard width of zero.

As opposed to a line characteristic created using the line drawing command, the program treats 3D polygon characteristics as a single item, this also applies when the object is being processed with the edit or undo functions.

3D Polylines may be created by selecting the option from the menu item **3D Entities**, in the **Draw** menu, or by typing in the command POLY3D.

Conclude the 3D Polyline function by pressing the RETURN key.

```
> POLY3D
First point: P1
Next point: P2
Next point: P3
Next point: P4
Next point: RETURN
```

### Options:

#### Undo

The **Undo** option allows the user to undo the last line created. The end point of the last remaining segment will act as the starting point for the next segment.

The **Undo** function can be repeated as many times as necessary until the Undo List is empty. Each



**Undo** step will erase the last drawn 3D polygon element.

### Close

This option enables the user to create closed 3D Polygon objects.

Either selecting **Close** from the options bar or typing the **close** command at the keyboard will cause the last two remaining end points to be joined together.

---

### See also:

2D Polylines

## Modify Polylines



**Command: POLYEDIT**

**Alias PE**

### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Edit Entire Polyline**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Polylines may be modified in the following fashion:

- Editing of a Polyline as a whole;
- Filleting of a polyline;
- Chamfering of a polyline; and
- Editing individual polyline segments by editing their control points.

### Polyline Editing

The command allows the user to:

- Open or close the Polyline;
- Join the polyline with other drawing entities such as line, arc or polyline;
- Change the width of the whole polyline; or
- Undo the last step.

The first option panel will display the Option Open or Close, depending on whether the selected polyline is currently opened or closed. Note that the option "Back" is only available after the execution of the first operation under the command.

### Close / Open

The option **Close** joins the start and end points of the polyline with a straight polyline segment. The width of this segment equals the ending width of the end point of the last drawn polyline.

The option **Open** is a possible option if the polyline to be edited has already been closed during drawing, or in a successive working step. The polyline segment between the start and end points of the polyline will in this case, be deleted.

### Join

**Join** allows lines, arcs and other polylines to be joined with the selected polyline. The prerequisite to this is that the polyline to be edited and the object to be joined meet in one ending point.

After the calling of the option *Join*, indicate the drawing entity to be joined to the previously selected polyline. The polylines chosen are now joined as one entity.

### **Width**

**Width** offers the opportunity to change the width of the polyline. Polylines that have different start and end widths as single segments, can be made a uniform width for all segments.

Define the desired width of the polylines by numeric input or indicating a width with the cursor. As a "help" parameter for this input, the value used last for the width of the polyline segment to be joined will be displayed. This value may be used by pressing RETURN.

### **Undo**

**Undo** will remove the last operation. The POLYEDIT command is continued, and the user will see the request "Input:".

---

### **See also:**

VTXEDIT

If the vertices only are to be edited that option may be picked from the sub-menu, or keyboarded by typing VTXEDIT.

## **Command POLYGON**

**Command: POLYGON**

**Alias PG**

### **Located:**

1. **Entered Directly at the Command Line**

The POLYGON command is a command variation of the NGON command that is used to draw polygons. Both commands operate in an identical fashion and the different command name is provided for users more familiar with the term "polygon".

### **Polyline**



**Command: POLYLINE**

**Alias PO**

### **Located:**

1. **Pull-down Menu: Draw – Polyline**
2. **Left Side Bar**
3. **Palette: Draw**
4. **Or Enter Command or Alias at the Command Line**

**Note: For boundary polylines see Command PCONTOUR.**



Polylines are drawing elements that consist of connected lines or connected arc segments. Different segments can be of different widths. The width of a polyline segment can also be defined differently for the starting and end-point.

A polyline is defined as a single element and treated as such for further operations (like Edit, or Undo), in contrast to continuous lines drawn with the command Line.

For Polyline Type control, please refer to the variable PLINETYPE.

Polylines will always be drawn with solid fill.

When you give the POLYLINE command, several options appear on the Options Bar or Menu::

- Arc
- Width
- Halfwidth
- Back
- Closing
- Append

Selecting Arc will make the next segment an arc,

Width will specify the width of the next segment, Tapered, if the initial (default) width is adjusted in the end width entry.

Halfwidth will specify a linewidth based in a half linewidth measurement. Like Width, the begin and end widths of each segment can vary, giving a tapered segment width.

Regardless of whether they are arcs or lines, polyline segments are always drawn by defining their starting and end as well as width.

The program expects that the user will want to draw more than one segment when drawing a polyline. After drawing one segment the user will be asked to name another end point for a new polyline segment. The starting point will be the end point of the segment last drawn.

The user can end the drawing of the **Polyline** by pressing RETURN or ESC.

- First point: **P1**
- Next point: **P2**
- Next point: **P3**
- ...

## Options

### Width / Halfwidth

The options **Width** and **Halfwidth** are available from the Options Bar to assist in defining the width of a polyline segment. **Width** defines the entire width of a polyline segment, whereas **Halfwidth** defines the width as the distance of an imaginary centerline to one side of the polyline.

Verify the last entered value by pressing RETURN or enter a new width manually or by "picking" it.

The following rules are in effect when using **both** options:

If the user chooses the option **Width** or **Halfwidth** before determining the starting point for the first polyline segment, then the chosen width is valid for the entire length of the segment.

If the user chooses the option **Width** or **Halfwidth** after entering the starting point for the first polyline segment, it is possible to enter different values for the width at the starting point and end point.

The last chosen width will be used for all following polyline segments on its entire length. To determine another width, choose the option **Width** or **Halfwidth** again.

## Arc

**Polylines** has an option for drawing arcs, and a width may be correlated to the arced polyline. The method of drawing arcs is identical to drawing elements using the command ARCS.

To draw polyline arcs with a special line width, follow these steps:

- Determine the first point of the arc. This can also be the end point of the previous polyline segment when drawing a polyline;
- Call up the chosen option, **Width** or **Halfwidth**, and enter the parameters for the starting width and ending width.
- Determine the other parameters for drawing arcs using the same procedure as drawing "regular" arcs.

## Append

**Append** allows the user to connect a polyline with another line or an arc.

Choose the element (line or arc) to which the polyline still to be drawn is to be appended. Call the options **Width** or **Halfwidth**, if required, and determine the parameters for the starting width and the ending width.

## Polyline to Line

To choose an object, select an already existing line. Remember that the polyline will be connected to the end point of the line that is closest to the point selected when choosing the element.

To append a polyline, the user must now determine the end point of the line to be connected. This point will be appended to the end point of the chosen existing line.

## Polyline to Arc

To append a polyline to an existing arc, the user must enter a polyline length after choosing the existing arc. A polyline with the specified length will be connected tangentially to the end of the arc.

To end the polyline (appending), press RETURN.

## Undo

The option **Undo** allows the user to "undo" the last step taken. The end point of the last segment may then be used as a starting point of another segment if required. Undo's can be repeated as long as a segment exists. The last segment will be erased with each successive UNDO.

## Close

The option **Close** allows the user to draw a closed polygon. The two end points of the chain will be connected to each other and the command will be terminated.

---

## See also:

[POLY3D](#)

Chain

## Precision Aids Parameters

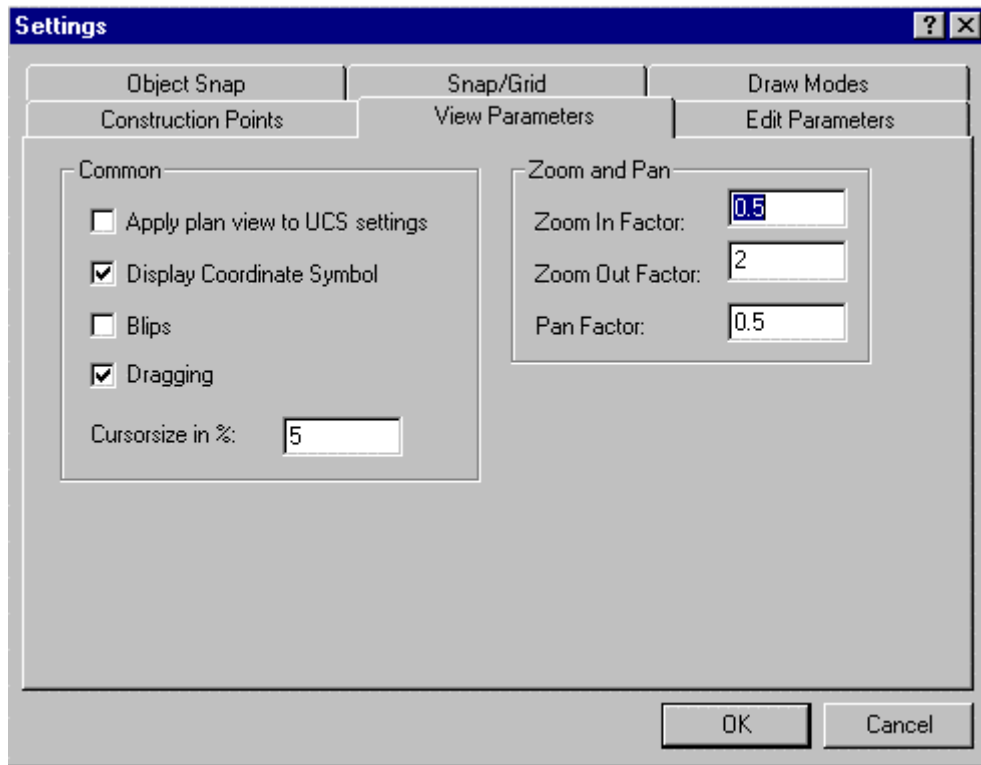
## Command: PRECPAR

## Alias PCP

**Located:**

1. **Pull-down Menu: Tools – Set Drawing Modes – Drawing With Precision**
2. **Or Enter Command or Alias Directly at the Command Line**

The PRECPAR command leads directly to various sections of the SETTINGS dialog, that control selection precision. Each tab has settings that can be adjusted to control drawing precision.



### Grid Display

Activate or de-activate the grid by clicking the check window on the upper left in the box **Grid Spacing**. A cross in this check box indicates the grid display is active.

In the input area **Factor** enter the value that defines the relation of the grid points to the snap grid. Alternatively, the user can specify a predefined value by clicking available buttons.

### Snap Grid

Activate or de-activate *Snap Grid* by clicking the check box on the upper left side in the area Snap Grid. A cross in the check box signals that the snap grid is switched on. From the dialog box, select the input fields Width and Height, and enter values for the horizontal and vertical distances between the points of the snap grid. Units of measurement for these values are, again, drawing units. The value 1.00 represents the distance of one drawing unit.

Using the palette button ( = ) for the value Width enables the user to choose a vertical value equal to the horizontal value.

The input fields marked "Origin" in this box enable the user to define the origin of the snap grid on the X and the Y-axis. The desired coordinates for the origin should always be entered.

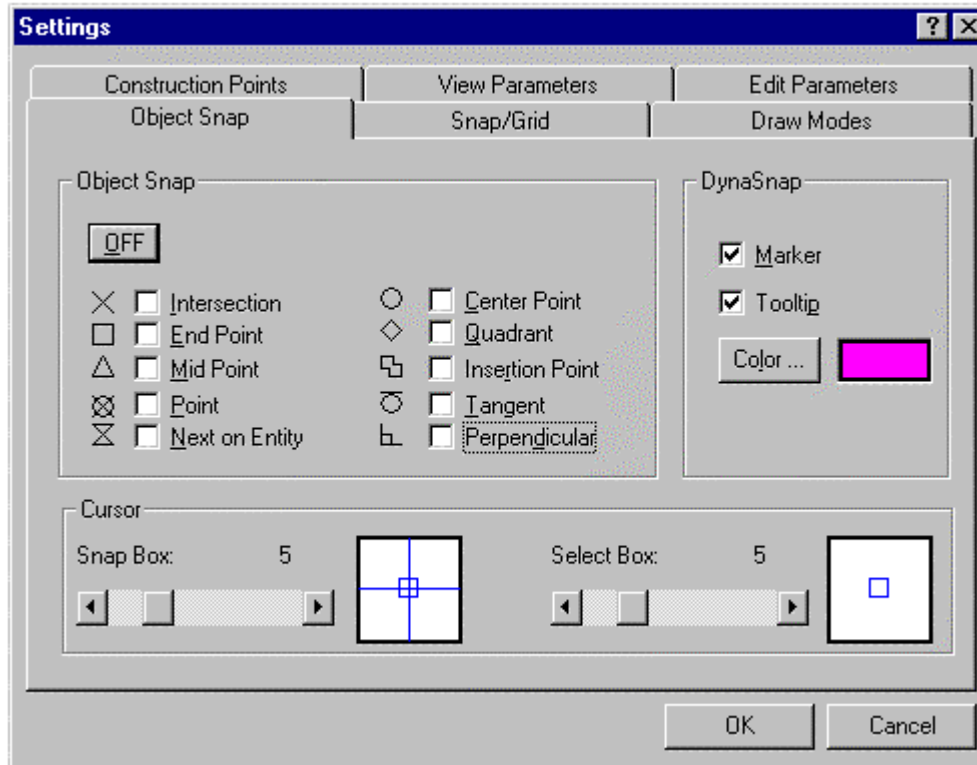
Sample

## Orthogonal Mode

Activate or de-activate **Orthogonal Mode** by clicking the check box.

ORTHOGONAL MODE: Drawing and Modifying Objects

## Object Snap



The Object Snap tab sets the active object snaps and controls whether dynasnap (the symbols for each are shown beside each object snap selection box) and tooltips are displayed by the program, or not. The color of both the dynasnap markers as well as tooltip color is also set on this dialog.

In the lower part of the dialog, you may specify the size (in pixels) of both the cursor snap and selection box.

## Cursor

Object Snap tab allows specifying the Select Box and Snap Box size.

View Parameters tab lets you control the size of the cursor axes.

## Drawing Output



**Command: PRINT**

**Alias PT**

**Located:**

1. **Pull-down Menu: File – Print**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

Start the output of a drawing on the printer/ plotter or plotter by selecting the option *Print ...* or by entering the command PRINT.

Select one of the options available in the options bar or menu (depending upon which you have selected for use in Desk Manager under Options. These options allow the user to determine the size and position of the print zone or the area of the drawing to be output. If you have Alternately, you may choose to use the first letter of the option to indicate your selection.

#### **View (V)**

The currently visible window of the drawing will be copied on the print.

#### **Border (B)**

This option determines the print zone so that all objects included in the drawing are covered.

Note that before using this option the drawing must be regenerated (Command REGEN).

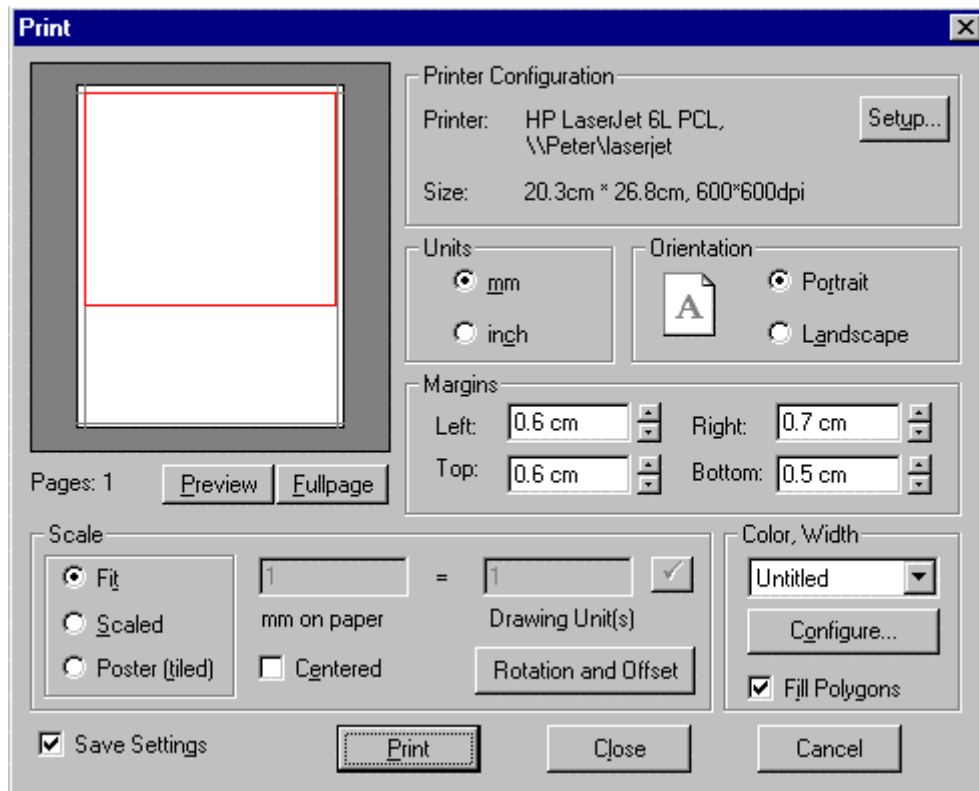
#### **Window (W)**

Window enables the user to determine the print zone by selecting a window in the drawing. Indicate or enter the coordinates of two diagonal opposite corner points inside the drawing.

The command PRINT controls output of drawings to printers or plotters configured on your system. It uses the Windows print drivers and by this requires that the appropriate drivers have been set up under Windows.

### **The Print dialog box**

After defining the size of the print, the Dialog Box *Print* will open, and additional print parameters required for the orientation of the print in relation to the edges of the paper format and to the picture size must be set.



The Dialog Box is divided into the areas Configuration, Layout, Color/ Width, Pages and Scaling. It also contains a preview window and selections for *OK*, *Escape* and *Setup*.

The *preview window* will assist the user in controlling the effects of setting changes in Layout and Scaling.

### **Printer Configuration Area**

The Printer/ Plotter configuration area contains the options for the current Printer/ Plotter, including the desired paper size and print resolution (in dpi).

#### **Setup**

Selection of the *Setup* switch allows the user to influence such parameters as the paper feed (printer/ plotter drawer), paper size, format position, resolutions and the number of the copies to be run.

Note: Only use settings supported by the selected device in the SETUP Dialog Box.

#### **Layout Area**

The parameters of the Layout Area control the size and position of the print on the available format.

##### **Margins**

Controls the area that is not to be printed (in mm) at the sides of the page. Enter the desired values in the proper window or select with the Cursor at the right edge of the fields and set the values for margin width.

##### **Fit**

Selecting the option *Fit* will result in a format-filling picture of the object in the print zone.

This option is activated by default during any print or plot output. To de-activate, click the control field. A cross in the control field displays the active status of the option.

Note that when the option *Fit* is active, it is not possible to set the options for scaling.

##### **Fill Area**

*Fill Area* will cause filled polygon areas to be displayed with fill in the print output. If the option not activated, only the contours of filled areas will be output.

The default position of this option is *Inactive*. To activate select the control field. A cross in the control field displays the *Active* status of the option.

##### **Center**

*Center* has a very close relation to the option *Fit*. Because the relation between the pages of the print zone remains unchanged during *Fit*, a format-filling picture may only be reached either to the length *or* width of the picture, but not with both.

*Center* results in a centered placement of the print zone on the paper in relation to the side that could *not have been fitted* to the margins of the page.

Note that *Center* is only available when using the *Fit* option.

##### **Area Scaling**

*Area Scaling* allows the user to define the exact scale of the drawing.

Note that *Area Scaling* is only available when the *Fit* option has been *de-activated*.

The definition of the picture scale is made in two steps:

1. Define the size of a drawing unit in mm. Place the cursor in the input field for the drawing unit - mm and enter the desired value.
2. In the input field Scale enter the scale for the print output or select a value by means of the arrow keys at the right side next to the input field.

The user can view the effects of the inputs in the preview window.

##### **Area Pages**

The *Area Pages* contain a Dialog. Enter the number of pages to be used in the drawing.

Multiple pages will result when the user has selected a picture scale that results in a print output larger than the selected page size (minus the defined margins).

##### **Area Color / Width**



By choosing the *Configure* button, the Color/Width Configuration Dialog will appear.

*Using saved stroke width-attachments*

Using saved stroke widths facilitates outputting drawings and assures constant settings.

In the selecting window of the area *Color/ Width* in the Dialog Window *Print* the current stroke width will be displayed. Selecting the arrow at the right side of the window will display a list of all saved configurations. Select the desired configuration.

### **Modify Object Properties...**



**Command: PROPEDIT**

**Alias PED**

#### **Located:**

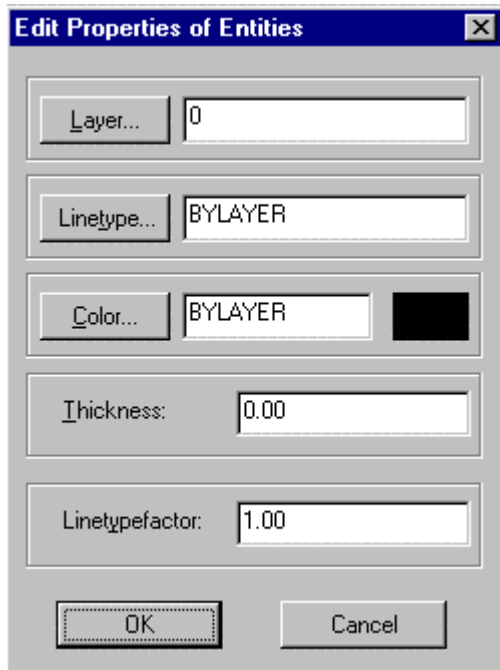
1. **Pull-down Menu: Edit – Modify Object Properties**
2. **Left Side Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

The function **Modify Properties...** invokes the dialog boxes that allow you to set the properties of entities, and make changes in **Color**, **Layer**, and **Linetype**.



As with the other commands in this set, first select one or more objects, using the selection dialog shown above, if required.

Then press RETURN, or use the right-hand cursor button.



The dialog box that appears displays the current properties Layer, Linetype, Color, Thickness and Linetype Factor (also called Linetype Scale Factor ) of the entities selected. You may then edit these properties.

Modify Object Layer ...



**Command: PROPLAYER**

**Alias PP**

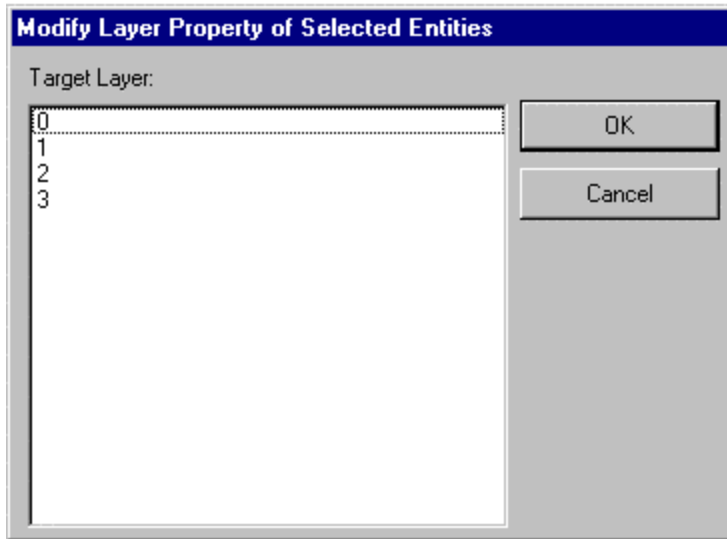
**Located:**

1. **Pull-down Menu: Edit – Modify Object Layer**
2. **Palette: Edit**
3. **Or Enter Command or Alias at the Command Line**

This command transfers selected objects from their current layer to a chosen layer.

**To modify the layer property of drawing objects:**

1. Select the drawing entities to be transferred to another layer.
2. In the dialog box (shown below) choose a target layer and confirm the selection by OK.



## Printer Setup

**Command:** PSETUP  
**Alias** PS

### Located:

1. **Pull-down Menu:** File – Printer Setup
2. **Or Enter Command or Alias at the Command Line**

The command PSETUP (Printer/ plotter Setup) or the selection of the option **Printer Setup...** from the *File* menu will start the selection and installation of the desired printer or plotter.

This procedure for installing printers calls up the standard Windows printer/plotter installation sequences.

The user will find detailed instructions regarding the operating mode of the specific options for the printer installation and for installing and connecting a printer or plotter in the Windows Manual or in the output device manual. Only the most important steps for printer/plotter installation are reviewed in this text.

The command will open a dialog box.

## Dialog Control Elements

### Printer

In the field Printer/ Plotter the user will specify the printer/ plotter or plotter to be used in the output of the drawing. Select either one of the standard devices defined in the Dialog Box control, or select another printer or plotter that is available under Windows. Available output devices are displayed in a list in the selection area under the heading *Special Printer*. To see the complete list, select the arrow at the right edge of the window. Select from the list of the installed output devices the desired printer.

### Orientation

Select the format **Portrait** or **Landscape** for the desired alignment of the paper.

### Paper

The size of the paper as well as the feed from a particular printer or plotter is controlled from the field **Paper**. Select the proper size and feed.

### Options

The **Options** switch (at the right edge of the Dialog Box) opens another Dialog Box that contains parameters that are mainly a function of the capability of the device to be connected.

## Command PULLDOWN

**Command:** PULLDOWN

**Alias** PN

**Located:**

### 1. Entered Directly at the Command Line

The PULLDOWN command operates in an identical fashion to the Menu command. It is used to call a different set of pull-down menus as defined in individual .MNU files. Please click on this link for a more detailed explanation of the commands use.

## Purge

**Command:** PURGE

**Alias** PRG

**Located:**

### 1. Pull-down Menu: Edit – Drawing Database Tables – Purge Tables

### 2. Or Enter Command or Alias at the Command Line

The PURGE command cleans the drawing file of unused named objects. You can remove blocks, layers, linetypes, text styles, or dimension styles that you have defined or loaded before but are not utilized in the drawing.

The command line prompt of the command is as follows:

> PURGE

Purge un-referenced symbols (Blocks, Dimstyles, LAyer, LTypes, Textstyles or All):

The PURGE command can be utilized on drawing database tables and table entry names *selectively*:

- You can use the command for deletion of unused named objects within only one of the drawing database tables. To do so, first select the type of table to be purged. You will then be prompted with the names - one after the other - of unused named objects of the specified table.
- If you want to purge all drawing database tables, reply *All* to clean up the entire drawing in one operation. You may respond with a name pattern to limit the selection to certain names by using wildcard specifications (for example: TEMP\*).

### To purge a drawing file

To purge all un-referenced symbols in any drawing database table, respond as follows:

> PURGE

Purge un-referenced symbols (Blocks, Dimstyles, LAyer, LTypes, Textstyles or All): **ALL**

Names to delete <\* = All>: \*

Confirm each name? <Y>: **N**

### Notes

Purging a drawing can significantly reduce its file size. This operation is recommended before transmitting, archiving or backing up a drawing file.

Some standard objects cannot be purged: These include the layer named 0, the linetype CONTINUOUS, and the text style STANDARD.

The PURGE command does not handle named views and user coordinate systems, because these named objects are never referenced by another component of the drawing. You may delete unused views and UCS. The corresponding commands (VIEW and UCS) provide options to delete named items from these tables.

With nested blocks in the drawing file, only one level of nesting is purged at a time.

## "Q"

### QARRAY command

**Command:** QARRAY

**Alias** QR

**Located:**

#### 1. Entered Directly at the Command Line

AutoCAD® compatible command line variant of the ARRAY command.

```
> QARRAY
Select objects:
Select type of array (Rectangular or Polar)? <R>: R
Number of rows <1>: 3
Number of columns <1>: 3
Distance between rows: 4
Distance between columns: 4

> QARRAY
Select objects:
Select type of array (Rectangular or Polar)? <R>: P
Center point:
Number of items: 8
Angle to fill <360>:
Rotate objects when copying? <Y>:
```

### Define Attribute (Command line variant)

**Command:** QATTDEF

**Alias** QAD

**Located:**

#### 1. Entered Directly at the Command Line

The command QATTDEF creates attribute definitions in the current drawing.

QATTDEF is the command line variant of ATTDEF that utilizes a dialog box to let you specify all the parameters required to define an attribute.

---

**See also:**

ATTDEF

### Edit Attributes (Command line variant)

**Command:** QATTEDIT

**Alias** QAE

**Located:**

**1. Entered Directly at the Command Line**

The command QATTEDIT allows you to edit attributes of block insertions in the current drawing.

QATTEDIT is the command line variant of ATTXEDIT that utilizes a dialog box to let you alter the attributes of a selected block.

---

**See also:**

ATTXEDIT

## Command QBMPOUT

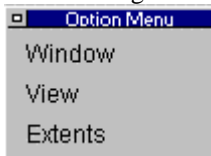
**Command:** QBMPOUT

**Alias** QB

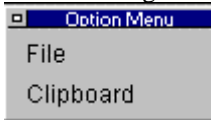
**Located:**

**1. Entered Directly at the Command Line**

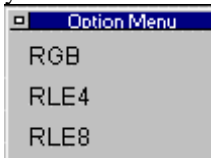
This is the command line version of command BMPOUT. When the command is first issued, you are prompted to specify the area that is to be saved in BMP raster file format and at the same time presented with a dialog containing area selection options to choose from.



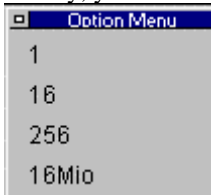
You are then asked to specify the width and height of the BMP image (in pixels) and then whether you want the image saved to clipboard or file. You are presented with a dialog to choose from.



Next, you are asked to confirm which BMP format to save the image in and presented with a dialog for your selection.



Finally, you are asked to specify the number of colors to be used in the saved file.



Upon completion of this input, you are asked to specify a filename (If you have designated saving as a file) and then the file is created.

Sample Input:

> QBMPOUT

Specify region to handle (Window, View, or Extents): Window

First corner:

Second corner:

Width (10...2000 pixel) of bitmap <290>:

Height (10...2000 pixel) of bitmap <298>:  
Output to (File or Clipboard): File  
Specify bitmap format: RGB  
Number of colors: 16  
Filename: Sample

## Text Style Definition (Command Line Version)

**Commands:** QFONT or STYLE

**Alias** QF

**Located:**

### 1. Entered Directly at the Command Line

The command QFONT, or STYLE, allows the user to load text styles from the specified font file and customize it to individual requirement.

These commands are command line variants of the FONT command.

```
> STYLE
Style name (or ?) <STANDARD>:
New text style definition.
Font file <TXT.FSH>:
Text height <0.00>:
Text style width factor <1.00>:
Text style obliquing angle <0>:
Backward alignment (Yes or No) <N>:
Upside-down alignment (Yes or No) <N>:
DIM set as current text style.
```

---

**See also:**

Command FONT

QFONT and STYLE are the command line versions of the FONT command.

## Command QHATCH

**Command:** QHATCH

**Alias** QH

**Located:**

### 1. Entered Directly at the Command Line

This is the command line version of the HATCH command. When it is issued, you are prompted to select the objects forming the boundary of your hatch. You are next asked for the hatch pattern name to be applied. You are then asked to confirm or overwrite default values for the pattern's angle and factor. The Hatch pattern is then applied.

Command Input sample:

```
> QHATCH
Select objects:
1 selected.
Select objects:
*** 1 selected. ***
```

Pattern name :ansi31  
Pattern angle <0>:45  
Pattern factor <1.00>:2

## QHPEDIT command (Command Line Version)

**Command: QHPEDIT**

**Alias QHE**

**Located:**

1. **Entered Directly at the Command Line**

QHPEDIT is the command line version of the HPEDIT command provided for usage in Lisp routines, Macros, etc.

The command QHPEDIT works like the command HPEDIT but with one difference: QHPEDIT is performed at the command line.

---

**See also:**

Command HPEDIT

## Insert Part ...



**Command: INSERT**

**Alias I**

**Located:**

1. **Pull-down Menu: Parts – Insert Part**
2. **Palettes: Draw and Parts**
3. **Or Enter Command or Alias Directly at the Command Line**

**Command: QINSERT**

**Alias QI**

**Located:**

1. **Pull-down Menu: Parts – Quick Insert Part**
2. **Or Enter Command or Alias Directly at the Command Line**

The command INSERT offers broad support for the user in its Dialog Box. The command QINSERT is the command line version of the command. The command macro QINSERT ? allows fast insertion of internal parts (parts already defined in the drawing).

The command INSERT provides selection areas for:

- Selection of external parts;



- Definition of the insertion point, scale factors, and rotation angle, with future choices of using coordinate/ numeric input or cursor selection;
- Parameter values may be reset; and
- The Part may be "exploded" into its components.

### **Selecting a Part**

To select the Part to be inserted, type the name of the Part (or select an existing Part) in the field Part Name. Use the buttons External and Internal to select the correct Part listing.

### **Defining the Insertion Point**

Defining of the Insertion Point either by entering the proper coordinate values, or by selecting Interactive, and "picking" the point with the cursor.

### **Scale Factor**

The Scale Factor input areas will offer 1.0 as the default XYZ scale factor. This will insert a part at its original size. Remember that a scale factor greater than 1.0 will increase the size of the part, and a factor less than 1.0 will reduce the size of the part to the percentage of the original size specified by the Factor. A factor of 2 will result in the part being twice it's original size, while a factor of 0.5 will be half original size.

By default, AllenCad prompts you for x and y scale factors only. If you need to specify a z scale factor, simply respond to the X scale factor prompt with an xyz response. You will then be prompted for all three scale factors in turn.

The option *Interactive* allows the user to define the size of the Part during the insertion as well. To insert the Part using the Interactive option, simply define by opposite corners the rectangle into which the Part is to fit.

Since dragmode is active during this operation, the user will have the benefit of seeing the selection results before confirming the selection.

### **Rotation Angle**

The Rotation Angle can be set from the Dialog Box. Enter a value for the Rotation Angle in the proper input window. For counter-clockwise rotation enter a positive value, for clockwise rotation enter a negative value between 0 and 360 degrees

### **Default**

Default allows the user to reset the parameters for the Insertion Point, the Scaling Factor and the Rotation Angle back to the original parameter values.

### **Exploded**

Exploded allows the user to "explode" the part into its component entities. Exploding will cause the loss of the characteristics that were retained as a part.

### **Troubleshooting**

If you find that the blocks you are inserting are not placed in the specified location, check the settings for INSBASE and ELEVATION in both drawing files. These should be set to 0,0,0 and 0 respectively, if you do not want them to affect your block insertion.

## **QLAYER (Command Line Version)**

**Command: QLAYER**

**Alias QL**

**Located:**

1. **Entered Directly at the Command Line**

The command QLAYER is the command line variant of the LAYER command.

> QLAYER

Select option (Make, Set, New, ON, OFF, Color, Ltype, Freeze, Thaw, LOkk, Ulock, or ?):

---

**See also:**

Command LAYER

Layer Attributes

## QMIRROR command

**Command: QMIRROR**

**Located:**

1. Entered Directly at the Command Line

AutoCAD® compatible command line variant of the MIRROR respectively FLIP command.

> QMIRROR

Select objects:

First point of mirror axis:

Second point:

Delete original objects? <N>:

## Creating and Viewing Slides

**Commands: MSLIDE**

**MSLIDE is Located:**

1. Enter Command or Alias at the Command Line

**VSLIDE (Alias VS),**

**VSLIDE is Located:**

2. Pull-down Menu: View – View Slide
3. Or Enter Command or Alias at the Command Line

**SLDOUT (Alias SO)**

**SLDOUT is Located:**

1. Pull-down Menu: View – Take Snapshot (Slide)
2. Or Enter Command or Alias at the Command Line

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.

You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

#### **To create a slide file**

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.
3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

#### **To view a slide file**

Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

## **Command QMTEXT**

**Command:** QMTEXT

**Alias** QM

#### **Located:**

1. **Entered Directly at the Command Line**

This is a command line version of MTEXT, where multiline text is generated without the MTEXT dialog box. It begins by requesting that you pick opposite corners of a rectangle, upon which the program establishes its justification requirements. You are presented with the familiar MTEXT options dialog that should be used when other than default MTEXT values are required.

You then simply type in the text at the command prompt and it immediately appears onscreen as typing progresses. Pressing the Return key twice in a row terminates the command and pressing the Esc key cancels the command in progress.

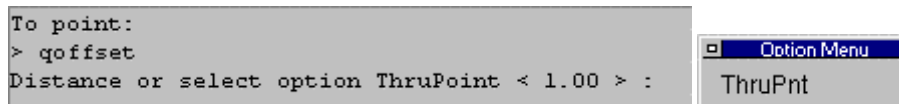
## **Command QOFFSET**

**Command:** QOFFSET

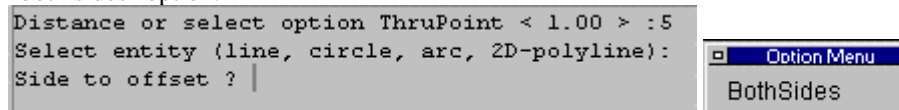
#### **Located:**

1. **Entered Directly at the Command Line**

This is the command line version of the OFFSET command. When the command is issued, you are prompted at the command prompt for a distance to offset as well as presented with a dialog offering the option of placing the offset through a specified point. .



You are then prompted for the side to offset to at the same time that a dialog appears, offering you the "both sides" option.



Sample Input:

```
> QOFFSET
Distance or select option ThruPoint < 1.00 > :5 (default 1 overwritten by an entry of 5)
Select entity (line, circle, arc, 2D-polyline): (Pick)
Side to offset ? (Pick)
Select entity (line, circle, arc, 2D-polyline): (Select next object to offset, etc.)
```

## Command QOPEN

**Command:** QOPEN

**Alias** QO

**Located:**

### 1. Entered Directly at the Command Line

This is the Command line version of the OPEN command. When it is issued, it simply asks for a filename to open. This file is then opened in a new window, leaving the existing file open as well.

## Command QPRINT

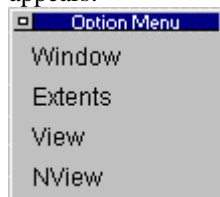
**Command:** QPRINT

**Alias** QP

**Located:**

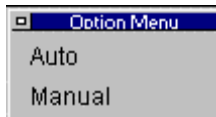
### 1. Entered Directly at the Command Line

This is the Command line version of the PRINT command. When the command is issued, a dialog appears:



Here, you designate the area to be printed.

```
IE. > QPRINT
Specify region to print: Window
First corner:
Second corner:
```



A second dialog appears offering you the chance to select printer setups other than the one currently set. Pressing the Return key twice confirms default settings are to be used and the print is sent to the printer:

Configuration <Auto>:  
Print 1 page(s). Press <Return> to start output!

## Change Properties (Command Line Version)

**Command:** CHPROP , QPROPEDIT

**Alias** CHP, QPR

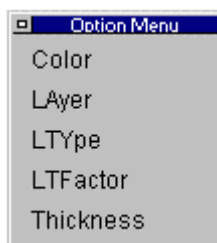
**Located:**

### 1. Entered Directly at the Command Line

These commands modify the properties of drawing objects.

They are the command line variants of the PROPEDIT command.

When you issue the command, you will be asked to select objects, and be presented with the object selection dialog to help you make this selection. Next, you will be prompted at the command line to identify which properties you want to change. A dialog appears (as below) on screen to help you enter your selection.



After making your selection, you will be asked to enter a new value for the property you want changed.

As with most commands, command line input can be used, instead of using the dialog. You simply type in the option abbreviations as are shown bold in the following text screen recording. The requests of these commands are as follows:

```
> CHPROP
Select objects:
Specify property to change (Color, LAyer, LType or Thickness):
```

---

**See also:**

PROPEDIT

## Command QSAVE

**Command:** QSAVE

**Alias** QS

**Located:**

### 1. Entered Directly at the Command Line

This command simply saves the current drawing without any prompts. The file is saved in the current directory and with the filename as is currently assigned.

## Command QSLDOUT

**Command:** QSLDOUT

**Alias** QSL

**Located:**

### 1. Entered Directly at the Command Line

This is the command line version of the SLDOUT command. When the command is issued, you are asked to provide a filename for the .SLD file to be created. After this, the screen image is saved to this new file.

Sample Input:

> QSLDOUT

Filename: sample

## QTEXT command (Command Line Version)

**Command:** QTEXT

**Alias** QT

**Located:**

### 2. Entered Directly at the Command Line

The QTEXT function is used to create text entities in the current drawing.

The command QTEXT works like the command TEXT but with one difference: QTEXT is performed at the command line.

>QTEXT

Specify start point or select option (Justify, Style): **Justify**

Specify justification (Align, Fit, Center, Middle, Right, TL, TC, TR, CL, CC, CR, UL, UC, UR): **Middle**

Insertion point: **10,10**

Text height <0.20>: **0.40**

Insertion angle <0>:

Text: **3D-Trafo**

---

**See also:**

TEXT

QTEXT is the command line version of the TEXT command.

## Set Named View (Command Line Version)

**Command:** QVIEW

**Alias** QV

**Located:**

## 1. Entered Directly at the Command Line

Command line variant of the VIEW command.

> QVIEW  
Delete/Restore/Save/Window/?:

---

**See also:**

VIEW command

## Layout Viewport Control (Command Line Version)

**Command: QVIEWPORT**

**Alias QVP**

**Located:**

### 1. Entered Directly at the Command Line

The command QVIEWPORT allows you to create rectangular areas displaying the model while in layout mode and to control the view direction or display portion of the model in those viewports.

The command QVIEWPORT works like the command VIEWPORT but with one difference: QVIEWPORT is performed completely at the command line or used in macros or LISP files..

---

**See also:**

VIEWPORT command

## Creating and Viewing Slides

**Commands: MSLIDE**

**MSLIDE is Located:**

### 1. Enter Command or Alias at the Command Line

**VSLIDE (Alias VS),**

**VSLIDE is Located:**

2. **Pull-down Menu: View – View Slide**
3. **Or Enter Command or Alias at the Command Line**

**SLDOUT (Alias SO)**

**SLDOUT is Located:**

1. **Pull-down Menu: View – Take Snapshot (Slide)**
2. **Or Enter Command or Alias at the Command Line**

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.

You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images

from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

#### **To create a slide file**

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.
3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

#### **To view a slide file**

Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

## Command QWMFOUT

**Command:** QWMFOUT

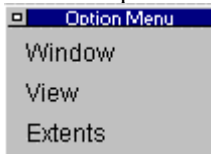
**Alias** QWO

### **Located**

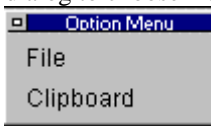
#### **1. Entered Directly at the Command Line**

This is the command line version of the WMFOUT command.

When the command is first issued, you are prompted to specify the area that is to be saved in WMF (Windows Metafile) raster file format and at the same time presented with a dialog containing area selection options to choose from.



Then you are asked whether you want the image saved to clipboard or file. You are presented with a dialog to choose from.



Upon completion of this input, you are asked to specify a filename (If you have designated saving as a file) and then the file is created.

Sample Input:

```
> QWMFOUT
Specify region to handle (Window, View, or Extents): View
Output to (File or Clipboard): File
Filename: junk ple
```



## Copy a Drawing Viewport

**Command:** QWOPEN

**Alias** QW

### Located:

1. **Pull-down Menu:** Window – Copy Current Viewport 1:1
2. **Or Enter Command or Alias at the Command Line**

The option **Copy Viewport** on the *Window* menu allows you

1. to make a quick copy of the current viewport or
2. a plan view, by choosing the desired option button.

## Link Externally Referenced Drawings (Command line variant)

**Command:** QXLINK, XREF

**Alias** QX

### Located:

1. **Entered Directly at the Command Line**

The command QXLINK, or XREF, allows you to attach and manage externally referenced drawings (XREF's).

QXLINK, and XREF, are command line variants of the XLINK command that utilizes a dialog box to let you link and handle externally referenced drawings.

> XREF

<Attach>, Bind, Detach, Reload, Path, Open, Insert:

---

**See also:**

XLINK

## "R"

## Raster Frame On/Off



**Command:** RBORDER

### Located:

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

The rectangular frame border around the images can be turned off or on by using this command.

All frames are affected at the same time. When on, the frame shows the outer edge of the image and is in the same color as the image.

## Render Settings



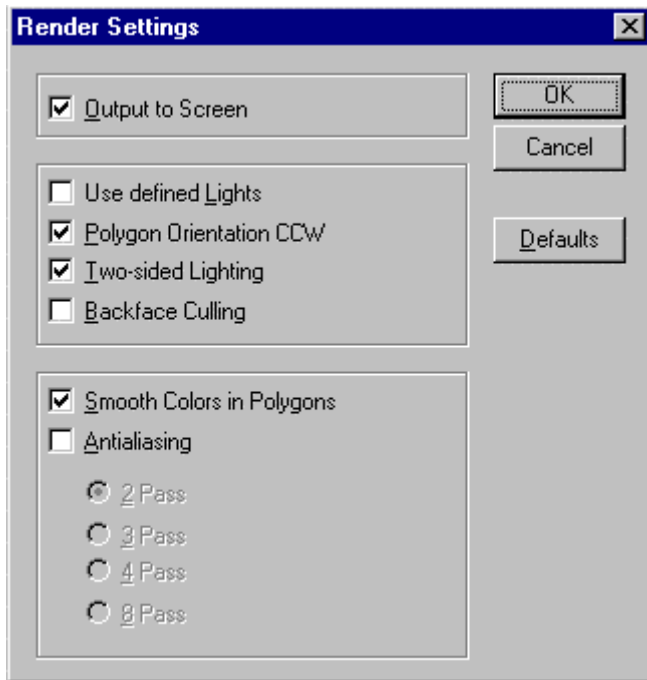
## Command: RCONFIG

### Alias RC

#### Located:

1. Pull-down Menu: View – Render – Render Parameters
2. View Palette
3. Or Enter Command or Alias at the Command Line

The command RCONFIG displays a dialog box to view and set general preferences for subsequent rendering.



The button **Defaults** resets the preferences to default values (shown in the illustration above). This is convenient, because a number of settings can influence the performance of render computations dramatically.

The option **Output to Screen** controls whether the rendered polygons are output on the screen (to the front buffer) during computations or are written into a display buffer before presenting the rendered image. If this check mark is not set, the rendered image is not displayed before the entire rendered image is computed.

The option **Use Defined Lights** allows you to activate or deactivate the use of light sources. If disabled, the program generates - in computer graphic terms - a simple **shading** of the 3D model, otherwise the inserted lights are utilized during the rendering.

Three other options allow users who are familiar with computer graphics and advanced rendering techniques to save time when careful drawing methods have been used during the creation of a 3D model: **Polygon Orientation CCW**, **Two-sided Lightning**, and **Backface Culling**. It is recommended to use the default values if you are not familiar with these terms.

The option **Smooth Colors in Polygons** specifies whether upon each individual polygon surfaces smoothing colors are computed or not. Applying an adequate light environment allows smooth coloring on individual surfaces.

The option **Anti-Aliasing** allows you to control the process of smoothing jagged edges caused by aliasing. Aliasing is a term in computer graphics. It describes certain display effects like the staircase

effect seen on diagonal lines displayed in a drawing. Anti-Aliasing can reduce those artifacts.

Turn anti-aliasing on for high-quality rendered images. To improve the rendering significantly you can choose between **2, 3, 4 or 8 passes** with the anti-aliasing algorithm. Because multiple passes will increase the rendering time dramatically, anti-aliasing should be applied only for high-quality image output (usually when rendering to a file).

By default the anti-aliasing option is disabled. It produces a faster output for viewing a rendered image on the graphics display, and you do not normally notice the elimination of artifacts on the computer screen.

---

**See also:**

RENDER

## Cut Raster with Polyline Boundary



**Command: RCUT**

**Located:**

### **1. Palette: Raster**

### **2. Or Entered Directly at the Command Line**

This command applies to imported or internally converted RLC format graphics only. If you have a portion of the raster that you do not want and you wish to remove it from the job you can use this command to do so.

You must first draw a closed polyline around the area to be clipped. The polyline **MUST** be a series of straight line segments, arcs will be treated as if the chord of the arc was the boundary, and you **MUST** close the polyline.

You will be prompted to pick the raster image to clip, just as in the move image and rotate image commands.

Select raster /<point>/?/Last/#n [#3]:

Once it knows the correct raster image you will be asked to pick the polyline to use for the clipping. Then the program will ask you

Save properties for 'C:\RLC\_Images\MyImage.rlc ?':

If you answer NO then the image is simply clipped and that portion falling within the polyline boundary is removed from the screen and the properties remain unchanged. If you answer YES the following is displayed;

Properties saved for 'C:\RLC\_Images\MyImage.rlc'.

and then the image is cut or clipped inside of the polyline and the properties are saved for the original raster image.

## Command FLXRECOVER

**Command: FLXRECOVER**

**Located:**

### **1. Pull-down Menu: File – Recover**

### **2. Or Enter Command or Alias at the Command Line**

Beginning with the ancestral FelixCAD 4.01 (Service Pack2) release, the FLX drawing format modifications permit the partial recovery of damaged drawing files. This utility, called by the RECOVER command, is therefore available to FLX files saved in this (or later) release format. For damaged drawing files saved in AutoCAD DWG or DXF format, applicable AutoCAD recovery utilities should be used.

Usually, the need to use this utility becomes apparent when you experience difficulties opening an existing drawing file. File damage may apply to only a portion of the drawing database and the recovery utility will help you to salvage undamaged drawing elements from the file.

Upon issuing the command, select the damaged filename in the dialog presented.

## Rectangle



**Command: RECTANGLE**

**Alias RT**

### **Located:**

1. **Pull-down Menu: Draw - Rectangle**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**

The command RECTANGLE allows you to construct rectangles of any size and side ratios.

To draw a rectangle, enter or pick two opposite corners.

When the command is active, the option *Width* is available in the Options Panel. The option **Width** allows you to specify a value for the line width applied on the rectangle creation. To use the last entered value (that will be offered by the prompt) just press RETURN.

### **Examples**

```
> RECTANGLE
First point of the rectangle: P1
Opposite corner point: P2

> RECTANGLE
First point of the rectangle: Width
Width <0.05>: 1.00
First point of the rectangle: P1
Opposite corner point: P2
```

## Command REDEFINE

**Command: REDEFINE**

**Alias RE**

### **Located:**

1. **Entered Directly at the Command Line**

This command restores AllenCad internal commands overridden by UNDEFINE and is issued at the command line.

When the command is issued, you are prompted for a command name. Enter the name of a AllenCad command turned off by the UNDEFINE command. If a command has been undefined, you can still use it if you precede the command name with a period.

## Redo (Restore an Undo Operation)



**Command: REDO**

**Alias RD**

**Located:**

1. **Pull-down Menu: Edit – Redo**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The command REDO offers the ability to Redo an Undo. The state of the task is restored to the last state prior to issuing the command U (or UNDO).

This is only possible if no other command has been executed between the commands U (or UNDO) and REDO, that have changed the drawing database.

REDO is used in this fashion:

- Call the Redo function by clicking the Redo symbol, or
- Enter REDO at the command line.

**Note:**

Only the last relevant working step can be restored.

---

**See also:**

Command U

## Redraw



**Command: 'REDRAW**

**Alias R**

**Located:**

1. **Pull-down Menu: Window – Redraw**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Or Enter Command or Alias at the Command Line**

Refresh current drawing window.

Regenerate

**Command: REGEN**

**Alias RG**

**Located:**

1. **Pull-down Menu: Window – Regenerate**
2. **Or Enter Command or Alias at the Command Line**

Regenerate current drawing.

## Regenerate All

**Command: REGENALL**

## Alias RGA

### Located:

1. **Pull-down Menu: Window – Regenerate All**
2. **Or Enter Command or Alias at the Command Line**

Regenerate all drawings.

## Rejoin broken Lines or Arcs



**Command: REJOIN**

## Alias RJ

### Located:

1. **Pull-down Menu: Modify - Rejoin**
2. **Palette: Modify**
3. **Or Enter Command or Alias at the Command Line**

Rejoin will unite separated co-linear lines segments or arc segments that share the same center and the same radius. The resulting joined line or arc will be a single entity.

A common function for this command is to reunite separated line or arc segments that have been divided by the command DELPARTIAL (Delete partial).

Select the command, then pick the entities to be rejoined.

> REJOIN Select first entity (line or arc): *P1*  
Select second entity (line or arc): *P2*

---

### See also:

**Delete partial**

**Rename**

**Command: RENAME**

## Alias RNM

### Located:

1. **Pull-down Menu: Edit – Drawing Database Tables – Rename Table Items**
2. **Or Enter Command or Alias at the Command Line**

The RENAME command allows you to alter the names associated with blocks, layers, linetypes, text styles, dimension styles, user coordinate systems, and views.

### To rename a drawing database table entry

- First, specify the drawing database table that contains the item to be renamed.
- Next, enter the name of the named object you want to rename.
- Finally, type in the new name of the item.

### Example

> RENAME

Rename symbol (BLock, Dimstyle, LAyer, LType, Style, Ucs, or View): **LA**

Current name: dimensions

New name: dim

### Tip

You can rename these named symbols used within the current drawing from within the *Drawing Explorer*.

## Render



**Command: RENDER**

Alias **REN**

### Located:

1. **Pull-down Menu: View – Render – Render**
2. **Palette: View**
3. **Or Enter Command or Alias at the Command Line**

Rendering allows you to create an image of your 3D drawing that illustrates very realistically the model or plan. To increase the realism, the rendering may perform smooth shading on the objects that meet at a specific angle and evaluate the lights set in the drawing.

Using rendering, you visualize and present your 3D geometry as it would appear, after construction.

To render the current viewport of a drawing, call the command **RENDER**. The command is immediately performed after you have specified where to display or output the rendered view:

| Screen | File | Clipboard |

The rendering is based on the preferences set with the **RCONFIG** command and evaluates the light sources found in the model. The command **LIGHT** allows you to place lights in the drawing and define their type and settings. The command **LIGHTEDIT** is used to modify the settings of light sources.

---

### See also:

**RCONFIG**

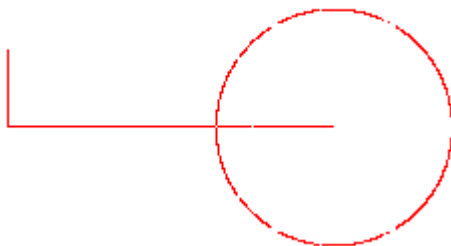
**Surface of Revolution**

**Command: REVSURF**

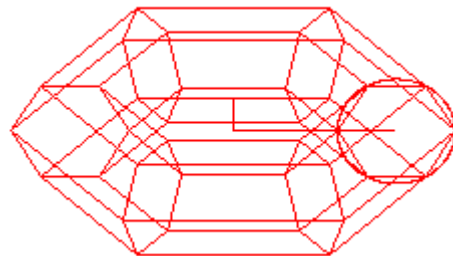
Alias **RS**

### Located:

1. **Pull-down Menu: Draw – 3D – Surface of Revolution**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**



**To:**

The REVSURF command creates a surface by revolving a path curve (the profile) around an axis of revolution.

The curve sweeps around the selected axis to define the surface in the *N* direction of the mesh. The *M* direction of the mesh is determined by the axis of revolution. The result is determined by the SURFTAB1 and SURFTAB2 settings.

#### To generate a surface of revolution

First, pick the entity (a line, arc, circle, or polyline) to revolve. This determines the profile entity.

Next, pick the axis (a line or open polyline) around that the profile entity revolves.

Finally, you are prompted to determine the start angle and the included angle to revolve the profile. The default is a full circle.

```
> REVSURF
Select path curve to revolve:
Select axis of revolution:
Start angle <0>:
Included angle to revolve entity (+=ccw, -=cw) <360>:
```

#### Tip

To close the mesh in the *N* direction, select a circle or closed polyline as the curve.

Please check in the Learning Center for a fully documented example of this command's use.

## Rings



**Command: RING**

**Alias RI**

#### Located:

1. **Pull-down Menu: Draw - Ring**
2. **Palette: Draw**
3. **Or Enter Command or Alias Directly at the Command Line**



A ring is a special case of a closed polyline. It is used to draw solid rings or circles. Two concentric circles are drawn, the smaller one is the inner diameter, and the larger one the outer diameter, of the ring. The space between the circles will be filled.

Rings can now be edited using any of the editing commands for polylines and will behave just like a polyline object.

The last entered values will be used as an option for the next entry when drawing rings. If the user wants to keep using the same values, just enter a new center point. This step can be repeated as often as needed, always creating identical rings.

```
> RING
Center point: P1
Center point: P2
...
```



## Options:

### Diameter

To determine new values for the inside or outside diameter, just choose the option **Diameter** from the options panel.

With this option it is possible to set the inside and outside diameter of the rings, where new values can be input either via keyboard or by picking points on the screen.

```
> RING
Center point of the ring: Diameter
Inside diameter <0.500>: 0.75
Outside diameter <1.00>: 1.25
Center point: P1
Center point: P2
...
```

With command repeat terminated by Esc, it is possible to draw as many identical rings as required by simply locating their successive center points.

### Filled Circle

A filled circle is a special case of rings. It is a ring with the interior diameter equal to zero.

## Command\_RLC2BMP

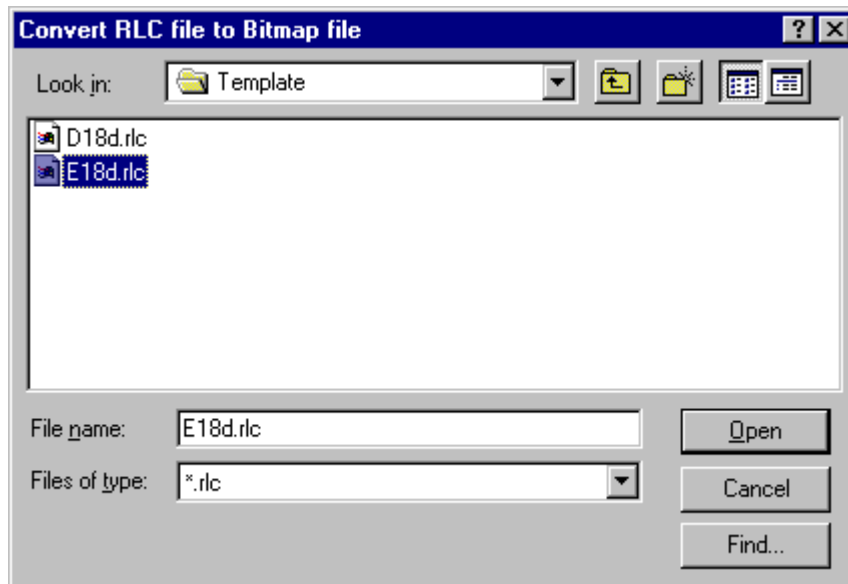


### Command\_RLC2BMP

#### Located:

1. **Palette: Raster**
2. **Or Enter Command Directly at the Command Line**

Like its counterpart (BMP2RLC), RLC2BMP converts an external RLC graphic file into bitmap (BMP) format. When the command is issued, the following dialog appears:



To complete command execution, simply navigate to your RLC file through the "Look in:" drop down window and select the file. The file is then immediately converted with the new file remaining in the same location as the original and having the same filename but with .bmp extension.

## Load Raster Image File

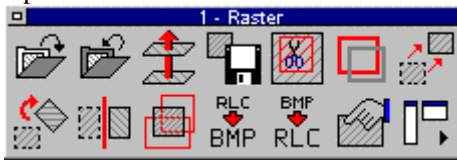


### Command RLOAD

#### Located:

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This item allows you to insert a scanned, raster image that has been stored in the RLC, TIF or BMP format, into your current drawing, thus allowing you to see the raster image and the vector information at the same time. In addition, you can import any graphic that has been placed on the AllenCad clipboard by a previous COPYSELECT or CUTSELECT command.



As with most Raster commands, you can find this command on the Raster palette that is placed on the drawing editor when you click on the Raster entry on the View pull-down menu. This command cannot be entered directly from the keyboard.

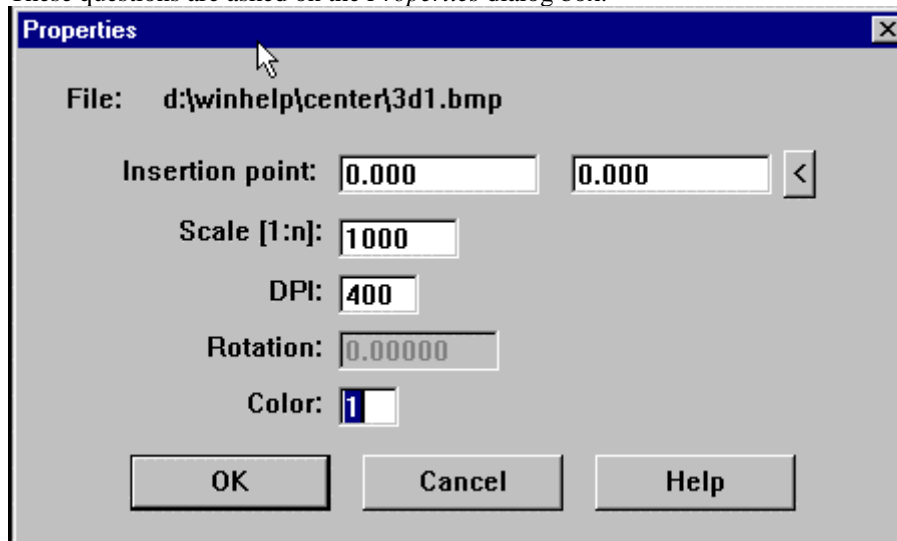
There are many different formats that a scanned image can be stored in such as BMP, GIF, CALS, TIFF, etc., and RLC.

At this time the program will only edit RLC format and import RLC, BMP and TIF files. AllenCad has a utility program that converts files from BMP to RLC formats, however, there are many other conversion programs on the market that may assist you in changing from another format to RLC.

When you start the routine, you will be shown the following dialogue box to allow you to select the raster image you wish to insert into the drawing. You must specify the format of the file that you are trying to locate.

Once you have selected the file to load, you will be asked where to insert the image, the scale to insert it at, the color of the image, the DPI (resolution in dots per inch), and the rotation factor.

These questions are asked on the *Properties* dialog box.



Once you have entered the correct information, your raster image will then appear on screen.

## Move Raster Image



**Command: RMOVE**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command moves an raster image on your screen. If you have more than one raster image in the drawing, you will be asked to select the one to be moved in the following prompt;

Select raster /<point>/?/Last/#n [#0]:

You can point at the raster image, choose Last to pick the last one selected, or enter in the number of the image and the ? mark displays all the numbers to choose from.

You will then be asked for a Base point and a Destination point, just like the standard move command.

## Mirror Raster Image



**Command: RMIRROR**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command allows you to mirror image the image of your choice.

You must first select the correct image on screen:

Select raster /<point>/?/Last/#n [#0]:

After selecting the image, you will be asked;

Mirror axis:

There will be two options showing on the button bar - Horizontal and Vertical. Pick the one you wish to use.

You now have the option to edit the image properties.

The original image is not removed and the mirrored image will be displayed where and how the image properties are set.

## Assemble / Fit Raster Files



**Command: RMONT**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command lets you designate a portion of an imported RLC file for saving in a new file after defining the area by window selection. In effect, you will be saving a "cropped" image in a different file.

You will be prompted for the First Corner and then the Second Corner.

After picking the area to copy, you will be asked to confirm the properties of this new image.

The new file will only be created after you save the property changes and you select the Save Raster button on the palette. You will also be prompted to save the file when you remove the raster, if the file has not been saved to this point.

### Rotate



**Command: ROTATE**

**Alias RO**

**Located:**

1. **Pull-down Menu: Modify - Rotate**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Rotate will serve to rotate an entity or entities through an angle, about an axis.

As with the other modification commands, Rotate requires the selection of the object/ objects to be rotated. There are several methods suggested by the Option Bar of selecting the object(s). Choose a method, and make the selection.

**Options**

**Basepoint**

The command will prompt for the Basepoint. The Basepoint is the point about that the rotation will occur. It is handy to pick a corner, or a center, but any point may be picked that will give the desired result. Enter, or pick, the Basepoint.

Note that the Basepoint may not be a point on the described geometry of the object or objects, but may fall off of their geometry.

The next prompt will ask for the angle of rotation. The user may elect, through the Option Bar, either the **Angle of Rotation**, or a **Reference Angle**.

**Angle of Rotation**

The option Angle of Rotation is the standard option. It is used if no other option has been selected.

Angle will prompt the user for a rotation angle. Enter that angle, and rotation will take place to the specified angle.

**Reference Angle**

The Reference Angle will ask the user to specify the angle at that the object is now placed, and then prompt for a New Angle, to which the object will be rotated. Rotation will always be counter-clockwise. As before, rotation will occur about the base point.

## Edit Raster Image Properties

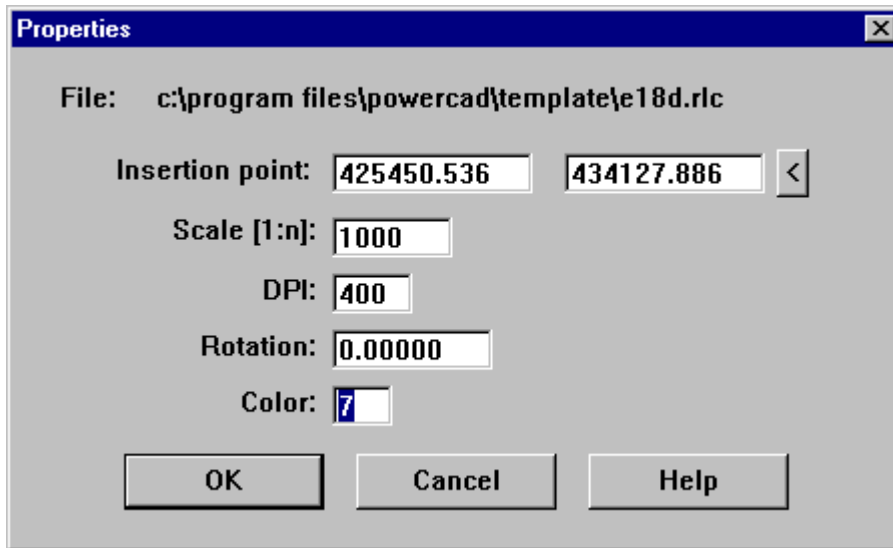


**Command: RPROP**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command brings a dialog box up that allows you to change the properties of an imported raster image. It raises a dialog in which any of the fields can be modified as needed. The changes are only done in the current drawing and will not affect the original image on the hard drive.



Insertion point:

Location to insert the image on your drawing.

Scale [1: ]:

Sets the relative scale factor to match your drawing.

Scales the raster image larger or smaller.

DPI:

Dots Per Inch. This sets the resolution of the Image.

Rotation:

Affects the orientation of the image in reference to the drawing.

Positive rotation is counterclockwise, negative rotation is clockwise.

Color:

Sets the color the image will take on when inserted into a drawing.

The whole image is the same color.

Each raster can be brought in with it's own color.

## Rotate Raster Image



**Command: RROTATE**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This command applies to imported or internally converted RLC format graphics only. You can rotate the selected raster image to match your vector drawing by using this command. The prompt is as follows;

Select raster /<point>/?/Last/#n [#0]:

This allows you to select the correct raster image to rotate, similar to when you move a raster image.

Rotation angle:

This is the amount of rotation, clockwise is a negative rotation and counterclockwise is a positive rotation.

## Save Raster Image and Properties



**Command: RSAVE**

**Located:**

1. **Palette: Raster**
2. **Or Entered Directly at the Command Line**

This function allows you to save all of the changes to the image, done by the edit image properties item, back to your hard drive. As edits can be made on RLC format graphics only, this command is ineffectual for other formats, unless they have been converted by the program from RLC format after editing.

The following message is displayed.

Properties saved for 'C:\RLC\_Images\MyImage.rlc'.

**Note:**

These changes are permanent so be careful and make sure you have a backup of the original raster image before saving the changes. The save function

### Reference Text...



**Command: RTEXT**

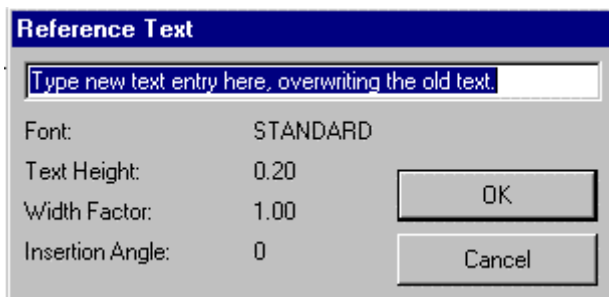
**Alias RX**

**Located:**

1. **Pull-down Menu: Detail – Reference Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

This command will permit you to select regular text entries (not mtext entries) and enter new text with the same characteristics as the text entry selected.

When you issue the command, you will be asked to select the text that you want duplicate the properties of. Once you have done this, the following dialog will appear:



Simply overwrite the text that appears in the dialog box and press OK. Lastly, you will be asked to specify the location of the new text entity in the drawing. Simply drag the text to the location required and click to anchor it. The new text will have the same font, height, width factor, and rotation angle as displayed in the dialog, along with the same layer and color.

The command scheme is:

```
> RTEXT
Select reference text:
<Dialog box to edit text>
Insertion point:
```

---

**See also:**

TEXT

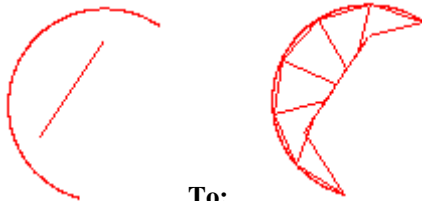
## Ruled Surface

**Command:** RULESURF

**Alias** RSF

**Located:**

1. **Pull-down Menu:** Draw – 3D – Ruled Surface
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**

The RULESURF command allows you to create a ruled surface between two boundary curves (two profiles).

Valid entities that define the edges of the surface can be points, lines, arcs, circles, 2D polylines or 3D polylines.

The system variable SURFTAB1 determines the number of equal intervals between the resulting vertices on the defining profile entities. The number of intervals is the same for each curve.

The resulting surface is a polygon mesh (stored in the drawing database as an complex polyline entity).

### To create a tabulated surface

First pick one entity defining the first profile, then select another entity for the other boundary outline:

> **RULESURF**

Select first defining curve of ruled surface:

Select second defining curve of ruled surface:

### Note

If the first entity is an open entity (such as an arc or line), the second entity must also be an open entity. If the first entity is a closed entity (such as a circle or a closed polyline), the second entity must also be a closed entity. If one entity defining an edge is a point, the other entity cannot be a point entity.

### Tip

To create a commonly used polygon mesh, select each entity at the same end. If you select each outline entity at opposite ends, you generate a self-intersecting polygon mesh.

Please check in the Learning Center for a fully documented example of this command's use.

## Remove Raster Image



**Command** RUNLOAD

**Located:**

1. **Palette:** Raster
2. **Or Entered Directly at the Command Line**

This option will remove the selected image from the drawing and optionally allow you to save any changes to the properties of that image.

The following prompt is displayed;

Save properties for 'C:\RLC\_Images\MyImage.rlc ?':

If you answer NO then the image is simply removed from the screen and the properties remain unchanged. If you answer YES, the following is displayed:  
Properties saved for 'C:\RLC\_Images\MyImage.rlc'.  
The image is then removed from the drawing.

## Command RUNMACRO

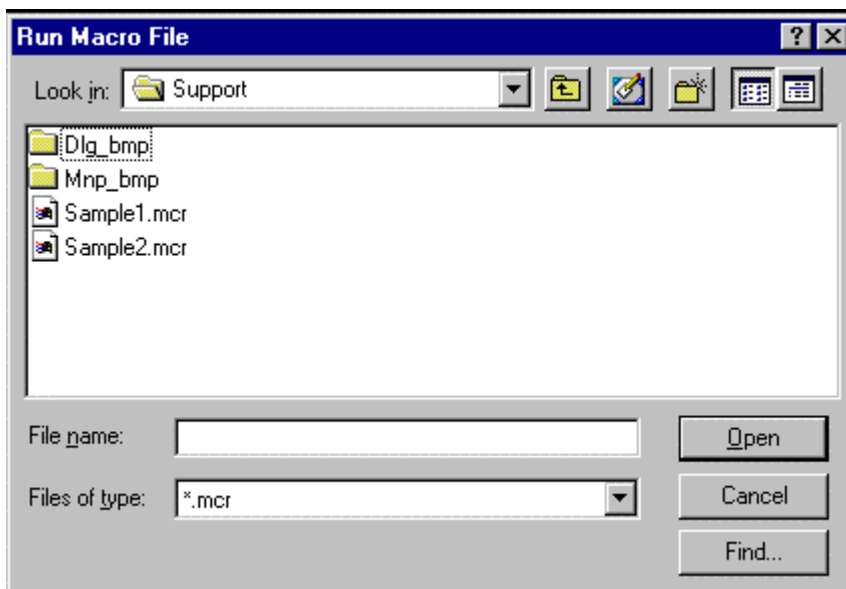
**Command: RUNMACRO**

**Located:**

1. **Pull-down Menu: Tools – Resource Manager – Run Macro**
2. **Or Enter Command or Alias at the Command Line**

Unlike a LISP program that remains in memory for repeated use, Macros (sometimes referred to as Script files) are executed as they are loaded. The RUNMACRO command can be found on the Tools pull-down menu as "Resource Manager – Run Macro" selection, or else it can be simply typed at the command prompt for execution.

The following dialog appears once the command has been issued, allowing you to browse to the macro file you want.



The macro is executed as soon as selection has been confirmed.

## Transformation Vectors



**Command: RVECTORS**

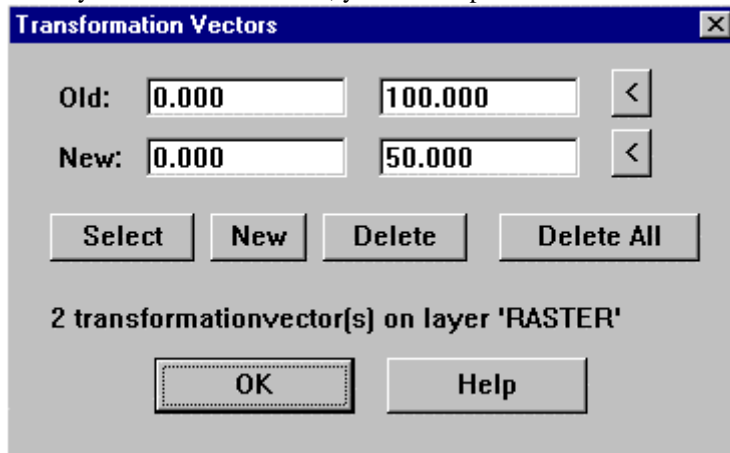
**Located:**

1. **Palette:Raster**
2. **Or Entered Directly at the Command Line**

This command allows you to stretch your imported RLC graphic to the extent and with a rotation as indicated by the relative length and alignment of two lines (vectors)



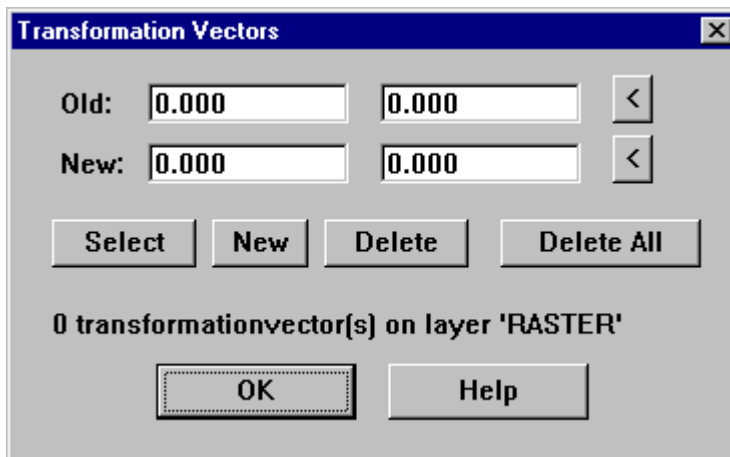
Before issuing the command, you will need to create a new layer called "Raster". When you issue the command, you will be presented with the following dialog:



The dialog box titled "Transformation Vectors" has a close button (X) in the top right corner. It contains two rows of input fields. The first row is labeled "Old:" and has two text boxes with values "0.000" and "100.000", followed by a button with a left-pointing arrow. The second row is labeled "New:" and has two text boxes with values "0.000" and "50.000", followed by a button with a left-pointing arrow. Below these are four buttons: "Select", "New", "Delete", and "Delete All". At the bottom, it says "2 transformationvector(s) on layer 'RASTER'" and has "OK" and "Help" buttons.

This dialogue box allows you to enter and edit the transformation vectors required to adjust the raster image to match the vector drawing.

The vectors you define must be drawn on the layer RASTER, and you can actually draw the vectors before starting this command and it will detect them for you. Simply draw one line (vector) on the raster image and then a second line (vector) to match the size, location and orientation of the first line after the raster is transformed to match your drawing elements.



The dialog box titled "Transformation Vectors" has a close button (X) in the top right corner. It contains two rows of input fields. The first row is labeled "Old:" and has two text boxes with values "0.000" and "0.000", followed by a button with a left-pointing arrow. The second row is labeled "New:" and has two text boxes with values "0.000" and "0.000", followed by a button with a left-pointing arrow. Below these are four buttons: "Select", "New", "Delete", and "Delete All". At the bottom, it says "0 transformationvector(s) on layer 'RASTER'" and has "OK" and "Help" buttons.

## Old Vector Coordinate

This is displaying the current starting location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

## New Vector Coordinate

This is displaying the current ending location of the currently selected transformation vector. You can type in a new location if you wish or use the button to the right to allow you to visually pick a location on the screen.

## Select Button

This button will let you select that vector you are currently editing by allowing you to pick that vector on screen.

## New Button

This button will create a new vector from the coordinates displayed in the OLD and NEW fields above. It will duplicate a vector if you pick this button while old values are in the fields above but you can still

manipulate those vectors by using the two pick buttons to the right of the NEW and OLD coordinates displayed.

## Delete Button

This button will delete the currently selected vector, both from the routine and the screen.

## Delete All Button

This button will delete all the vectors on the layer RASTER.

## Raster Layer Display

This location displays the number of vectors detected on the layer RASTER.

Note: Do NOT use this layer for anything else or you will confuse the program.

## Pick Button for New Coordinate

This button allows you to pick a new location for the vector to end. This is the ending location of the transformation for this vector. You can pick any where on the screen as the original coordinate for the vector.

## Pick Button for Old Coordinate

This button allows you to pick a new location for the vector to start. This is the starting location of the transformation for this vector. In other words, the amount of transformation will be determined by the start and end of the vector. The amount of transformation is the delta X and delta Y of this vector. You can pick any where on the screen as the starting (old) coordinate for the vector.

# "S"

Save Drawing File



Command: SAVE

Alias S

### Located:

1. Pull-down Menu: File - Save
2. Top Function Bar
3. Or Enter Command or Alias at the Command Line

## Save

The command SAVE causes the drawings in the active window to be saved. At this time the file will be saved under its present name. The file is "saved" in the same drawing format and to the same storage device and area as was specified at the time of opening. The file last saved in this area will be replaced.

The command to save can also be invoked from the menu *File* or by clicking on the symbol *Save* in the Function Bar.

Local variables will be saved along with the drawing and the enclosed drawing entities current at the time the automatic save occurs. Such settings as the current coordinate system, the dimensioning mode and other adjustments will remain at the user's disposal.

## Auto-Save

To avoid unforeseen loss of data we strongly recommend periodically saving drawings while working on them. For this purpose use the function Save Automatically that can be activated in the configuration dialog window. While an automatic save is occurring the current drawing will not be closed.

---

**See also:**

SAVE AS...  
SAVE ALL  
OPEN  
NEW  
CLOSE

## Save All

**Command:** SAVEALL

**Alias** SAL

**Located:**

1. **Pull-down Menu: File - SaveAll**
2. **Or Enter Command or Alias at the Command Line**

The SAVEALL command will not only save the drawing in the current window but also all other open drawings.

---

**See also:**

SAVE  
SAVE AS...  
CLOSE

**Save As...**

**Command:** SAVEAS

**Alias** SA

**Located:**

1. **Pull-down Menu: File - SaveAs**
2. **Or Enter Command or Alias at the Command Line**

The command SAVEAS saves the drawing in the current window, as the command SAVE does. However, it allows the user to save the drawing as a different type, name, or in a different area. The command SAVEAS will require that the user answer these questions, all contained in the dialog box:

Filename  
File Type  
Drive  
Folder

The command will then save the work done as a specific file type (.dwg, .dxf or .flx), under a new filename, or in a new storage area. At the same time, a small bitmap (thumbnail) image of the drawing file is also created and stored with the file. This enables users to view the image before opening the file, thereby simplifying file identification.

---

**See also:**

SAVE  
SAVE ALL  
CLOSE

## Command SAVEAS21

**Command: SAVEAS21**

**Located:**

**1. Entered Directly at the Command Line**

This command is used to save the current drawing in a format compatible with Version 2 and is entered at the command line. Please note that information incompatible with Version 2 format will not be included in the new file.

## Command SAVEAS40

**Command: SAVEAS40**

**Located:**

**1. Enter Command at the Command Line**

The SAVEAS40 command will save the current drawing file in FelixCAD 3 and 4 FLX drawing format. This is primarily intended to help you exchange drawing files with users of earlier FelixCAD program versions.

See Also: SAVEAS21

## Scale



**Command: SCALE**

**Alias SC**

**Located:**

- 1. Pull-down Menu: Modify - Scale**
- 2. Palette: Modify**
- 3. Or Enter Command or Alias Directly at the Command Line**

The **Scale** command allows the user to change the "scale" of the objects that is, to enlarge or reduce them while keeping all proportions the same.

The first prompt will be to select the object(s) to be scaled. Use one of the options presented in the Options Bar for this. Remember that if more than one object is selected, the objects will be treated as one by the command.

The next prompt will be to identify the **base point**, and that should be done as it was in the two previous commands. This will bring up the Options Bar with the following options:

Sc.Factor | Reference

Options

## Scaling Factor

The option **Scaling Factor** is the standard option for scaling objects; and is used if no other option has been selected.

Note that it is important to realize that the **base point**, if it is a part of the selected object, will hold its original position during the scaling, while all the other points will be moved by the same relative values in direction of the X- and Y-axis. As in other cases, however, the user may select any other point, whether on or off the geometry of the object(s). In this case, the distance from the base point to all of the other points that comprise the object will be changed by the same scale factor.

To continue, enter a scaling factor. The use of a scaling factor bigger than 1 results in enlarging the selected objects by this factor. A scaling factor between 0 and 1 results in reducing the objects by that factor.

For instance, the factor <3> enlarges the selected objects three times, while the factor <0.25> results in reducing the objects to one-quarter of their original size.

## Reference

This option allows an exact change of the size of an object to a particular scale.

For instance, if the user were to enlarge an object with a side length of 3.45 drawing units up to 6.90 drawing units, he or she would first have to calculate the scaling factor. **Reference**, however, facilitates this calculation by the input of a reference length (here 3.45) a precise changing of the size up to the desired scale (here 6.90), without the user having to calculate a scaling factor.

To use this variation of scaling, select the option **Reference** from the Option Bar. Enter the **Reference length**, and then enter the **New length** that is desired.

## Select



**Command: SELECT**

**Alias SL**

### Located:

1. **Palettes: Edit and Modify**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The SELECT command allows you to choose drawing objects "in advance".

In subsequent modification commands the user can refer to the selection by applying the *Previous* option.

## Send Drawing(s)...

**Commands: SENDFLX, SENDALL**

**Alias SD, SDA**

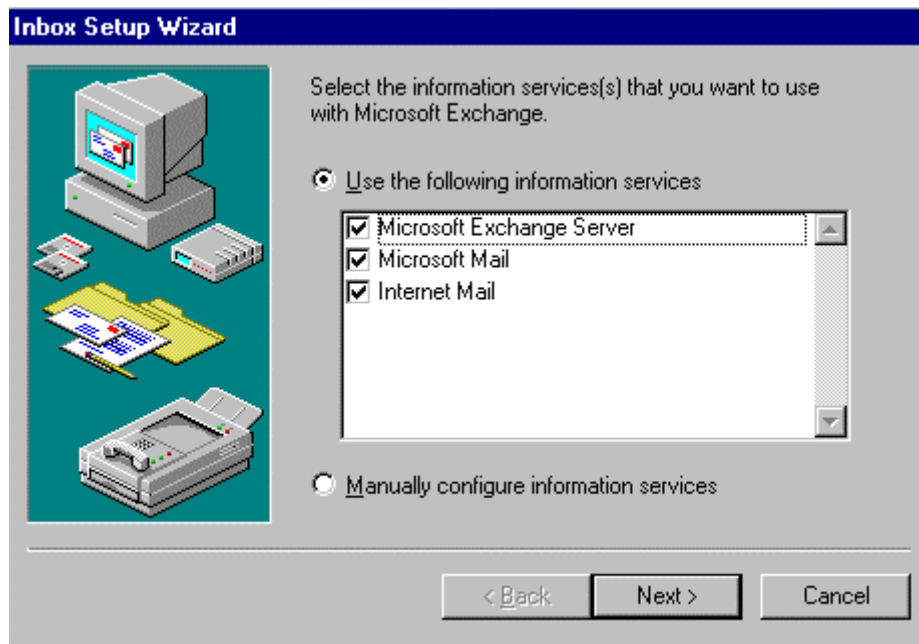
### Located:

1. **Pull-down Menu: File - Send**
2. **Or Enter Command or Alias at the Command Line**

The command SENDFLX allows you to send the current drawing via MS Mail, MS Exchange Server or Internet Mail.

The command SENDALL allows you to send all active drawings via MS Mail, MS Exchange Server or Internet Mail.

In the dialog below indicate which service you wish to use and have previously configured. You will be prompted to setup a selected service if it has not been previously configured.



Send Drawing(s)...

**Commands:** SENDFLX, SENDALL

**Alias** SD, SDA

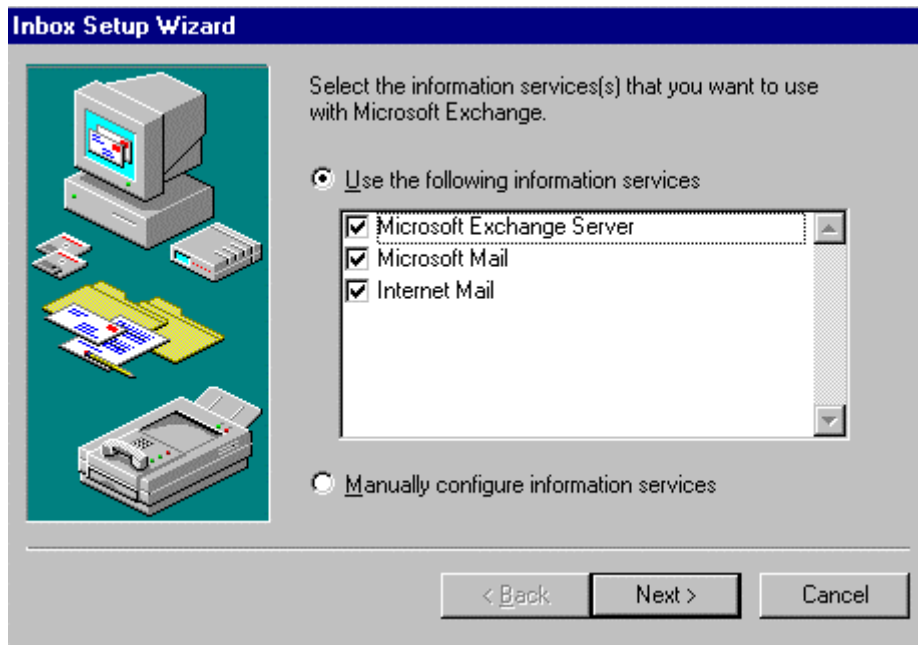
**Located:**

1. **Pull-down Menu:** File - Send
2. **Or Enter Command or Alias at the Command Line**

The command SENDFLX allows you to send the current drawing via MS Mail, MS Exchange Server or Internet Mail.

The command SENDALL allows you to send all active drawings via MS Mail, MS Exchange Server or Internet Mail.

In the dialog below indicate which service you wish to use and have previously configured. You will be prompted to setup a selected service if it has not been previously configured.



## Quick Color Setting...

**Command: SETCOLOR**

**Alias STC**

**Located:**

1. **Pull-down Menu: Tools – Set Color – Set Layer's Color**
2. **Or Enter Command or Alias Directly at the Command Line**

With the SETCOLOR command the user can specify a line color that over-rides the default layer color, or that returns color control to the layers color setting. This command resets the CECOLOR variable to the selected setting.

## Quick Layer Setting...

**Command: SETLAYER**

**Alias STL**

**Located:**

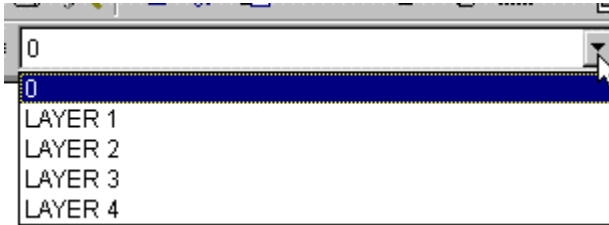
1. **Pull-down Menu: Tools – Set Layer**
2. **Properties Bar**
3. **Or Enter Command or Alias Directly at the Command Line**

This function allows the user to switch the active layer. It is not possible to create a new layer or to change any characteristics of the existing layer through the selection of this button. These changes must be made in the fashion described previously in Layer Control.

The letters in front of the layer names show the status of the characteristics visibility, access, and thawing/freezing. They have the following meaning:

- ON = On (Visibility)  
OFF = OFF (Visibility)
- U = Unlocked (Access)  
L = Locked (Access)  
T = Thawed (Thawing/Freezing)

When changing layers on the Property Bar, click on the down arrow to the right of the layer window, to produce a drop-down window showing all drawing layers.



You can then select the layer you want to be current from this list. After selection, the drop-down window closes and the layer selected should appear in the collapsed window. This indicates that it is now the current layer.

**Note:** A frozen layer cannot become the active layer, so that choice will not appear for the current layer. Highlight a layer in the list with the cursor and verify your choice by clicking on OK.

---

**See also:**

Layer Manager

Command SETLIB



**Command: SETLIB**

**Alias STB**

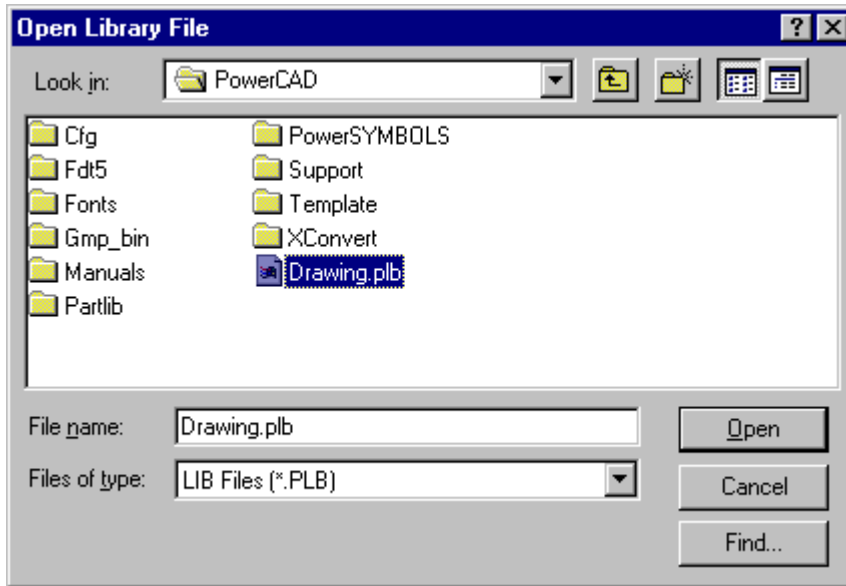
**Located:**

1. **Pull-down Menu: Tools – Resource Manager – Set Default Part Library**  
**Also - Parts – Set Current Library**
2. **Palette: Parts**
3. **Or Enter Command or Alias at the Command Line**

The LIBRARY command displays the Set Current Part Library / Symbol Library file dialog box. Select a part library (.plb) file.

The **SETLIB** command, on the other hand, sets the default library to be loaded whenever LIBRARY is executed. Simply select the .plb file you want as default in the following dialog:





Alternately, enter the PARTLIB command, which prompts you for the name of a part library at the command prompt. This command is meant for use with macros and FLISP functions. Enter the name of a .plb file, such as:

> partlib

File name: c:\AllenCad\partlib\partlibdrawing.plb

Once a part library has been chosen, the Library command continues to use that same library as default. Use the **SETLIB** (Part > Set Current Library) to change the part library.

## Change the Linetype...

**Command: SETLINETYPE**

**Alias SLT**

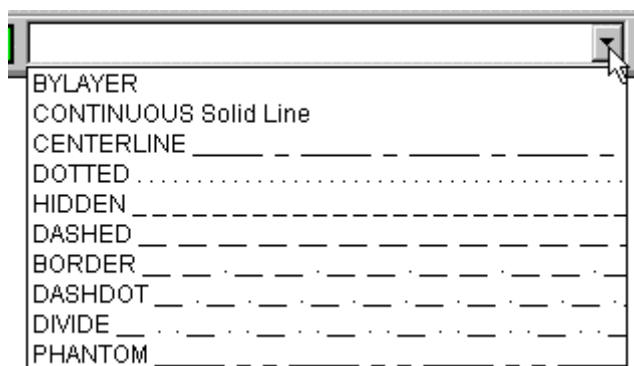
**Located:**

1. **Properties Bar**
2. **Pull-down Menu - Tools - Setlinetype - Set Explicit Linetype**
3. **Entered Directly at the Command Line**

For reasons of clarity, the user may want to draw different objects with different linetypes than the one marked BYLAYER. This command sets the default linetype to be used for subsequent line drawing, overriding bylayer settings. Using SETLINETYPE – Bylayer returns the program to layer-assigned linetype control.

Remember that a new linetype has to be loaded before it is available for use, and that linetype should be designated before drawing a new object.

To set a current layer (other than bylayer) on the Properties Bar, click on the down arrow to the right of the linetype window to see a drop-down menu that lists all loaded linetypes.



Click on the linetype you want to make it current. When you wish to return to layer control over linetype, simply repeat the procedure and select "bylayer" from the list.

---

**See also:**

Load Linetypes...

## Settings



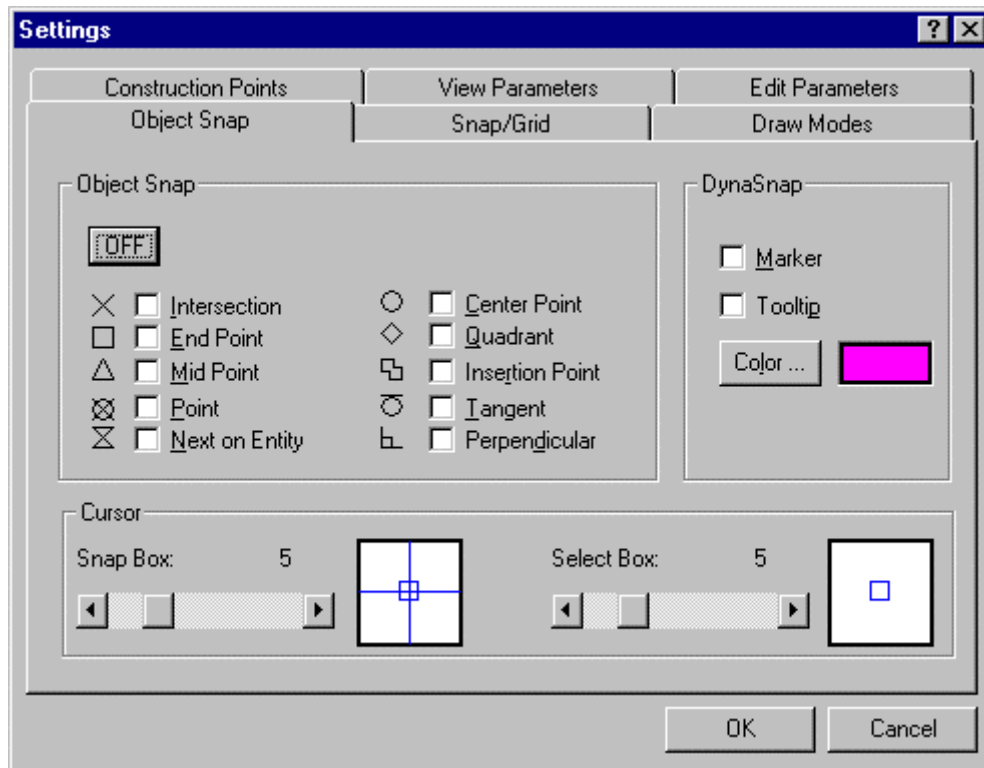
**Command: SETTINGS**

**Alias ST**

**Located:**

1. **Pull-down Menu: Tools – Settings**
2. **Palette: Snap**
3. **Or Enter Command or Alias Directly at the Command Line**

The command SETTINGS is the main command to set the parameters and modes of the drawing environment. Some of the tabs and settings shown below can be accessed directly, by other commands, in addition. These include: Object Snap – Command PRECPAR; Draw Modes – Command DRAWMODE; Edit Parameters – Command EDITPAR and View Parameters – Command VIEWPAR. The command calls a dialog containing a number of tabs that each have specific windows for entries that control the drawing environment.



For a more detailed explanation of each of these tabs and their entries, click on the links below:

[OBJECT SNAP](#)  
[SNAP/GRID](#)  
[CONSTRUCTION POINTS](#)  
[DRAW MODES](#)  
[EDIT PARAMETERS](#)  
[VIEW PARAMETERS](#)

## Command SETUP

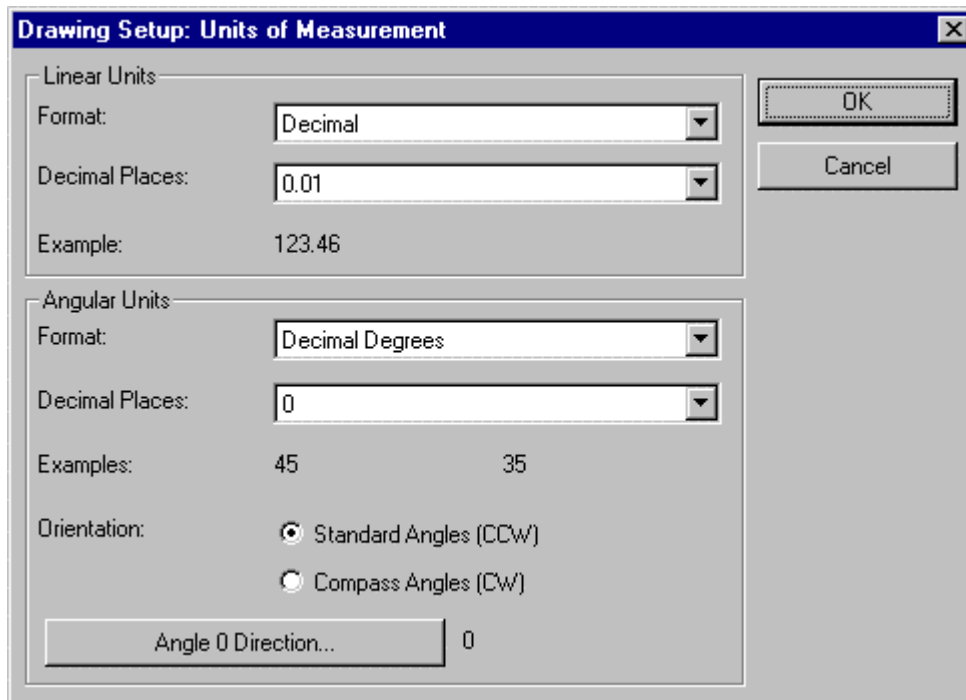
**Command:** SETUP

**Alias** US

**Located:**

1. **Pull-down Menu:** Tools – Drawing Setup Parameters
2. **Or Enter Command or Alias at the Command Line**

AllenCad allows you to set the format and display accuracy of numbers and angles. The settings are saved with each drawing, so they can be set differently in each drawing. The Setup command displays the Drawing Setup dialog box:



The command UNITSETUP is identical to this command and has been provided for the convenience of users familiar with AutoCAD commands.

### Linear Units

Linear units can be given a display accuracy ranging from zero to eight decimal places. In the case of fractional units, the display accuracy ranges from 0 (no fractions) to 1/256. You can select five kinds of units:

- Decimal units (metric) - 12.3456.
- Fractional units (unitless fractions) - 12 3/4
- Architectural Units (feet and fractional inches) - 12'-3/4" (See note below)
- Engineering units (feet and decimal inches) - 12-3.456'
- Scientific units (exponential notation) - 1.234E+02

### Angular Units

The Angular Units can be given a display accuracy ranging from zero to eight decimal places. AllenCad recognized these angular measurement systems:

- Decimal degrees - 12.3456
- Degrees - minutes - seconds - 12d34'56.
- Grads (400 grads in a circle) - 12.34g
- Radians - 12.34r
- Surveyors units (Quadrant Bearings)- N12d34'56 E

**Note:** Please refer to "Using Architectural Units" for details on the setup and use of architectural units.

### Orientation

In most drawings, angles are measured in the counterclockwise direction. This is the default in AllenCad and is called Standard Angles.

You can, however, set the orientation so that positive angles are measured in the clockwise direction, called Compass or Azimuth angles.

### Angle Zero Direction

In most forms of drafting, zero degrees is located at the positive X-axis (East, or 3 o'clock). AllenCad allows you to change the location of zero degrees. For example, surveyors normally set zero degrees as up on the page (North, or 12 o'clock).

The display of the status bar is updated depending upon the settings of the angle zero direction and the

orientation.



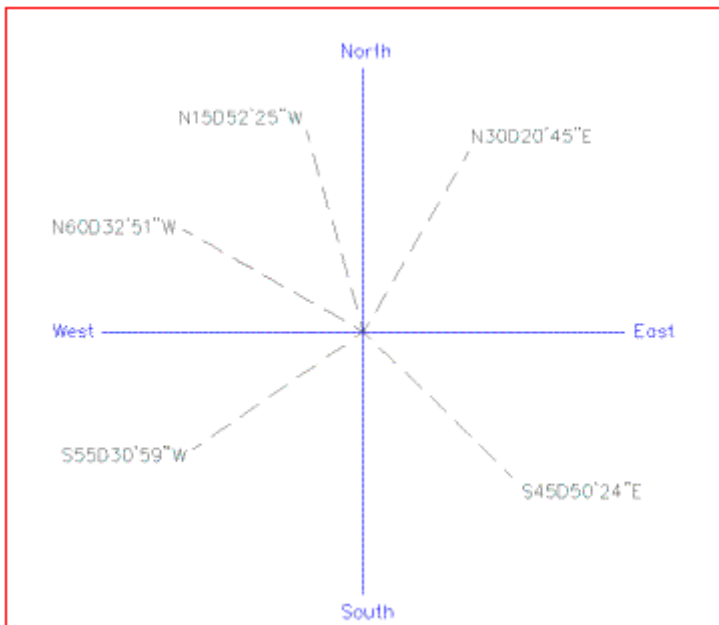
### Surveyor Units Entry

Because the surveyor units are so different from the other forms of angle entry, we have included a few examples:

Type of coordinate	Input	Example (Surveyor Units)
<b>Absolute Coordinates</b>		
Polar	distance<angle	7.5< N34D30'45"W
<b>Relative Coordinates</b>		
Polar	relative distance<angle	@7.5< N34D30'45"W

In surveyor's notation, the N stands for North. The 34D is 34 degrees from North. The 30' is an additional 30 minutes, and the 45" is an additional 45 seconds from North. The W indicates that the angle is towards the West.

There are 60 seconds in a minute, 60 minutes in a degree, and 360 degrees in a circle. Surveyors divide the circle into four parts, each with 90 degrees measured from the North or South poles in the direction of East or West.



In the figure above, we see the four points of a compass: North, South, East, and West. As examples of how the Surveyor's Units work, several lines have been drawn in various directions. We labeled these lines with the correct bearings. Note that all the letters used in the bearings are in capital letters, as required by AllenCad.

Note: When entering the N, S, E, W and D, be sure to enter them as capital letters -- lower-case letters

will not work.

## Command SETVAR

**Command: SETVAR**

**Alias SV**

**Located:**

### 1. Entered Directly at the Command Line

The SETVAR command is used to check and change current system variable settings. At any "Enter system variable name" prompt, you can enter the variable name to show current value and be offered the opportunity to change the value as needed.

The command sequence shown below, shows the input required to produce an alphabetical listing of all system variables (20 at a time). This listing is produced in the text window, that is accessible by pressing the F2 function key.

```
> SETVAR
Enter system variable name ? for wild card pattern:? [? Entered to produce a listing]
<*> [Return key pressed to list all variables]
List of global variables
NAME  VALUE      read only saved
ACTDB   0          X
AREA   0.00      X
CDATE   20001209.12394969 X
CIRCLERAD 0.00
CIRCLERES 48          X
CMDACTIVE 1          X
CMDECHO   1
DATE     2451888.52765046 X
DEFANGBASE 0          X
DEFANGDIR  0          X
DEFAUNITS  0          X
DEFAUPREC  0          X
DEFDIMZIN  0          X
DEFLUNITS  2          X
DEFLUPREC  2          X
DIASTAT   0          X
DISTANCE  0.00          X
ERRNO     0          X
EXPERT    0
FCTEMPLATE ""          X
```

Press any key to continue... [Return key pressed to indicate a continuation]

```
NAME  VALUE      read only saved
FCVERSION "5.00"      X
FILEDIA  1
HIGHLIGHT 1
HLDOTTED  0          X
LANGUAGE  2
LASTVAR   ""          X
OFFSETDIST 0.00
```

```

PALETTE1  "d:\Pcad2000\SUPPORT\.. X
PALETTE2  ""      X
PALETTE3  ""      X
PALETTE4  ""      X
PALETTE5  ""      X
PALETTE6  ""      X
PALETTE7  ""      X
PALETTE8  ""      X
PALETTE9  ""      X
PALETTE10 ""      X
PANSCALE  0.50      X
PDMENUNAME "D:\Pcad2000\SUPPORT\.. X
PERIMETER  0.00      X
PLATFORM  "Win32"    X

```

Press any key to continue... [etc., etc. to continue the listing to the end]

To make changes:

```

> SETVAR
Enter system variable name ? for wild card pattern:FILEDIA
New value for FILEDIA <1>:0 [Zero value entered]
>

```

In the above example, we have changed the current value of the FILEDIA variable from 1 to 0.

## Set View Direction (Command Line Version)

**Command: SETVIEWDIR**

**Alias STV**

**Located:**

1. **Palette: View (9 Options)**
2. **Or Enter Command or Alias at the Command Line**

It is possible for the user to look at a drawing from any point in space that he or she wishes to define.

The user must respond to the prompt for the definition of the point of view. The correct response will be the spatial coordinates of the viewpoint:

```

SETVIEWDIR
Point of view (X,Y,Z): 2,3,-5

```

Along with the ability to view the drawing from any vantage point, it is also possible to rotate the view in the X – Y plane.

The command is intended for use within palettes and menus to retrieve standard side views or isometric views.

---

**See also:**

3DVIEW

Saving/Retrieving Views

## Command SKETCH

### **Command: SKETCH**

**Alias SK**

#### **Located:**

**Entered Directly at the Command Line**

The SKETCH command allows you to generate straight-line segmented polylines that trace cursor movement while the command is active (while the left mouse button is depressed). When the command is issued, you are prompted to input a dimension for minimum segment length. This will determine how "smooth" the curved portions of your resulting line will appear onscreen.

See also: FREEHAND

## Creating and Viewing Slides

### **Commands: MSLIDE**

#### **MSLIDE is Located:**

1. **Enter Command or Alias at the Command Line**

### **VSLIDE (Alias VS),**

#### **VSLIDE is Located:**

2. **Pull-down Menu: View – View Slide**
3. **Or Enter Command or Alias at the Command Line**

### **SLDOUT (Alias SO)**

#### **SLDOUT is Located:**

1. **Pull-down Menu: View – Take Snapshot (Slide)**
2. **Or Enter Command or Alias at the Command Line**

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.

You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

### **To create a slide file**

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.
3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

### **To view a slide file**



Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

## Command SOLID

### Command SOLID

**Functional in:** AllenCad CE Pro **YES**

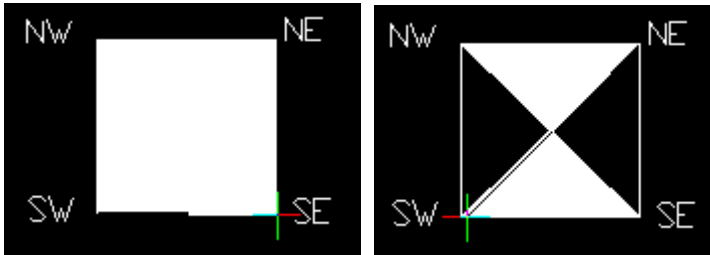
**AllenCad CE** **YES**

### Located:

1. **Draw Menu > Surfaces... > 2D Filled Faces**
2. **Enter Command or Alias Directly at the Command Line**

The SOLID command is used to "fill in" regularly-shaped boundaries with color. The command prompts for the entry of four points that represent the corners of a polygon that lie on the boundary vertices. Thereafter, the solid is extended (as required) with prompts for two additional points defining a new boundary with the previously entered third and fourth point.

The entry order is important, however and should follow the format NW, NE, SW, SE for the four points, rather than NW, SE, NE, SW, as the latter will give an "hourglass" fill, rather than a rectangular one. The difference is as illustrated below:



### Notes:

1. This command is generally used for simple boundaries with straight sides. For complex or curved boundaries, you may prefer to use a solid Hatch (command HATCH with Solid Pattern).
2. Color of the fill will be the currently set color, or else the bylayer color of the current layer.
3. Osnaps should be used for greater accuracy

## Command SPLINE

### Command: SPLINE

**Alias SP**

### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Polyline to B-Spline and Decurve B-Spline**
2. **Or Enter Command or Alias at the Command Line**

The SPLINE command operates on Chains, Polylines and LWPolylines (2-D only), and converts the entity to a B-Spline.



#### Changes

#### To

**Note:** The same command can also be issued to reverse the process. On the pull-down menu, however, it has been made as a separate selection, "Edit Polyline – **De-curve** B-Spline". Issuing the command at the Command prompt is also supported. In the following command sequence, note that the "smoothing factor" is controlled by the "number of segments for approximation" entry. The higher the number input here, the greater the degree of smoothing performed.

```
> SPLINE
Loading d:\Pcad2000\SUPPORT\flx_spln.lsp... OK.
Select polyline to alter to B-spline:
Number of segments for approximation (2 to 1024) <8>: 100
```

#### Stretch



**Command: STRETCH**

**Alias SR**

#### Located:

1. **Pull-down Menu: Modify - Stretch**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The **Stretch** command is useful for the enlargement or reduction of objects in only one direction (in contrast to scaling). Subsequent stretching of the same object in a second direction is not supported. Standard geometric entities (circles, rings, etc.) are not distorted, only shifted in position relative to the other objects being stretched.

Selecting entities for stretching must be done very carefully to ensure the desired effects, as **selection is by crossing and crossing polygon only**. During stretching, the method of selection has an influence on the question of which entities are moved and which entities are stretched.

- All entities included totally in the selecting windows will be moved;
- All entities crossing the selecting window will be stretched (lengthened); and
- All other drawing entities will retain their position and shape.

After having completed object selection, choose an option for stretching from the Option Bar..

#### Options:

##### Basepoint

The option **Basepoint** is the standard option for moving entities. It is used if no other option has been selected.

The option is useful for moving selected drawing entities to a particular position and for "stretching", or moving, entities on a border, without changing the relationship to the border. The target position may be a point defined by coordinates of a position determined by the geometry of the object.

Next, determine the end point of the stretch. The base point for the stretch will be moved to the end point.

### **Displacement**

The option **Displacement** allows stretching selected entities by a straight line (a vector). After selecting the objects, select the option from the Option Bar. Enter the value for the move, a straight line, in the form X,Y, (Z).

## Text Style Definition (Command Line Version)

**Commands: QFONT or STYLE**

**Alias QF**

### **Located:**

#### **1. Entered Directly at the Command Line**

The command QFONT, or STYLE, allows the user to load text styles from the specified font file and customize it to individual requirement.

These commands are command line variants of the FONT command.

```
> STYLE
Style name (or ?) <STANDARD>:
New text style definition.
Font file <TXT.FSH>:
Text height <0.00>:
Text style width factor <1.00>:
Text style obliquing angle <0>:
Backward alignment (Yes or No) <N>:
Upside-down alignment (Yes or No) <N>:
DIM set as current text style.
```

---

### **See also:**

Command FONT

QFONT and STYLE are the command line versions of the FONT command.

## **"T"**

## Tablet Menu Configuration

**Command: TABCONFIG**

**Alias TC**

### **Located:**

#### **1. Or Enter Command or Alias at the Command Line**

To utilize a tablet menu as input device, it is required to determine the exact location of the individual tablet menu section (areas). The command TABCONFIG allows you to specify these settings.

## How to configure a Tablet Menu

A dialog box assists the user to configure a tablet menu. Follow this procedure to configure your tablet menu:

1. Select by clicking the section to be configured. Note: In addition to the pointing area you can determine up to 8 tablet menu sections
2. Click to the button *Start Configuration*.
3. With the pointing device of the digitizer identify the upper left corner of the specified tablet menu section. When the point has been picked, the coordinates are shown in the corresponding edit box.
4. In the same way specify the coordinates of the lower right corner of the menu section.
5. If necessary, repeat step 1 and 4 for other tablet menu sections.
6. Terminate the configuration of the tablet menu areas with OK.

---

### See also:

Load Tablet Menu (TABLET)

Replace Tablet Menu Section (TABSECTION)

## Command TABLES



### Command TABLES

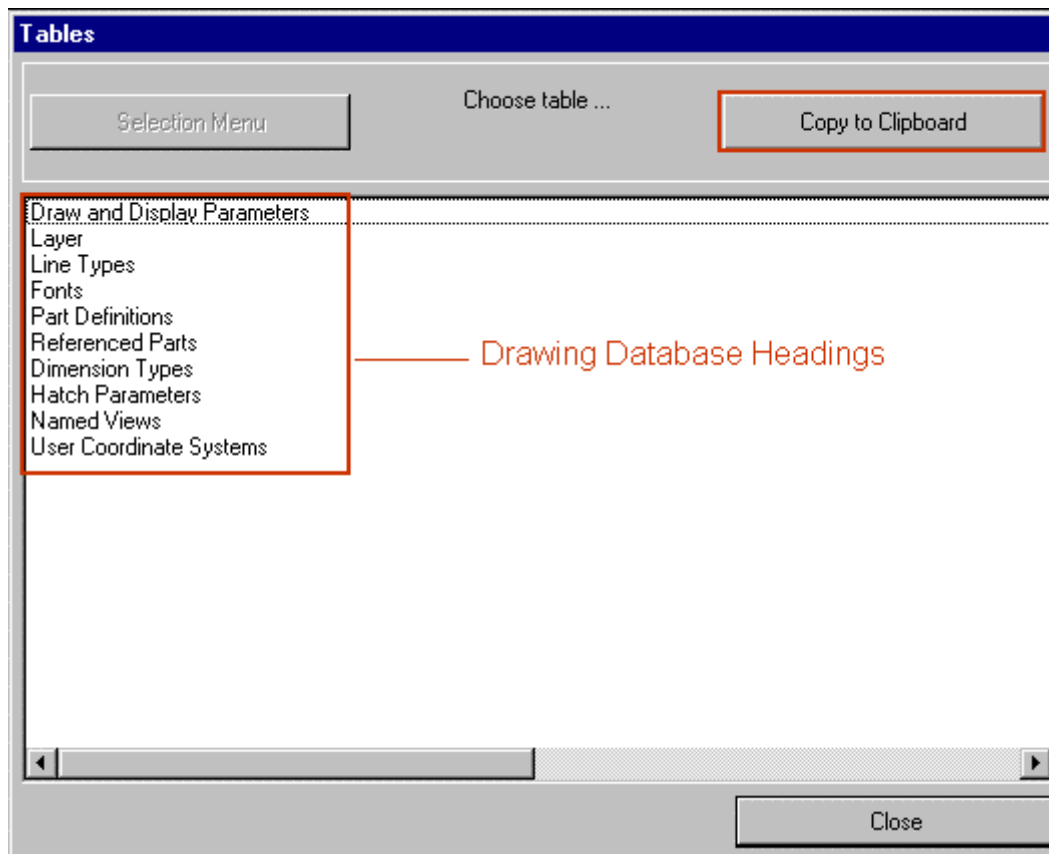
#### Alias TBL

##### Located:

1. **Pull-down Menu: Edit – Drawing Database Tables – Tables Info**
2. **Left SideBar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

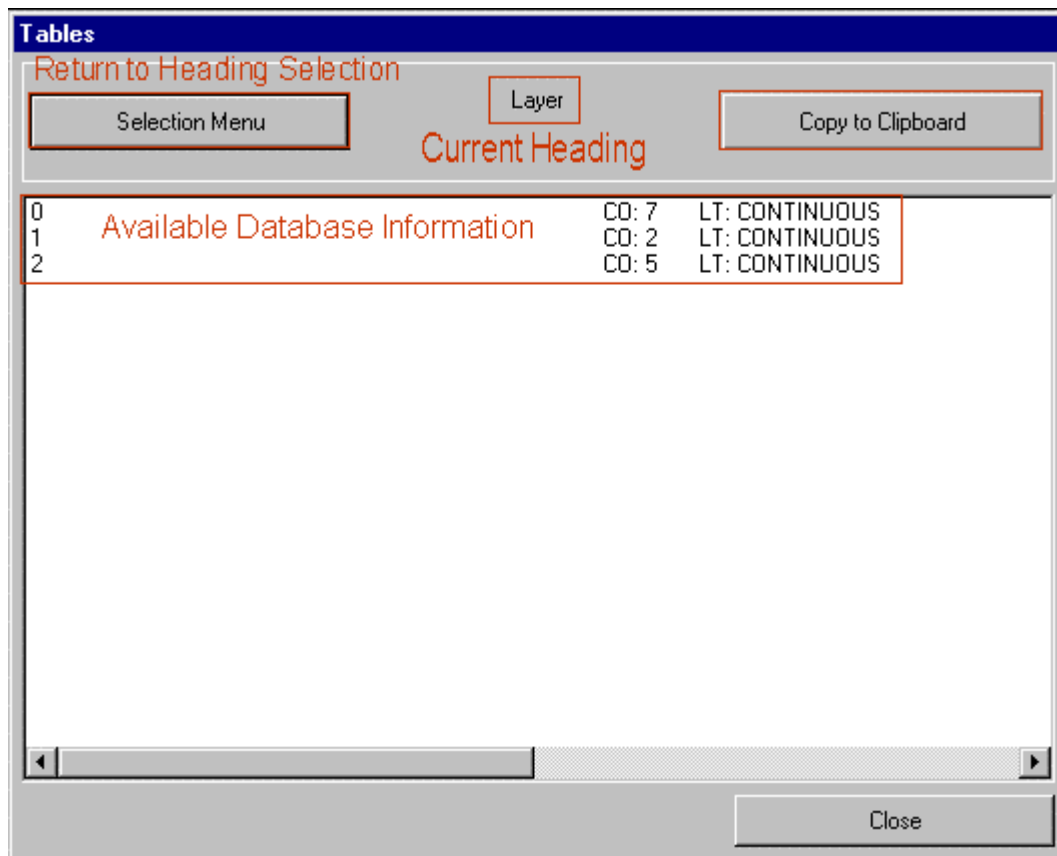
Through the use of the TABLES and associated commands, AllenCad offers the user simple control and auditing capabilities over the entire drawing database. The TABLES command is located on the Edit pull-down menu as the Drawing Database Tables – Tables Info selection. It can also be entered at the command line if preferred.

Upon execution, the following dialog is opened:



Here, information on drawing parameters used can be listed out and copied to the clipboard, to be used in reports or even imported into the drawing, itself.

If we select the option LAYER, for example, we see a listing of all layers and layer parameters in the current drawing:



This listing is copied to the clipboard by pressing the Copy To Clipboard button and you can return to the original listing of drawing and display parameters by clicking on the Selection Menu button.

## Load Tablet Menu...

**Command:** TABLET

**Alias** TA

### Located:

#### 1. Entered Directly at the Command Line

The program supports the use of a digitizer tablet as input medium and cursor control.

Coordinates for drawing points can be entered as well as commands and functions by the proper symbols.

To control the tablet it is necessary to load a Tablet Menu file. This load is executed automatically during the program start if the tablet configuration has been selected as a portion of the program configuration.

The command TABLET enables the user to change the tablet menu file in the same fashion as the exchange of a Menu file.

### To load a Tablet Menu, follow this procedure:

1. Select, from the cascading menu *Menu / Customization* the item *Tablet menu..* The user can also select the *Menu* symbol in the function bar or input the command TABLET; then
2. Select the desired tablet menu file from the dialog box. Confirm the selection by OK.

A tablet can also be loaded by selecting the symbol for program customization. In this case the user must

select the file format **.mnt**. The remainder of the selection is made according to the previously described procedure.

---

**See also:**

Tablet Menu Setup (TABCONFIG)  
Replace Tablet Menu Section (TABSECTION)

## Replace Tablet Menu Section...

**Command:** TABSECTION

**Alias** SEC

**Located:**

**1. Enter Command or Alias at the Command Line**

*Replace Tablet Menu Section* enables the user to load an individual section of the tablet menu, changing the commands and functions of the complete menu.

Enter for this the command TABSECTION or call the function *Replace Tablet Menu Section* from *Menu / Customization* under the *File* menu.

Determine that tablet menu section should be loaded. Identify the desired section by its section number.

Enter the name of the tablet menu file that contains the defined area.

```
> TABSECTION
Replace Tablet Menu Section
ID (S1...S8) or section name: S1
File name of the new tablet menu description: tabmenu1
```

The identified area with the commands and functions from the entered file will be restored.

---

**See also:**

TABCONFIG  
Load Tablet Menu  
Command TABSERVICE



**Command** TABSERVICE

**Alias** TB

**Located:**

- 1. Pull-down Menu: Edit – Drawing Database Tables – Tables Maintenance**
- 2. Top Function Bar**
- 3. Palette Edit**
- 4. Or Enter Command or Alias at the Command Line**

The TABSERVICE command offers the user the opportunity to simply alter and maintain parameters used in the current drawings drawing database. The command is located on the Edit Pull-down menu as the "Drawing Database Tables – Tables Maintenance" selection. It can also be typed in at the command line, if preferred.

When the command is issued, you will be presented with the dialog pictured below. Here you can select the parameter that you wish to modify:



If you wish to modify Layers, for example, you can eliminate all empty (containing no entities) or unused table entries, by clicking on the Purge Table button. You can also rename or delete items, such as layers, by clicking on the other two buttons.



### Tabulated Surface (Extruded Surface)

**Command:** TABSURF

**Alias** TF

#### Located:

1. **Pull-down Menu: Draw – 3D – Tabulated Surface**
2. **Or Enter Command or Alias Directly at the Command Line**



**From:**

**To:**



The TABSURF command creates a tabular surface using a path curve and a direction vector.

You can extrude lines, arcs, circles, 2D or 3D polylines. The entity provided as path curve defines the surface of the mesh. The direction vector (extrusion path) must be determined by picking a line or an open polyline.

The system variable SURFTAB1 defines the number of *M*-direction surfaces on the extruded entity. When extruding a line, arc, or circle, the number of *M*-direction surfaces is evenly divided determined by the value of the SURFTAB1 system variable. When extruding a polyline, extrusion lines are drawn at the ends of the line segments, and the arc segments are divided evenly into the number of surfaces determined by the value of the SURFTAB1 system variable.

If the polyline has been curved (fitted) or splined, the number of *M*-direction surfaces is evenly divided using the SURFTAB1 variable. Like with the command RULESURF, you only need to set SURFTAB1 system variable.



The resulting surface is a polygon mesh (stored in the drawing database as a complex polyline entity).

### To create a tabulated surface

After you select the entity for the path curve, you specify the direction vector:

> **TABSURF**

Select path curve:

Select direction vector (line or open polyline):

The direction and length of the line determine the direction and length of the extrusion. Only the first and last points of a polyline are used when determining the extrusion path. The point at that you specify the entity determines the extrusion direction.

Please check in the Learning Center for a fully documented example of this command's use.

### Correct Text



**Command: TCORRECT**

**Alias TR**

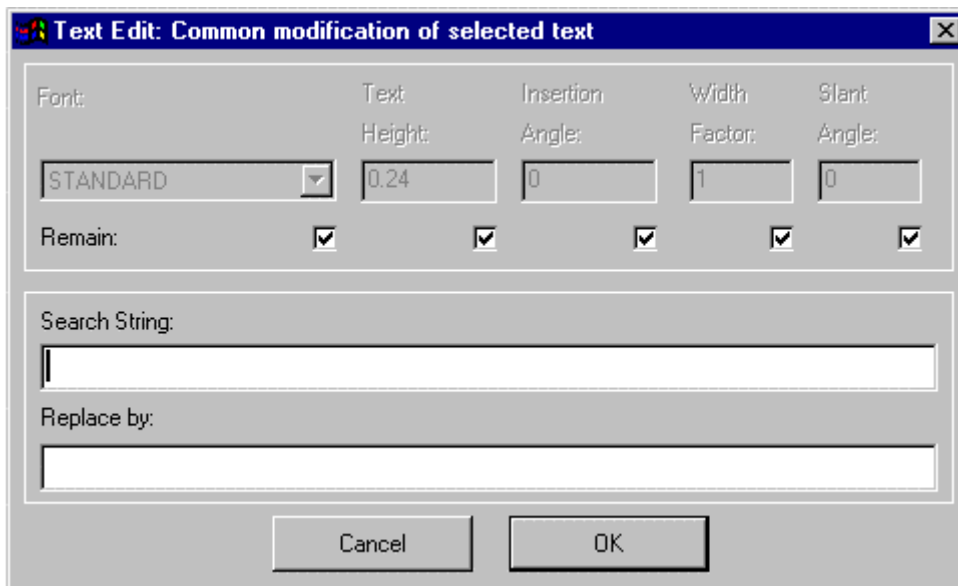
#### Located:

1. **Pull-down Menu: Detail – Correct Text**
2. **Palette: Detail**
3. **Or Enter Command or Alias at the Command Line**

The function **Correct Text...** allows the user to edit multiple text entities and attribute definitions. It is possible to correct texts or text attributes as well as to change the font or the point size.

Call the function Correct Text by keyboarding TCORRECT or select Correct Text... from the pull-down menu *Detail*.

First, you are requested to select the text lines or attribute definitions. You may use the usual object select functions. Finish the object selection by ENTER.



As can be seen from the dialog above, this command will replace all instances of an existing text string in the selection set, with a replacement string. Alternately, or in addition, you can change existing text

parameters in the selection set by un-checking the appropriate "Remain" box and entering new values.

---

**See also:**

TEXTEDIT

Template Drawings

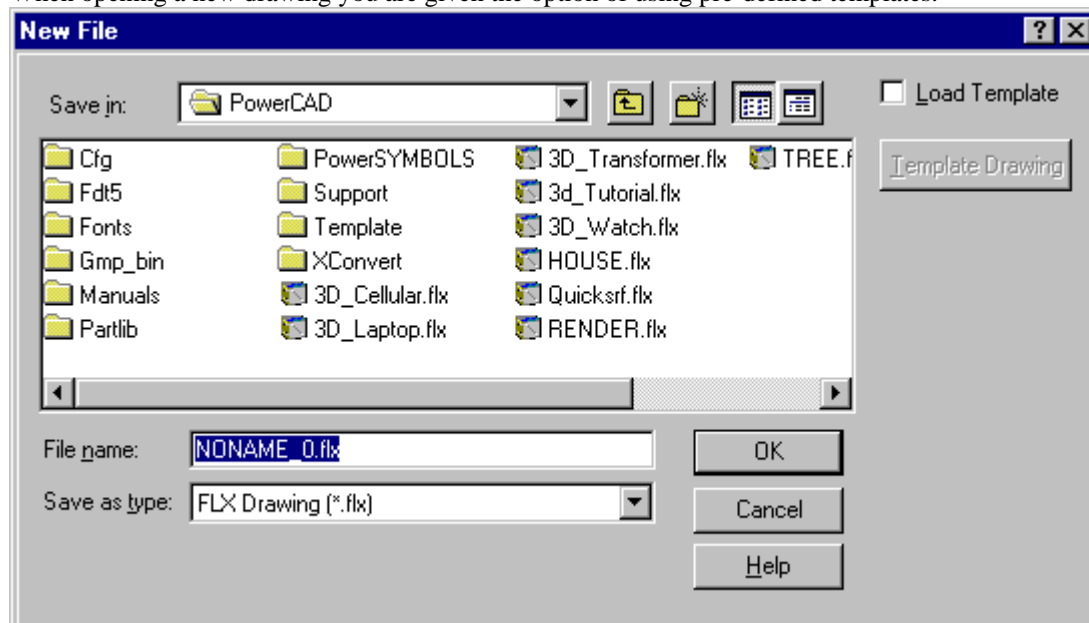
**Command TEMPLATE**

**Alias TP**

**Located:**

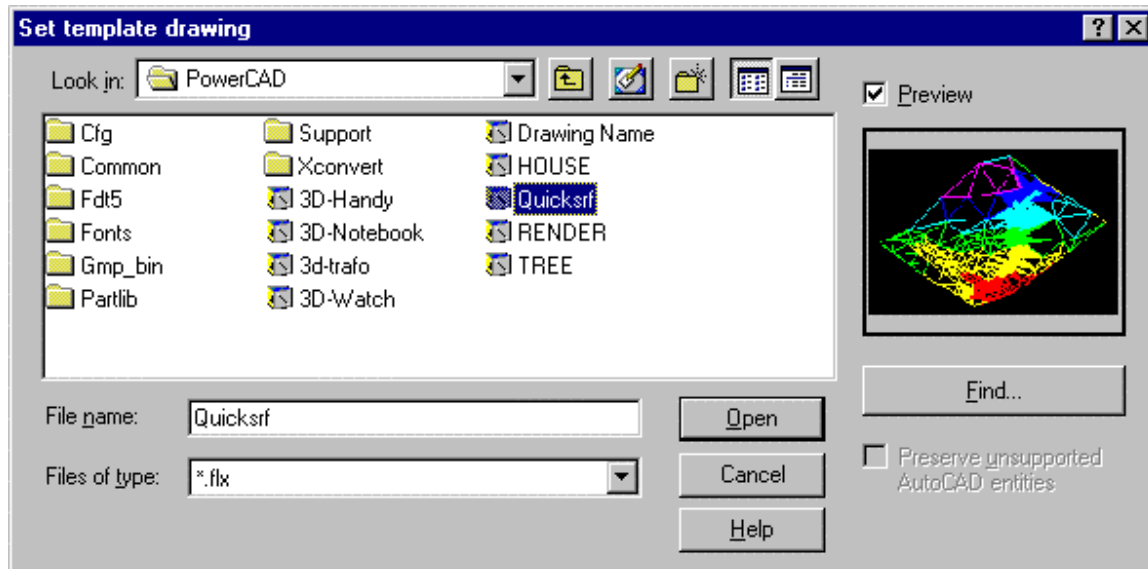
1. **Pull-down Menu: Tools – Resource Manager – Set Template Drawing**
2. **Or Entered Directly at the Command Line**

When opening a new drawing you are given the option of using pre-defined templates.



This is the standard file OPEN dialog, where you name your new drawing or choose to over-write an existing drawing with a new blank file. In the top right corner, you see the option to Load Template. If selected, this will load the default template drawing into your new file. If you wish to load another drawing as template, (with Load Template selected) select the Template Drawing button and browse to select another drawing file as template.

To set a new default template drawing, however, you must use the TEMPLATE command from within the program. All subsequent new drawing files will have the option of using this new template that you have just made default.



When you issue the **TEMPLATE** command, you will be presented with the dialog shown above. You simply navigate to the drawing you want as template, select it and press the **Open** button. That drawing will then be the new default template drawing until changed, using this same procedure.

#### Notes On Templates:

1. Templates are a form of specimen drawing that either act as a basis for your finished work, or set up the required drawing environment.
2. The main advantage of using templates is that all elements and settings of an already existing drawing can be adopted and as such do not need to be re-entered.

This is particularly useful for parameters such as drawing boundaries, frames, guidelines, measurement settings, views etc. In order to facilitate this, a number of local variables containing stored settings are loaded along with the template. These settings are then used to display the drawing, drawing elements and objects.

For a listing of commands that can be used in template drawing setup, please refer to Drawing Setup. Alternatively, if you have a drawing file that has all your standard setup elements in it, you can simply delete any drawing elements you do not wish to have in your template drawing and **SAVEAS** a new drawing file, to be used as the template.

Further information on the use of template drawings can be found in Chapter 4 of the Customization and Programmer's Guide.

Text...



#### Command: TEXT

#### Alias TX

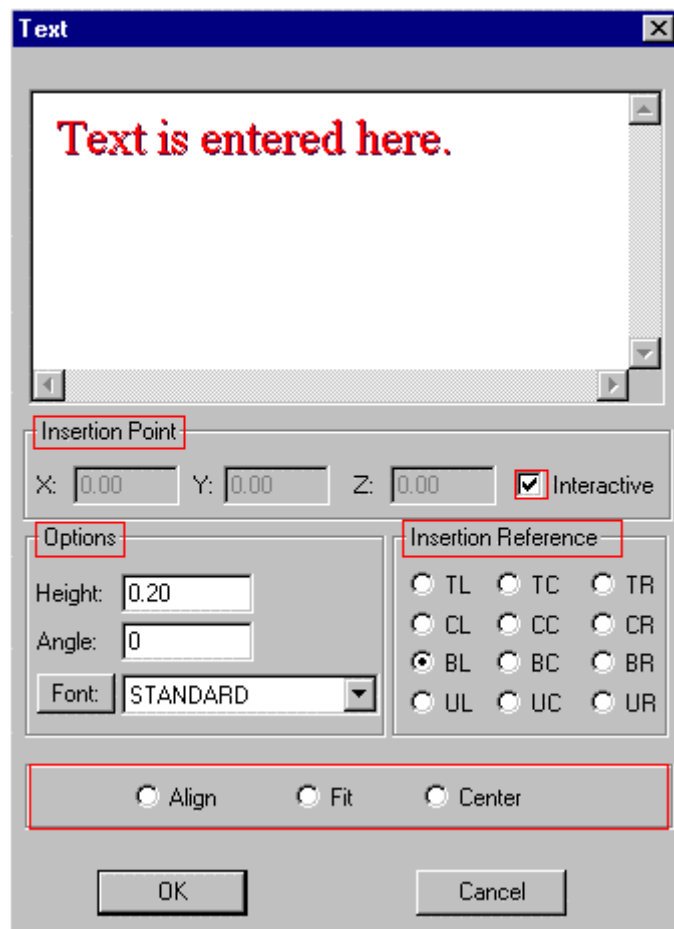
##### Located:

1. **Pull-down Menu: Detail – Text**
2. **Left SideBar**
3. **Palettes: Draw and Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

The **TEXT** functions are used for inserting text into drawings. They provide labeling, create legends,

descriptive notes, and other text elements in a drawing. Most of these commands and options are found under the menu *Detail*.

The command TEXT comes up with a dialog box that allows you to input text and to set several parameters for the text to be added to the drawing:



### Text input

Text is entered into the input field in the upper area of the dialog box, as indicated. The cursor will be positioned in this field upon the opening of the dialog box, so the user may start the input of the text immediately.

The text editor in this dialog box offers the basic functions for editing text input. Incorrectly entered text can be marked and erased just as with a word processor.

When the text entry is completed, the user should make any of the changes that are desirable and within the capabilities of this dialog box. The changes possible are discussed below.

When a line or block of text is complete, press ENTER.

**Note:** Text copied to the clipboard from any Windows application can be simply imported into AllenCad's TEXT command dialog window, either through picking on the window and using Windows' "Ctrl V" function, or by right-clicking on the input window and selecting the "Paste" option. The text that you import in this fashion will assume all the new text characteristics specified for the text entry

### Insertion point

The Dialog Box offers two methods to define the insertion point. The insertion point may be entered using (absolute) coordinates (boxes are available for input of the X-,Y- and Z-coordinates). The second option is to define the insertion point interactively. The option **Interactive** (shown highlighted) must be checked to activate this option. Unchecking interactive will enable absolute coordinate entry.

### Options:

The following options aid the user in determining the height, the rotation angle, and the font (typeface) of the text object to be inserted.

#### Height

Define the size of the text in the *Height* edit box. Enter the desired text height in drawing units.

#### Angle

Define the insertion angle of the text object by this input field. Any rotation is possible, and the text can also be aligned to vertical or sloping objects. Parameter value for this is a rotation angle of 0 degrees (the text object is inserted horizontally). For a counter-clockwise rotation, enter a positive value; and for a clockwise rotation, enter a negative value between 0 and 360.

#### Font

A window allows the user to define the font. The list of available fonts is displayed in this window. Select the desired font from this list.

### Alignment:

The alignment of the text object(s) can be specified by the following options:

#### Insertion Reference

Defining an **Insertion Reference** point places the text in a relative position to the selected insertion point. For this, a reference point on the text object is defined, as described below.

If the reference point, for instance, is situated at the lower left corner of the text object the text will be placed such that the insertion point is at the lower left corner of the text, with text running to the right of the insertion point. If bottom right is selected, the text will lie to the left of the insertion point.

In total there are twelve reference points available for the user's choice. They are represented by the initials of the locating corner (or center) of the text block. UL, for instance, represents the Upper Left corner of the block. Select the position for the reference point by selecting the appropriate radio button.

#### Align

Align allows text objects to be rotated and scaled during insertion. The rotation angle and the scaling factor are defined by indicating two points on the drawing. These are the procedures for using Align:

- Activate the option by clicking the box *Align* at the lower left side of the Dialog Box *Text*. The options insertion point, text height, angle and position will not be available any longer. The parameters required for alignment, result from the definition of two points during the insertion of the text object. The reference point of the text object is positioned left justified, lower corner;
- Confirm the use of the option Align by clicking OK;
- Enter the first point for insertion of the text object. This point is interpreted by the program as both the insertion point and the first point for the rotation angle and scaling factor;
- Enter the second point that completes the definition of the rotation angle and the scaling factor, and sets the text height.

#### Fit

Fit enables the simultaneous rotation and expansion of a text object during the insertion. The text object is not scaled that is, the text height remains unchanged, while the string is expanded or compressed to fit. The procedure is:

- Activate the Fit option in the dialog box;
- After inputting the text string(s) confirm the settings by clicking OK;
- The prompt will be for the first point - it may be specified or "picked" - do this; then
- The prompt will be for the second point - specify or "pick" this point.

The text will be fit between the two points, rotated so that it follows the line between the points.

### Center

Center enables the user to insert a text object with both vertical and horizontal alignment centered. Text height and rotation angle may be chosen.

For onscreen display of text characters, see TEXTMINDISP.

## Command TEXTSCR

Alias TN

### Located:

1. **Enter Command or Command Alias Directly at the Command Line**

This command is provided primarily for use by programmers. It explicitly calls the program's text screen into view. Users will normally use the ``tscreen` (toggle screen display) or more simply press the F2 key to toggle between text and graphics screens.

## Modify Text...



Command: TEXTEDIT

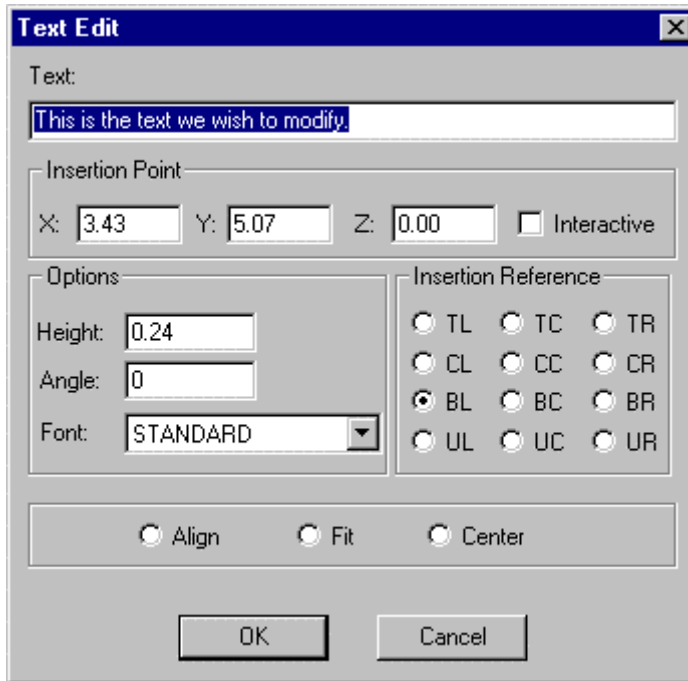
Alias ED

### Located:

1. **Pull-down Menu: Detail – Modify Text**
2. **Left SideBar**
3. **Palette: Detail**
4. **Or Enter Command or Alias Directly at the Command Line**

You can modify an existing text line in the drawing by selecting **Modify Text...** from the menu *Detail*.

Select a text entity in the drawing. The dialog box *Text Edit* is displayed that is similar to the dialog to create text (with exemption that only one line can be edited).



Simply edit the existing text in the window meet the requirements. The dialog box also presents the parameters attached to the selected text. These parameters can be modified as required, for the replacement text.

If the check box labeled *Interactive* is marked, you will be prompted to relocate the text when exiting the dialog. Otherwise you can change the insertion point coordinates in the dialog box, if desired.

> TEXTEDIT

Select a text:

<Dialog box appears to edit the selected text line>

Insertion point:

---

#### See also:

TEXT  
RTEXT  
FONT

#### THICKNESS

#### System Variable - Thickness

Alias TH

#### Located:

1. **Pull-down Menu: Tools – Entity Extrusion - Set Thickness or No Thickness**
2. **Or Enter Command or Alias at the Command Line**

Sets the value for the default thickness (extrusion). Specifying the value 0.00 means that no thickness is applied.

#### Properties

<i>Type</i>	Real
<i>Range</i>	Any positive or negative number
<i>Default</i>	0.00
<i>Status</i>	Read / Write

*Saved*            In the drawing file

**See also**

System variable ELEVATION

**Toggle Grid Display Mode**



**Command: TGRID**

**Alias TG**

**Located:**

1. **Pull-down Menu: Tools – Set Drawing Modes – Grid Reference Mode**
2. **Properties Bar**
3. **F-7 Function Key (default)**
4. **Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Grid Display ON / OFF. The grid can be used as a visual reference to points that are established parallel to the axes and a set distance apart. IF Snap is also switched on, these grid points can be quickly and precisely selected onscreen, while working on your drawing.

See command PRECPAR for grid setup

**Turn Coordinate Symbol on/off**

**Command: 'TICON**

**Alias TI**

**Located:**

1. **Entered Directly at the Command Line**

The transparent command TICON toggles the Coordinate Icon on or off.

**Toggle between Model Mode and Layout Mode**



**Command: TLMODE**

**Alias TM**

**Located:**

1. **Pull-down Menu: File – Layout / Model Mode**
2. **Left SideBar**
3. **Or Enter Command or Alias at the Command Line**

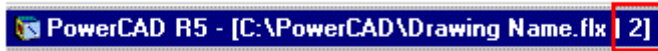
To switch from model mode to layout mode and vice versa, type in the command TLMODE (Toggle Layout Mode).

The tiled drawing windows of the current drawing in Model Mode, that display the model in different views, are replaced by a single window. The title bar of this window shows the caption **Layout:** and the



drawing name. The model mode references to drawing window disappear.

#### Model Mode - Drawing Window 3



Window Number (0,1,2 or 3)

#### Layout Mode

No Window Identifier



#### Notes:

When you switch from model space to paper space the first time, the Layout window is an empty window that represents a blank drawing sheet. This print space is not blank, however, if your template drawing already contains paper space objects (in this case paper space usually contains a company title block). In layout mode, all viewing of your drawing (which is in model mode) is through viewports that "cut through" the layout mode drawing surface to permit viewing the drawing in model mode that lies "behind it".

The first steps when entering layout mode are usually to perform two types of operations:

1. Determination of the sheet size for printing/plotting particularly by insertion of a title block;
2. Creation of viewports that display individual view onto the model.

You can draw in paper space as used to in model space. All the draw and edit commands are available. Usually, you will want to restrict your layout mode drawing exercise to the creation of sheet-specific notes, dimensions and title block information.

As paper space is designated to lay out drawing output, a number of 3D commands are disabled in layout mode: 3DVIEW, PLANVIEW, RENDER, HIDE, Fhide, LIGHT, LIGHTEDIT, SETVIEWDIR, WOPEN, and QWOPEN. As described in the following sections, layout mode allows you to arrange individual 3D views of the model by creating and modifying viewports.

---

#### See also:

Layout Mode

VIEWPORT

Toggle Orthogonal Mode



**Command: TORTH0**

**Alias TO**

#### Located:

1. **Pull-down Menu: Tools – Set Drawing Modes – Orthogonal Mode**
2. **Properties Bar**
3. **F-8 Function Key (by default)**
4. **Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Orthogonal Mode ON / OFF. When Ortho mode is on, lines that you draw will be forced parallel to the axes (IE straight up and down or sideways).

## Trapezoid

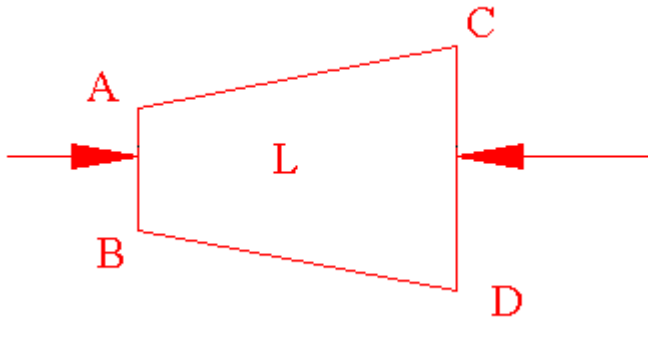


**Command: TRAPEZOID**

**Alias TZ**

### Located:

1. **Pull-down Menu: Draw - Trapezoid**
2. **Palette: Draw**
3. **Or Enter Command or Alias at the Command Line**



With the command TRAPEZOID it is possible to draw a single conical element (trapezoid) on the active layer.

Start this function by choosing the like-named option in the menu *Draw*, its palette symbol, or through entering the command TRAPEZOID.

After entering the point and insertion angle, the cone is defined by its length and the diameter of its two ends. To do that, determine the needed points successively through pointing or entering the values or coordinates manually.

The cone can also be drawn using a **Ratio** of tapering. To do that, press RETURN after the question of the diameter of the second side and enter a tapering ratio at the asking.

Command input sequence:

> TRAPEZOID

Insertion point: **Point A**

Insertion angle <0>: **Angle L will be oriented to**

Diameter of first side: **Dimension A-B**

Length: **Dimension L**

Diameter of second side: **Dimension C-D**

## Trim



**Command: TRIM**

## Alias T

### Located:

1. **Pull-down Menu: Modify - Trim**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

The TRIM command cuts off sections of linear objects, using another linear object as the "cutter".

The first prompt for this command will be a request to identify the entities to make up the "cutting edge". These entities will be the ones that will result in the "trimmed" intersection of lines. Pick these entities with the cursor.

The next prompt will ask you to select the entity to trim. When you pick an object, that portion of the object on the "selected" side of the cutting object, is trimmed off and deleted.

Note: when trimming multiple objects at once, using the same "cutter", consider using the object selection tool "fence" to trim all linear objects at once that cross the "cutter" along its selected length.

## Turn Command Line Window On/Off



**Command: 'TSCREEN**

## Alias TN

### Located:

1. **Pull-down Menu: Help – LISP/History Window**
2. **Top Function Bar**
3. **Palette: Zoom**
4. **F2 Key**
5. **Or Enter Command or Alias at the Command Line**

Toggles the Command Line Window on or off.

The Command Line, or the Text Window, serves as communication between the program and the user. In its default condition, the program uses a Command Line situated directly underneath the Menu Bar.

The button T on the right of the Text Window will allow the user to switch between this mode of display and a bigger Text Window, placed freely on the drawing field.

### See also:

Command Line and Text Window

## Turn Maximized Command Line Window On/Off

**Command: 'TSCREENFULL**

## Alias TSC

### Located:

1. **Entered Directly at the Command Line**

Toggles the maximized Command Line Window on or off.

Both, the commands TSCREEN and TSCREENFULL, allow you to display the command line area in an

independent window. TSCREEN shows the text window that covers only parts of the screen, whereas TSCREENFULL shows the text window covering the entire display.

## Toggle Snap Grid Mode



**Command: TSNAP**

**Alias TS**

### **Located:**

- 1. Pull-down Menu: Tools – Set Drawing Modes – Snap Grid Mode**
- 2. Properties Bar**
- 3. F-9 Function Key (default)**
- 4. Or Enter Command or Alias Directly at the Command Line**

This command simply toggles Snap Grid ON / OFF. With snap grid on, you can quickly and easily snap to points that are established at grid intersections. This mode is operational, with or without the visual representation of the grid, that is controlled by the TGRID command.

See command PRECPAR for grid setup.

See Also Snapmode

## Command TUTOR

**Command: TUTOR**

### **Located:**

- 1. Pull-down Menu: Help - Tutorial**
- 2. Or Enter Command or Alias at the Command Line**

This command is used to call up the program's "Learning Center", where you will find an introduction to AllenCad commands and concepts.

## "U"

## User Coordinate Systems



**Command: UCS**

**Alias UC**

### **Located:**

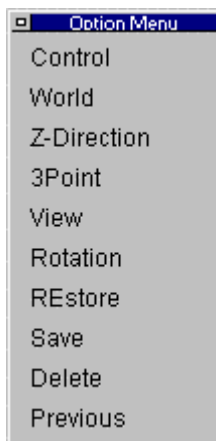
- 1. Pull-down Menu: Tools – World Coordinate System and  
Tools - User Coordinate System – UCS Manager (UCS\_C) and**

### Tools - User Coordinate System – UCS Origin (UCS) and

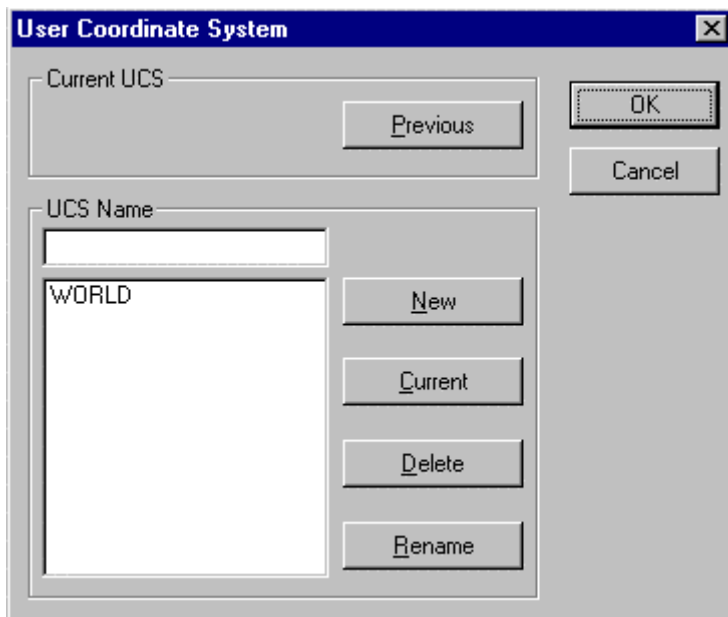
### Tools - User Coordinate System – Rotate Around Z Axis (UCS\_R-Z)

2. Left SideBar
3. Palette: View
4. Or Enter Command or Alias at the Command Line

When the UCS command is issued, an option menu is presented that lead directly to individual UCS functions.



The UCS Manager function is available by clicking on the Control option in the menu above. The UCS Manager controls the setup and definition of User Coordinate Systems.



It is possible to create as many user coordinate systems for one drawing as wanted. The user coordinate systems will be saved along with the drawing and can be used again when opening the drawing later on.

#### Active coordinate system

From all the user coordinate systems in one drawing, only one can be active at any given time.

Choosing a coordinate system to activate is done in a dialog box. Using the same dialog box it is possible to rename, create or delete already existing user coordinate systems.

### Create new user coordinate system

User coordinate systems are always created in two steps. First the parameters of the user coordinate system (place of the origin and direction of the coordinate axis) are determined, second follows the saving under a specific name. Thus it is possible to use the user coordinate system as an active coordinate system again, without having to determine the parameters again.

To create a user coordinate system, enter the command UCS. Through the message UCS **origin** <0,0,0> in the text window you are asked to enter the coordinates of the origin of the user coordinate system. The coordinates <0,0,0> are offered as a help. If you don't want to change the origin of the coordinate system then verify this choice through enter. Otherwise please enter new coordinates and verify this entry also with return.

After calling upon the command UCS, the following options for interactively determining the place of the origin of the user coordinate system and the direction of its axis are presented..

Control      World      Z-direction      3point      View      Rotation

The option **Control** calls up a dialog box that can be used to create or choose a new active coordinate system, or to rename or delete already existing user coordinate systems. How to use this dialog box will be shown later on in connection to the saving of the parameters of a user coordinate system.

The option **World** chooses the world coordinate system to be the active coordinate system. This allows you to go back to the world coordinate system quickly without having to open the dialog box of the command control.

### Z-Direction

The option z-direction changes the parameters of the coordinate system in the following two steps:

1. The origin is determined through entering the new coordinates in the format of x,y,z or through choosing a point .
2. The direction of the axis of the coordinate system can be determined through fixing the z-axis.

To do that, choose a point through that the positive z-axis, starting from the (perhaps newly defined) origin, should go. The x- and y-axis will not be turned and remain the same. The xy-plane will now be lifted to correspond orthogonally to the z-axis.

```
> UCS
UCS origin <0,0,0>: Z-axis
UCS origin <0,0,0>: Enter
Point on the positive z-axis: PI
```

### 3Point

The option 3point makes it easy to adjust a coordinate system to a drawn object in space. This is advantageous for constructing elements, where the drawing plane is neither parallel to the viewing plane, nor to the axis of the presently active coordinate system.

It is easiest to adjust a user coordinate system with the 3point option when the points that are to be defined are selected with the cursor using the objectcatch function. This way the coordinate system is easily adjusted to the specific working situation.

1. Choose the origin first through entering the coordinates in the format x,y,z or through pointing to it.
2. After that, choose a point, through coordinate entry or through pointing to it, through that the positive x-axis should run. This way the axis is unambiguously determined.
3. The place or direction of the other axis can be determined through the fixation of a point in the positive area of the xy-plane. This point does not necessarily have to lie on the y-axis.

For practical reasons you should locate the necessary points so that they are oriented at the plane of the drawing object at that the user coordinate system should be adjusted. For example, choose the corner points of this plane with the objectcatch option. The chosen plane then becomes the drawing plane. All elements drawn after that will be drawn according to this plane.

```
> UCS
UCS origin <0,0,0>: 3Point
UCS origin <0,0,0>: P1
Point on the positive x-axis: P2
Point on the positive xy-plane: P3
```

### **View**

This option adjusts the user coordinate system parallel to the active viewing plane, the screen. Through this the viewing plane and the drawing plane are brought into harmony.

To call up this option, click onto its operation field. No further points will be asked before adjusting the user coordinate system. The x-axis of the coordinate system now lies parallel to the actual viewing plane horizontally, the y-axis is oriented vertically and the z-axis is orthogonal to that.

### **Rotation**

Through the option rotate it is possible to rotate the active coordinate system around one axis. The angle of rotation and the direction of the rotation can be chosen randomly.

To call up the option rotation, please click onto its field among the options.

With one call-up it is possible to rotate the coordinate system around one axis only. If you want to rotate the coordinate system around two or three axes, please repeat the following steps for each axis.

1. First determine the axis around that the coordinate system should be rotated. Either enter its character or click onto its option field (X,Y,Z) that will appear after the activation of the rotate option.
2. Now determine the direction and angle of the rotation. Entering or pointing an angle does this.

The angle of the rotation can be determined either by entering it manually or by the pointing of two points. The orientation point for the angle of the rotation is always the positive x-axis of the active coordinate system.

Entering a positive value for the angle of rotation results in a rotation to the left, a negative value (for example -30) results in a rotation to the right.

```
> UCS
UCS origin <0,0,0>: Rotate
Rotate around x-, y-, or z-axis: Z
Rotation around Z axis <0>: 30
```

### **Controlling the UCS**

Of course it would not be very user-friendly and not very effective for the precision of your work, if you had to make the above adjustments every time. Therefore the program offers a comfortable possibility to save and organize pre-defined user coordinate systems. The parameters of the active coordinate system are gathered in a user coordinate system and saved along with the drawing. When saving, the user coordinate system gets a definite name. Once a user coordinate system is saved, it can always be called up again to be the active coordinate system. They can also be renamed or deleted.

The necessary steps to gather and organize user coordinate systems are executed through a dialog window. To open this window please choose the option user coordinate system-manager that is found when using the standard menu in the pull-down menu view.

Also it is possible to open a dialog window by entering the command UCS and choosing the option *Control*.

### **Create a new UCS**

The creation of a user coordinate system is done in two steps

1. Arrange the coordinate system with the previously described options z-direction, 3point, view and/or rotate.
2. Open the dialog window through the option command and arrange a new user coordinate system. Now the parameters of the active coordinate system are saved.

For the following directions it is essential that the first step has already been taken, e.g. the coordinate system has been adjusted.

The dialog box has an option that is called user coordinate system name. Enter the name of the new user coordinate system in this field. The name can fill as many as 32 spaces and may contain alphabetical characters, numbers, as well as the fill-in characters - (dash) and \_ (underline), but no free spaces. It is not differentiated between capital letters and small letters, all characters will be changed to capital letters.

After entering the name, click onto the button **New** and verify your entry. The name of the new user coordinate system appears in the window below the entry- and change window. All useable coordinate systems in this drawing are listed there.

Please remember that all changes made in the user coordinate system dialog box are registered only after clicking OK as you leave the box.

### **Renaming a user coordinate system**

It may be necessary to rename user coordinate systems in order to have a clear name or association

1. Choose the user coordinate system to be renamed out of the list in the choice window by clicking onto it with the cursor. The chosen coordinate system is accentuated through color. At the same time the name of the chosen user coordinate system appears also in the entry- and change window.
2. Click onto the window with the cursor or enter it by repeatedly pressing the tabulator key, and then change the name of the user coordinate system into its new version.
3. After that, click onto the field rename. This verifies the renaming and the new name of the user coordinate system appears in the choice window.

Please remember that the world coordinate system cannot be renamed.

Please remember that all changes made in the user coordinate system dialog box are registered only when you click on OK as you leave the box.

### **Erasing a user coordinate system**

Erasing a user coordinate system is similar to renaming a user coordinate system

1. Choose the user coordinate system to be deleted in the choice window by clicking onto it with the cursor and marking it so.
2. Click onto the field delete. The user coordinate system is then deleted, its name no longer appears in the choice window.

Please remember that the world coordinate system cannot be deleted. Please remember that all changes made in the user coordinate system dialog box are registered only when you click OK as you leave the box.

### **Changing the active coordinate system**

In every drawing it is possible to create as many user coordinate systems as you wish next to the world coordinate system, but only one user coordinate system can be the active coordinate system.

The **active coordinate system** is that coordinate system, whose parameters are active (e.g. place of the origin, adjustment of the axis).

1. Choose the user coordinate system to be active in the choice window by clicking onto it with the cursor and thus marking it. The name of the chosen coordinate system appears at the same time in the entry- and change window.
2. Click onto the field active. The user coordinate system thus becomes the active coordinate




system.

3. The field "active user coordinate system" shows the name of the active coordinate system. Please check there that the name of your chosen coordinate system appears there.

Changes to the active coordinate system are registered only when you click OK as you leave the dialog box.

### **Change to the previous coordinate system**

An option exists in the dialog window "User Coordinate System" to change to a previous coordinate system. It enables to quickly change between two coordinate systems. When clicking onto this field you will change to the previous coordinate system without another entry or verification.

Note: While a brief overview of UCS concepts has been provided here, users are encouraged to consult the **User Manual** , as well as other CAD documentation for a more in-depth exploration of the subject.

## **Command UNDEFINE**

**Command: UNDEFINE**

**Alias UF**

### **Located:**

1. **Entered Directly at the Command Line**

This command allows an application-defined command to override an internal AllenCad command. Entering a command name as response to the Undefine command suppresses that command. The suppressed command name can then be redefined to perform some other function.

You can undefine only built-in AllenCad commands. If an external application has redefined a command with the same name as a built-in AllenCad command, the application-defined command is active.

You can always access a built-in AllenCad command by preceding the command name with a period (.). Command REDEFINE restores AllenCad built-in commands that have been disabled with UNDEFINE.

Undelete: Withdraw Deletions



**Command: UNDELETE**

**Alias UD**

### **Located:**

1. **Pull-down Menu: Edit – Undelete**
2. **Top Function Bar**
3. **Palette: Edit**
4. **Or Enter Command or Alias at the Command Line**

Entities that have been deleted by mistake can be restored by the command UNDELETE.

Call the command

- By clicking the UNDELETE symbol;
- By selecting the option **Undelete** in the menu *Edit*; or
- By keyboarding UNDELETE in the command line area.

This will return objects deleted by the last command that removed objects from the drawing.

---

**See also:**

Command DELETE

## Undo



**Command: UNDO, U**

**Alias U**

**Located:**

1. **Pull-down Menu: Edit – Undo**
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

Invoking the command U (for Undo) will cause the last command made to be withdrawn and all changes in the drawing caused by this command to be canceled.

The command can be repeated until all working steps stored in the Undo List have been canceled. The program will then send out the message "Undo List is empty".

**Note:** The command U allows you only to reverse drawing database modifications. It does not undo modifications of the view. Use ZOOMBACK to restore previous views.

---

**See also:**

Command REDO

Command UNDO (Lisp)

### Undo Control

**Command: UNDO**

**Alias U**

**Located:**

1. **Pull-down Menu: Edit - Undo**
2. **Or Enter Command or Alias at the Command Line**

The program archives all steps made during the task in a list called the Undo List. The steps registered in this list can be withdrawn by using the UNDO function sequentially, each UNDO removing the next sequential step in the task.

Certain commands are not registered in the Undo List. These are commands and functions that control the viewport (Zoom, Pan, Window), set the parameters, and other transparent functions. These steps cannot be withdrawn by using UNDO.

The Undo List is kept only with the current session of the program. It will not be saved with the drawing.

The command can be repeated until all working steps stored in the Undo List have been canceled. The program will then send out the message "Undo List is empty".

### Options

Canceling the last work step (the first step appearing in the Undo List) is the normal mode of this function. The function can be altered to perform groups of UNDO steps as well, by following this procedure:

Enter UNDO in the Command Line area.

Define the number of steps to cancel by entering a number; or  
Select an option from the option bar.

### Control

The option field "Control" allows access to these options for controlling the UNDO process:

#### All

By selecting this switch you indicate that all (chosen) working steps are selected for the UNDO function.

#### None

No steps are registered in the UNDO LIST. The command is switched off.

#### Once

This option fixes the number of the steps you can cancel at the value 1. Only the last working step is registered in the UNDO LIST.

#### Erase

Selecting this switch causes erasure of the current UNDO LIST.

#### Comment

This option is a Switch that will trigger a prompt asking for the confirmation of the user's intentions. After selecting this field the option bar will show the possibilities ON and OFF. The active status will always be indicated as the preset value. The user may change this value, or go with it.

The options *Group-B*, *Group-E* and *Group-D* allow the user to select groups of steps. A number of defined procedures can be canceled with a single UNDO.

### Group-B

This switch defines the **Beginning** of the selected group. This "beginning" may be canceled, and any previous registrations in the Undo List remain in the list.

### Group-E

This option defines the **End** the group of working steps.

### Group-D

This option specifies that the group of steps selected for UNDO is **Deleted**. The individual steps remain in the UNDO LIST and can be canceled one by one.

The two following options, **Mark** and **Back**, allow the user to "mark" the listed steps of a task, and cancel all unmarked steps.

### Mark

This option defines the current state of the drawing as a state that can then be restored by the option **Back**.

### Back

This option restores the task to the previously marked state.

---

### See also:

Command U  
Command REDO

## Command UNITSETUP

**Command:** UNITSETUP

**Alias** US

**Located:**

### Located:

1. **Pull-down Menu: Tools – Default Unit Setup Precision**
2. **Or Enter Command or Alias at the Command Line**

This command is used in exactly the same way as the command SETUP. It is used to set or change the drawing unit display parameters for the current drawing. Please click on this link for a detailed description of its use.

# "V"

View Manager... (Saving / Retrieving Views)



**Command: VIEW**

**Alias V**

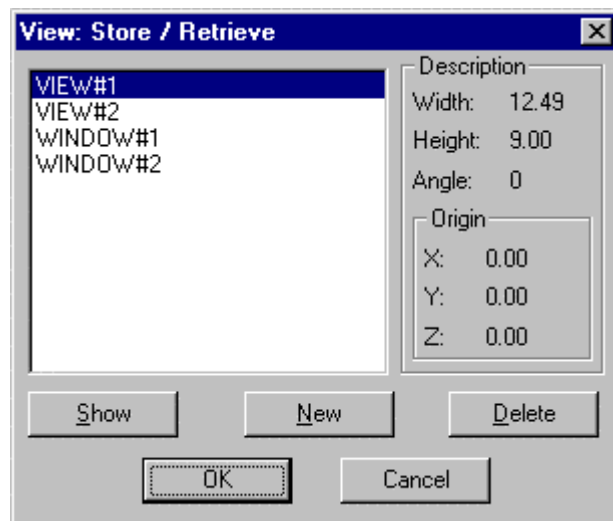
### Located:

1. **Pull-down Menu: View – View Manager**
2. **Left Side Bar**
3. **Palette: View**
4. **Or Enter Command or Alias Directly at the Command Line**

It is convenient when working in either Model mode or in Layout Mode to save different views of the drawing for later retrieval and display. The view of the drawing is essentially the viewer's 3-dimensional location and the direction of view toward the drawing. These are the elements that are stored in the named file and they correspond to the current drawing view when the view is saved.

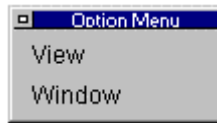
### Save a Named View

When selecting the option VIEW, the present portion will be saved as a user-defined view. Upon issuing the command, you will see the dialog:

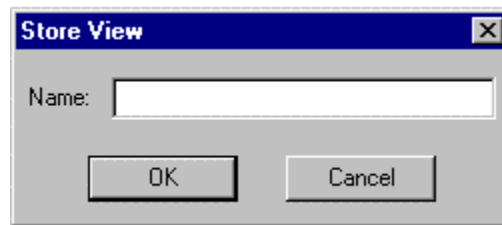


Here we can see that we already have two views and two windows already saved. To save another, simply

click on the **New** button. At this time you will be presented with the option:



If you choose view, the full screen image will be saved, whereas if you choose Window (a windowed view), you will be asked to define the window onscreen. This is done by picking upper left and lower right (or any diagonally opposite) window corners. After selection of option is complete and the window is defined, you are lastly presented with the dialog:



At this point, you specify the name for your view or window, and it will then be added to the list of other drawing views for later retrieval.

### **Retrieve a Named View**

A previously saved view can be retrieved through the same function, and can then become the present view. Follow these steps:

1. Choose one of the saved views in the dialog's list box;
2. Click on the **Show** button, located in the View Command dialog image above. The selected view will then become the present view;
3. Close the dialog box with OK to accept to display the chosen view. Choosing to abort will retain the previous view.

### **Delete the Name of a Defined View**

Views that are no longer needed, can be removed from the table of named views as follows:

1. Choose one of the names of a defined view in the list box;
2. Click onto the **Delete** button, located in the View Command dialog image above. The view will be removed from the list immediately.

### **View Parameters...**



**Command: VIEWPAR**

### **Alias VR**

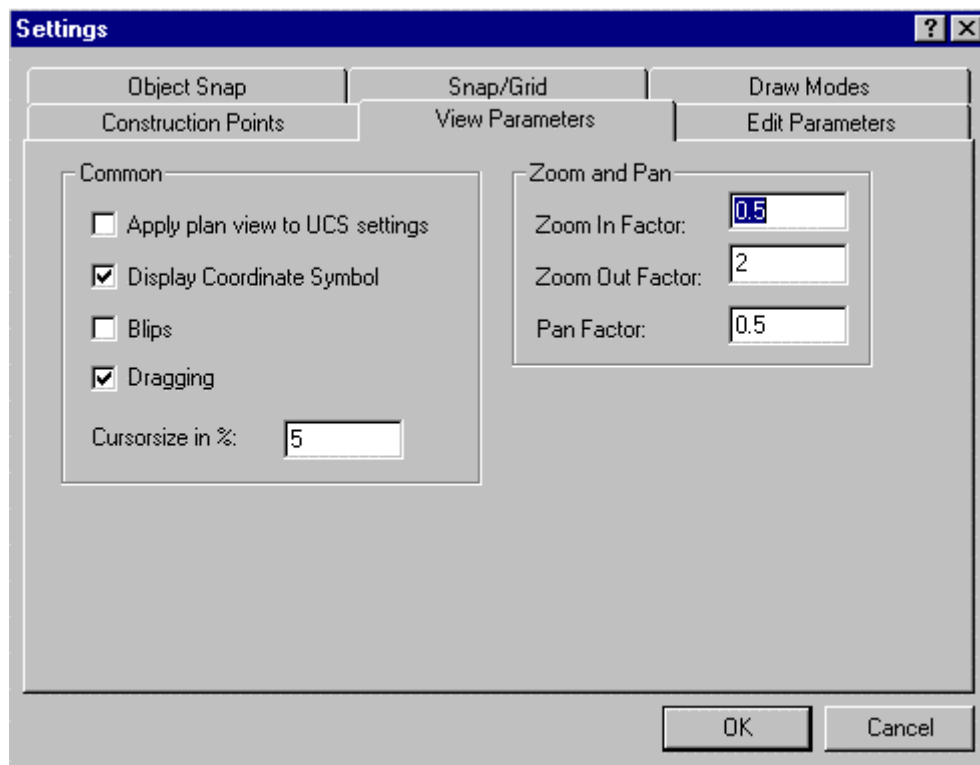
#### **Located:**

1. **Pull-down Menu: View – View Parameters**
2. **Palettes: Zoom and View**
3. **Or Enter Command or Alias at the Command Line**

Access to the View Parameters settings can be made either directly, as described above, or indirectly through the SETTINGS command. As the dialog illustrated below shows, View Parameters has its own

tab on the Settings dialog, which the dialog opens to when the VIEWPAR command is issued. Various parameters may be selected to define the direction and general size and shape of a given view of the drawing.

There are also default factor values inputs for the **ZOOM**ing and **PAN**ning of the view on the screen.



### Dialog Control Elements

#### Common

Here you see selections to be made that determine visibility of the coordinate symbol, pick blips (blipmode) and onscreen dragging representation. Here too you can specify the application of plan view to UCS settings and also control the size of the cursor symbol. Note that setting this value to 100% will extend the cursor axes across the entire visible screen area. This can be a helpful feature when aligning drawing objects visually onscreen.

#### Zoom and Pan

In the entry field *Zoom In Factor* and *Zoom Out Factor* the user may choose the default factor by which the drawing on the screen is to be enlarged or reduced. These factors affect only the visible portion of the drawing.

In the entry field *Pan Factor* the user may choose the factor by that the visible portion of the drawing is moved using the command PAN. A pan factor of 0.5, for example, moves the drawing portion to about half of the visible drawing area.

## Layout Viewport Control



**Command: VIEWPORT**

**Alias VP**

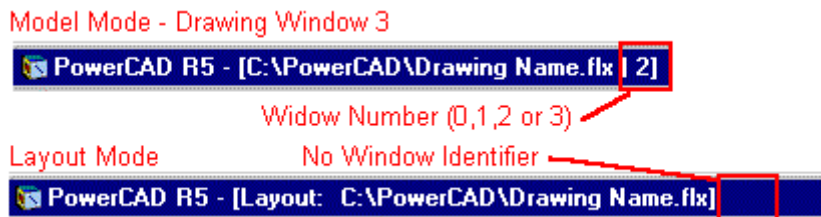
**Located:**

1. **Pull-down Menu: File – Layout Viewport Control**
2. **Left SideBar**
3. **Or Enter Command or Alias at the Command Line**

The command VIEWPORT command is used in layout mode to create rectangular windows that display the drawing model that resides in model space. In these viewports, you can control both the view direction and the display of a specific portion of the model. The VIEWPORT command options are: New, ON, OFF, Layer, and View. These functions are described in detail, below.

### Creating Using Drawing Windows

After switching to layout mode for the first time, usually you will create one or more viewports to show one or more views to your model.



Before creating a viewport, you must ensure that you are first in layout mode. At the top of the desktop window, you should check to see that "Layout" appears before your current drawing name, as in the above illustration. The command TLMODE toggles the display between layout and model mode and may need to be executed if you are still in model mode.

When you issue the VIEWPORT command, you are presented with the following dialog:



The option *New* of the command VIEWPORT predictably allows you to create a new viewport. New is also the default option of the VIEWPORT command. Next, you are asked to specify two opposing corners for the new viewport. If you have determined the size of the viewport that you need, you can either enter defining corner coordinates, or pick the corners of a rectangle drawn previously in layout mode, for this purpose. After specifying two corners of the viewport, the entire model is displayed in plan view and to its current extents, in the newly created viewport.

```
> VIEWPORT
First corner: 10,10
Opposite corner: @45,25
```

Then the next step will be to determine the size of the content of viewport to match to a certain scale and to maintain the accuracy of this scale between model space and paper space within these viewports.

Detailed hints and tips to apply appropriate techniques to set and keep a scaled relation between paper space viewports and model space are described later in this section after all the options of the command VIEWPORT have been discussed.

### Viewport Alteration using Modify Commands

A Viewport is a drawing entity represented by a rectangle and contains a certain view of the drawing.

Theoretically, any number of viewports may be created in layout mode.

Several editing commands can be performed on viewport entities, because these are treated as drawing objects:

- You can move a viewport to another position.
- You can change the size of a viewport with the Stretch or the scale command. In these cases, the view inside the viewport is not scaled with the viewport, only the frame, itself. If you scale a viewport, however, the displayed view inside of the viewport is scaled with the same factor as the viewport frame.
- You can remove viewports from the drawing, using the Delete command and picking the frame.
- Using `_Drawing_Windows` may be copied.
- Note also that, the commands Rotate, Array, Flip, and Mirror modify the position of a viewport (respectively a copy of a viewport), but the edges of the rectangle remain parallel to the X- and Y-axis of layout mode. Also, these commands maintain the displayed view in the rotated or copied viewport.
- When editing viewports and when drawing in paper space you can snap to the end point, mid point, etc. of the viewports rectangle, and you can utilize the coordinate filter options.

#### Command Options:

##### On/Off - Turning viewports off and on

The Viewport command options **On** and **Off**, control the visibility of the contents displayed in existing viewport windows. To turn the visibility of the viewport objects off, choose the option OFF and select the viewport frame or pick to an object inside the frame. To redisplay the representation of the model within a deactivated viewport, choose the option ON and pick an edge of the corresponding viewport frame.

The **visibility of the viewport frames** can also be controlled through layers. Like to any other drawing entity a viewport entity is located on a layer. It is recommended that you place viewport entities on one or two specific layers (for example LM\_VIEWPORT\_ON and LM\_VIEWPORT\_OFF). Now you can control the visibility of the frames of viewports by turning off or freezing a corresponding layer containing viewports.

##### Layer - Control of viewport contents

The VIEWPORT command option **Layer** permits manipulating the display in specific viewports by freezing and thawing drawing layers. This means that it is possible to have layers frozen and invisible in one viewport, yet unfrozen and viewable in another.

After choosing the option *Layer* the dialog box to control layer visibility in layout mode is displayed:

The list box on the left-hand side the dialog box contains all layers defined in the current drawing. The other list box is meant to determine those layers not to be displayed in the currently selected viewport ("Frozen Layers"). You can alter the visibility mode by marking layer names in one of the two lists and add or remove them from the list of frozen layers with clicks to the > button or < button. To freeze or thaw the status of entire layer groups, you can utilize the edit boxes by specifying strings with wild cards (for example "2D\_\*") and then click to the Filter button that will highlight all matching layers in the appropriate list.

##### View - Control in viewports

The VIEWPORT command option **View** toggles the layout window into a mode, where the selected viewport is displayed such that it covers the entire window. This mode serves to change the view to the model inside of the specified viewport.

```
> VIEWPORT
First corner: View
Select viewport: Specify viewport
Switch to viewport view ...
```



Viewport control:

When calling the option *View* the prompt "Viewport control:" is displayed. The following view options are at your disposal: Zoom, Pan, 3DView, View.

### **Zoom**

By intention the default prompt at the command line after choosing the Zoom function says "Zoom Scale factor: ". This will remind you that - once a scaled relation between model space and paper space has been established - you should manipulate the view within the viewport only via scaled zooming (usually by specifying whole numbers) to maintain the scale. In addition, however, the option bar contains the other zoom options (Window, Extents, Previous) as usual.

### **Pan**

The option *Pan* solely moves the visible portion within a viewport to a direction specified by two points without changing the scale of the model representation in paper space.

In *Viewport control* mode any transparent command may be used. By this, you can also utilize the pan functions from the control panel (Pan Left, Pan Right, Pan Up, Pan Down); for example:

Viewport control: 'PANLEFT

### **3DView**

Naturally, the layout mode can be utilized also to illustrate the model from different viewpoints in space. The option *3DView* of viewport control allows you to set individual 3D views or predefined views like plan view, front view, left side view.

The option *3DView* will open a dialog box similar to the one invoked by the command 3DVIEW in model space.

**Note:** Using the option *3DView* will zoom the contents of the viewport to the drawing extents. To achieve a scaled relation between paper space and model space for isometric or other individual 3D views of the model, it is recommended that you use named views prepared in model space, as discussed next.

### **View**

The option *View* allows you to display views in specified viewports, that have been saved as named views in model space. Layout mode efficiencies can be improved by saving named views in model mode designed for reuse in the viewports of the paper space. This is especially true if maintaining a scaled relationship between model and print space is required. See the section *Techniques and Tips for Layout Mode* below, for further hints and recommendations.

The option *View* will open a dialog box similar to the one called by the command VIEW in model space.

A particularly useful help feature is provided with the Show button in the Save/Retrieve named views dialog box, because this option allows you to preview a selected named view in the drawing before confirmation.

**Note:** When calling the command VIEW in layout mode outside the command VIEWPORT, you can create named views within paper space (for example to zoom fast to portions of the drawing sheet).

### **Utilizing transparent commands**

You can utilize in *View* control mode all transparent commands. As already mentioned, you can call the functions 'PANLEFT, 'PANRIGHT, 'PANUP, 'PANDOWN from the control panel. Another example is the transparent command 'SETVIEWDIR to set an individual or standard isometric 3D view within a viewport. Or, the transparent command 'DIST that allows you to measure the distance between two points.

### **Techniques and Tips for Layout Mode**

Create your construction (in model space) in real world units (at scale 1:1). Utilize specific layers for

dimensions, hatching, text, etc. When managing and using layers, keep in mind that you might want to illustrate different "logical views" of your model later, when presenting or plotting your construction with the aid of layout mode.

At a certain stage of drawing construction, you may need to print out the drawing for presentation within your company or to forward it to a manufacturer. In this situation layout mode can help in many cases with the quality of your presentation:

- a. Switch from model space to paper space using the command TLMODE. The title bar now says Layout: followed by the drawing name.
- b. Set the limits to equal the paper size in those dimensions (units) that will be used when printing (for example A-size, B-size, C-size, ISO-A0, ISO-A1, ISO-A2, ... Letter Portrait, Letter Landscape).
- c. Create a layer for the title block (unless not existent), set it current, and insert your title block in paper space at coordinate origin (0,0). Or, draw a rectangle on a temporary layer with the paper size you will use when printing.
- d. Create a layer to place the viewports on. Make this layer current before you create new viewports with the VIEWPORT command and make sure to use this specific layer only for viewports. It is recommended that you activate the snap mode with applicable units (inch, millimeter, centimeter, ...) when creating viewports.

The next step, once a viewport has been established, is to set up scaling between the model and its representation on the drawing sheet. This step sometimes causes headaches. Setting up the appropriate scale for displaying the model, can be prepared in advance, in model space in several ways:

- Positioning rectangles on a temporary layer over sections of your drawing in model space that match viewport windows in paper space at a particular scale;
  - Creating named views in model space with the command VIEW that then can be retrieved within the viewports in paper space with the option View.
- e. Once the scaled relation is set up, you should avoid uncontrolled zooms in the viewport, and confine yourself to Pan commands and to the ZOOMFAC command (usually specifying whole numbers).

To modify the height or length of a viewport, we recommend you use the command STRETCH or SCALE. The display of the model in the specified viewport will not be scaled with this command. It maintains the scaled relation between model space and paper space.

Keep in mind the "golden rule":

*Drawing entities you create in model space can only be edited in model space. Drawing objects you create or insert in paper space can only be modified in paper space.*

---

**See also:**

Layout Mode  
TLMODE command  
Command VNOTE



**Command: VNOTE (VoiceNOTE)**  
**Alias VN**

**Located:**

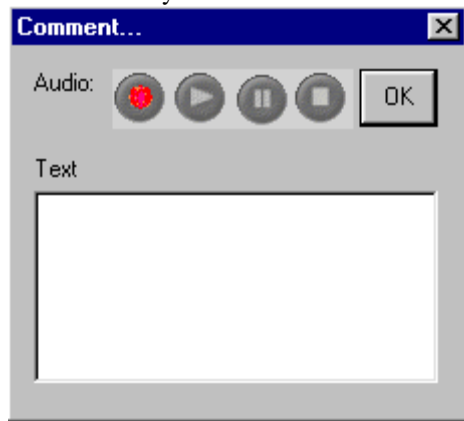
**9. Draw Menu > MarkUp > Insert/Edit VoiceNote**

**10. Left Side Bar**

**11. Palette Draw**

**12. Enter Command or Command Alias Directly at the Command Line**

VoiceNOTES may be embedded into a drawing by using a feature that attaches audio and text to a special comment entity.



**Adding VoiceNOTES**

VoiceNOTES can be added to a drawing by selecting Insert/Edit VoiceNOTE from the Draw menu.

Select a point in the drawing where you would like to place the VNOTE icon. The dialog box Comment... is then displayed allowing you to enter text and/or to record your voice.

Once a VNOTE is entered, select the OK button to place your comment in the drawing.

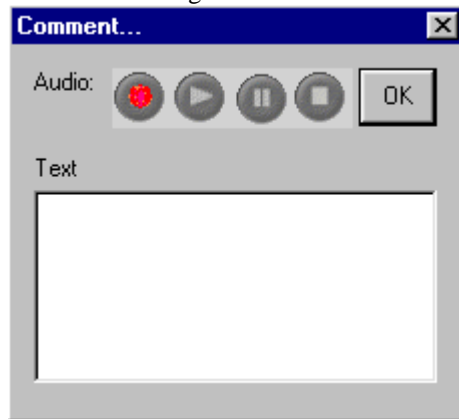


In the dialog above, simply type your text in the window, or use the recorder buttons to record an audio VoiceNOTE.

**Recording Your Voice**

Note that to record your voice, you will need a microphone, audio hardware and audio software installed and properly configured. To playback your recording, speakers are also required.

To record your voice as a VNOTE, either add or retrieve an existing VoicenNOTE icon after issuing the



VNOTE command. This will open the dialog box

In the dialog box Comment... there are four audio controls: record, play, pause and stop. Selecting the

record control will record audio until the stop control is pressed. Each new recording completely removes the previous recording. You may playback a recording by selecting the play control. When recording or playing back audio, the pause and stop controls can be used as needed.

### **Retrieving or Changing VoiceNOTE Comments**

VoiceNOTES can easily be retrieved for reading, listening or also editing. Firstly choose Insert/Edit an Audio Comment from the menu Tools or issue the VNOTE command in any of the above-mentioned locations.

After selecting a VNOTE icon in the drawing, the dialog box Comment...is display with the attached text and/or audio recording. You can read the text message (if there is one) in the dialog window, or you can click on the play button to play an audio recording. Alternately, the text edit box may now be edited or a new speech recording made that overwrites a previous recording. The text or audio recording is updated when the OK button is selected.

### **Removing VoiceNOTES**

VoiceNOTES can be removed by selecting Delete from the menu Edit.

Select the VNOTE icon containing the comment to be removed. Removing this icon will remove the attached audio and/or text comments.

#### **Notes:**

9. VNOTES are stored in Extended Entity Data format and appended to attributes to the fcomment.flx block in the drawing database and externally in the \support directory.
10. VNOTES are primarily for use when the drawing file is loaded in AllenCad. The drawing file can be saved in DWG or DXF format, without losing the recorded information, however this information is not directly addressable from the AutoCAD program. AutoCAD simply ignores their presence in the drawing file.
11. VNOTES stored externally in the \support directory can be exchanged with any other block. Within the drawing file, they are created with the current layer, color and linetype settings. The scale factor of the inserted block is screensize divided by 9.
12. VNOTES are not limited in size, as additional attributes are added for larger sized recordings..

**Tip:** In complex drawings, placing VoiceNOTES on a unique layer can make them easier to locate, especially when the drawing will be shared with mobile AllenCad CE users.

## **Creating and Viewing Slides**

### **Commands: MSLIDE**

#### **MSLIDE is Located:**

1. Enter Command or Alias at the Command Line

### **VSLIDE (Alias VS),**

#### **VSLIDE is Located:**

2. Pull-down Menu: View – View Slide
3. Or Enter Command or Alias at the Command Line

### **SLDOUT (Alias SO)**

#### **SLDOUT is Located:**

1. Pull-down Menu: View – Take Snapshot (Slide)
2. Or Enter Command or Alias at the Command Line

**The Remainder are Entered at the Command Line**

The command SLDOUT (or MSLIDE) lets you save a view of the current drawing as a slide. Creating a slide file is like capturing the screen of a drawing.

You can bring back to screen a saved slide file by calling the VSLIDE command.

The extension of slide files is *.sld*.

Slide files can be used for certain purposes as an alternative to WMF files or Bitmap files (see commands WMFOUT and BMPOUT):

- Especially, you can utilize Slide files in icon menus. Icon menus are called via the ICONMENU command (see above).
- You can collect slide files in **image libraries**. Image libraries have an extension *.ilb*. Images from image libraries (either WMF, BMP, or Slides) can be utilized in dialog boxes that are called by a Lisp routine or another application.

Detailed information on how to use icon menus and display images in dialog boxes is found in the *Programmer's Guide*.

#### To create a slide file

Follow these steps:

1. Display a view of the drawing that you want to save as a slide file.
2. Type SLDOUT.
3. In the Create File dialog box, specify the location and the name for the slide file and choose OK.

#### To view a slide file

Follow these steps:

1. Type VSLIDE
2. In the View Slide dialog box, choose the slide you want to view
3. Choose Open. The slide will then be displayed on the screen
4. Type REDRAW to return to your drawing.

**Note:** You cannot draw or edit a slide.

#### Vertex Editing



**Command: VTXEDIT**

**Alias VX**

##### Located:

1. **Pull-down Menu: Modify – Edit Polyline – Vertex Editing**
2. **Palette: Modify**
3. **Or Enter Command or Alias Directly at the Command Line**

Editing of single segments of a polyline is done through the command **Vertex Editing**. This command allows the user to change the vertices (the start and end points) of the polyline. Vertices of a polyline segment can be inserted, deleted, and moved. Also, the width of the polyline segments can be changed.

Select the function by selecting the option Vertex editing in the submenu *Modify Polyline* of the menu *Modify*, or by typing the command VTXEDIT.

#### Vertex Selection

Select the polyline containing the vertex to be edited. The selection of the polyline will also cause the selection of the vertex containing the start point as the first vertex to be edited.

The vertex selection may be changed by "stepping" through the vertices using the **Next** and **Previous**. The user should experiment with these selections until he or she understands how they work.

The selected control point will always be marked by a "rubber band" line joining the selected point with the current cursor position.

After selecting the control point to be edited, select one of the editing options from the Options Bar:

#### **OPTIONS:**

##### **Insert**

The option **Insert** allows the addition of vertices, and thereby of further segments, to an existing polyline.

Define (either by picking with the cursor or by input of coordinates) the position of the additional vertex.

The width of this new segment will correspond to the width values of the polyline, at the bordering control points, before the addition of the new segment.

After inserting a segment the command VTXEDIT stays active. The user may select another operation or complete the use of the command by pressing ESC.

##### **Move**

The option **Move** enables the user to relocate a vertex of the polyline to a new position.

First determine the vertex to be relocated. Then define (by "picking" with the cursor or by input of coordinates) the new position of the chosen vertex. The vertex will then be moved to this position. Other segments of the polyline will remain unchanged.

The command will remain active. Finish editing the polyline segments by pressing ESC.

##### **Delete**

Define the vertex to be deleted in the same fashion as was used with the previous options. Then select the option **Delete** from the Options Bar.

The segment having its start point in the selected vertex will be deleted. The previous (in order of drawing) segment will be lengthened to the next control point.

The command VTXEDIT remains active. Select another option or finish editing the polyline segments by pressing ESC.

##### **Width**

Width allows the width of single segments of a polyline group to be changed.

Define the vertex to be deleted in the same fashion as was used with the previous options. Then select the option **Width** from the Options Bar.

Input, or select with the cursor, the desired width of the polyline segment. The width of the selected segment will be changed according to the inputs.

The command VTXEDIT remains active. Select a another operation or finish editing the polyline segments by pressing ESC.

---

#### **See also:**

POLYEDIT

# "W"

## Cascade Drawing Windows



**Command WCASCADE**

**Alias WD**

**Located:**

1. **Pull-down Menu: Window - Cascade**
2. **Or Enter Command or Alias at the Command Line**

Arrange drawing windows in cascade format.

## Close Drawing Window

**Command: WCLOSE**

**Alias WC**

### **Located:**

1. **Pull-down Menu: Window – Close Current Viewport**
2. **Or Enter Command or Alias at the Command Line**

It is possible to close the current viewport, as in all Windows programs, by double-clicking on the system menu area (left corner on top). This activates the command WCLOSE and the window will be closed. This command can also be keyboarded.

## Arrange Icons

**Command: 'WICONARR**

**Alias WI**

### **Located:**

1. **Pull-down Menu: Window – Arrange Icons**
2. **Or Enter Command or Alias at the Command Line**

Arrange all drawing window icons.

## Copy Region to Windows Metafile



**Command: WMFOUT**

**Alias WM**

### **Located:**

1. **Pull-down Menu: File – Export Region to Metafile**
2. **Palette: Raster**
3. **Or Enter Command or Alias at the Command Line**

The following options allow the region to be copied into a WMF vector format file:

### **Window**

Define a window by setting two points.

### **View**

The complete current view will be exported into a bitmap.

### **Extents**

The region to be copied is defined by the extent of the drawing.

Finally, you may choose whether the picture should be written into a **File** or copied to the Windows **Clipboard**.

**Notes:**

1. Only **drawing entities** are affected by this command. Imported raster graphics will not be included in the selection set.
2. The WMF vectors remain in the clipboard until you clear it.

## Open New Drawing Window

**Command:** WOPEN

**Alias** WO

**Located:**

1. **Pull-down Menu:** Window – Open New Viewport
2. **Or Enter Command or Alias at the Command Line**

In order to open a viewing window a drawing must have been created, or an existing drawing must be open. If more than one drawing has been opened, the user should remember that all commands are executed in the active drawing.

From the menu *Window* choose the option **Open Another Viewing Window....**

Before the program opens another window you will be asked to determine the type of view in this new window. Choose an individual viewpoint or a standard view from the dialog box, and click on OK.

The chosen angle of view can be changed at any time. The new window will be created in standard size and arranged in the manner proscribed with the existing windows.

## Tile Drawing Windows



**Command:** 'WTILE

**Alias** WT

**Located:**

1. **Entered Directly at the Command Line**

Arrange drawing windows in a tiled format.

## Tile Horizontally



**Command:** 'WTILEHOR

**Alias** WH

**Located:**

1. **Pull-down Menu:** Window – Tile Horizontal
2. **Or Enter Command or Alias at the Command Line**

Arrange drawing windows in a horizontally tiled format.

In the case that more than one viewport has been opened, you can arrange them in a horizontal order by choosing the WTILEHOR command.

## Tile Vertically





**Command: 'WTILEVER**

**Alias WV**

**Located:**

- 1. Pull-down Menu: Window – Tile Vertical**
- 2. Or Enter Command or Alias at the Command Line**

Arrange drawing windows in a vertically tiled format.

In the case that more than one viewport has been opened, you can arrange them in a vertical order by choosing the WTILEVER command.

**"X"**

Attach an External Drawing File to the Current Drawing

**Command: XINSERT**

**Alias XI**

**Located:**

- 1. Entered Directly at the Command Line**

The command XINSERT is processed similar to the XLINK command applied with the *Attach* option. The command is provided for convenience, allowing you to directly attach an external drawing to the current drawing.

---

**See also:**

XLINK

## Externally Referenced Drawings



**Command: XLINK**

**Alias XL**

**Located:**

- 1. Pull-down Menu: Parts – External References**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Externally Referenced Drawings (XREF's) serve especially to

10. Create assembly drawings from different files
11. Insert detail drawings into a finished one
12. Temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is realized within a running project where a workgroup uses XREF's to create several drawings and details in a network.

Another advantage is that within the current drawing, only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file. This keeps the file size small and as such it is easier to work with. In this way, very complex drawings can be created from a group of

relatively small files. In these instances, XREF's have an advantage over the use of blocks.

## **XLINK command: Control externally referenced drawings**

The XLINK command allows you to create a link to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

### **Differences to Part Insertions**

Although the treatment of linked drawings is basically similar to those of part insertions (block insertions), some fundamental differences include:

An Xref establishes a link to another drawing file, but it does not become a permanent part of the current drawing.

### **Dependent Symbols**

When attaching an external file, those symbol tables are loaded into current drawing that are required to display the referenced coincident to its original: layers, linetypes, text styles, dimension styles, and block definitions.

The program utilizes the following scheme to name dependent symbols in the current drawing:

Logical\_Xref\_Name|Dependent\_Symbol\_Name

For example, the layer names of an externally referenced drawing attached with the logical name SHAFT are listed as:

SHAFT|CONSTRUCTION

SHAFT|DETAILS

SHAFT|DIMENSION

Other dependent symbol table entries like linetypes or blocks will carry the same naming convention.

Note that you cannot redefine or rename dependent symbols. Also, dependent blocks cannot be inserted and a dependent layer cannot be made the current layer.

But the dependent symbol conventions allow you to control the visibility, the colors, and the linetypes of externally referenced drawings. The layer dialog box displays the names of the dependent layers and you can apply the options On/Off, Thaw/Freeze, Linetype, and Color on them.

The logical name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

To keep the visibility and display settings over subsequent sessions, make sure to set the system variable **VISRETAIN** to 1. By default VISRETAIN is set to a value of 0 that means that visibility and display modifications made to the dependent layers are not maintained when you leave the current drawing.

### **The Dialog Box *Externally Referenced Drawings***

The dialog box displays a list box and a column of option buttons.

The list box displays the following information on externally referenced drawing files that have been inserted into the current drawing

1. The logical name assigned to it when the file was attached, and
2. The path and filename of the referenced drawing

The options available to manage externally referenced drawings are provided in a column of buttons:

<b>Attach</b>	Insert an external drawing file into the drawing as block
<b>Reload</b>	Update externally referenced drawing to display its current state
<b>Path</b>	Relocate/redirect or alter source of the referenced file

<b>Bind</b>	Transform an externally referenced drawing to an ordinary block in the drawing
<b>Detach</b>	Delete an externally referenced drawing from the current drawing and erase the dependent symbols from the drawing database tables

These options are described in detail in the following sections.

### **Attach**

The *Attach* option allows you to insert an external drawing as a block to the current drawing.

#### **To attach (set up a link and insert the contents) to an external drawing file:**

1. The standard file dialog box allows you to specify the drawing you want to attach to the current drawing.
2. Within the dialog box *Attach External Drawing*, a unique Logical Name (block name) needs to be determined for the associated referenced file. The program suggests using the name of the file. However, in many cases it is recommended to use a short logical name, because the logical name will be used as prefix for the names of dependant symbol names for layers, linetypes, etc.
3. The other options of the dialog box let you specify preferences for the insertion and are identical to those found in the dialog box of the INSERT command (insertion point, scale factor, and rotation angle), as described previously.

The drawing attached is based on the most recently saved version of the drawing file.

### **Reload**

The option *Reload* allows you to update an external reference any time during a current drawing session. Any Reload reflects any subsequent modification made to the external file.

Note that when you open a drawing that contains externally referenced drawings, these are displayed automatically in their current state. When working on a project in a workgroup over a network, it might be necessary to update one or more external references while you are working on a current drawing. The program will reload the specified drawing(s) in the shape as they have been saved most recently.

#### **To reload externally referenced drawings in a current drawing session,**

1. select the external reference to be updated from the list box
2. click the *Reload* button

#### **Path (Relocate or alternate source of the referenced file)**

Occasionally the storage location of an externally referenced drawing might have been changed or the referenced file has been renamed. Also, it might happen that, when sending the project files including referenced drawings to someone else, the path to the referenced files needs to be updated.

In such a situation the option *Path* of the dialog box allows you to re-establish the path to the referenced drawing, to reload a renamed referenced file, or to replace a referenced drawing by another file.

#### **To edit or relocate the path to a previously linked reference drawing,**

1. select the external reference to be relocated from the list box
2. click the *Path* button

From now on, the link to the referenced file will be maintained as specified and the data of the referenced file will be updated from that source.

### **Bind**

In certain situations it is useful to make an externally referenced drawing a permanent part of a drawing, for example when you need to send the drawing to someone else or if it has been finalized and is going to be archived. Binding the referenced drawing to the current drawing has the advantage that there is no need

to gather an assembly of drawings within an archive or mail.

The option *Bind* of the XLINK command allows you to transform an externally referenced drawing to an ordinary block in the drawing. Once you bind an externally referenced drawing, the link to it is broken and you will no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definition, text styles, and dimension styles into the current drawing. Within each corresponding dependent symbol name, the vertical bar symbol (|) is replaced by a new three character symbol \$n\$, where n is a number that is usually 0. For example, the layer name HOUSE|BASEMENT will be altered to HOUSE\$0\$BASEMENT.

**To bind one or more external references to the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Bind* button

**Detach**

Externally referenced drawings that are not needed any longer in the current drawing can be removed from the base drawing with the *Detach* option. This will remove all of its dependent layer, linetype, and block references as well.

Note: Although you can erase an externally referenced drawing with the DELETE command, it is recommended to use the Detach option of the XLINK command, as this operation will also remove the Block definition from the drawing database.

**To remove one or more external references from the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Detach* button

**Open**

As the Multiple Document Interface allows you to display up to four drawings on the desktop, the option *Open* of the dialog box allows you to open an externally referenced drawing simultaneously on the CAD desktop. This allows you to modify a referenced drawing, for example, if you have detected discrepancies or inconsistencies in the linked drawing that needs to be fixed.

**To open a referenced drawing in another drawing window (viewport),**

1. select the external reference you want to open from the list box
2. click to the *Open* button

Once you have modified a referenced drawing, save the drawing (and, if the modification has been finished, close it). To display the revised external reference in the master drawing, call the command XLINK and reload the corresponding file (see above: *Reload*).

**Insert**

The option *Insert* allows you to insert another copy of the Xref chosen from the list box into the current drawing. You may then define the insertion point, the scale factor and the rotation angle for that additional instance of the external drawing in the current drawing.

You can move, scale, rotate, and copy externally referenced drawings. But, you cannot explode an external reference.

**Note:** You should not remove an Xref from the current drawing with the DELETE command, because it does not purge the Xref block definition from the drawing database and the associated symbols will be maintained. To remove an external reference correctly from the current drawing, it is recommended you use the Detach option of the XLINK command.

The Insert option, however, can serve to reinsert an external reference that accidentally has been deleted.

---

See also:

XINSERT

## Explode Complex Objects...



**Commands:** EXPLODE, XPLODE

**Alias** EX. XPL

### **Located:**

1. **Pull-down Menu:** Edit – Explode Complex Objects
2. **Palettes:** Edit and Parts
3. **Or Enter Command or Alias at the Command Line**

Parts and other complex objects, like polylines, crosshatches, and dimensioning, inserted in the drawing constitute one single drawing object.

In this mode, it is not possible to edit any of the individual entities that make up the part. If, for any reason, there is a requirement to edit one of the entities comprising a part, then the part must be "exploded" (broken into its individual entities), the element(s) edited, and then the component parts reassembled again.

The explosion of complex objects is executed by the command EXPLODE, or by selecting and activating the option **Explode Complex Objects...** in the *Edit* menu.

When the object selection is complete, the objects (parts, polylines, hatches, or dimensions) will be exploded.

Now you may choose any of the entities for individual editing.

### **Notes:**

1. It is important to realize that, if there are attributes attached a part to be exploded, these attributes will be lost. Instead, the attributes defining names will appear.
2. Exploding a part does not change the part definition in the drawing. Another part or parts with the same name and original definition can be re-inserted into the drawing at any time.
3. To change the part definition in the drawing, you must modify the part components after exploding, then create a new internal or external part with the PARTEXP or PARTDEF commands, using the old part name. The program will ask you to confirm the re-definition when you do so.

Drawing Navigator...



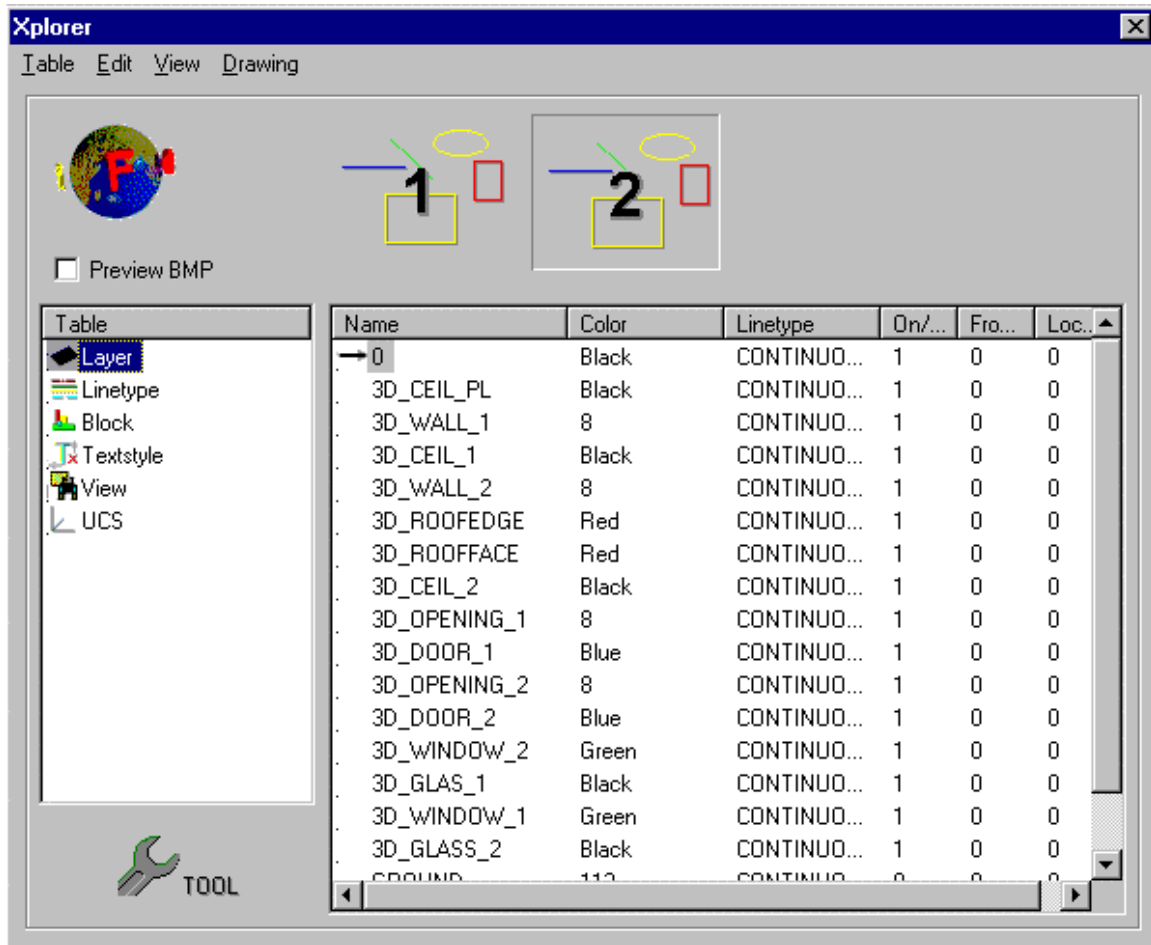
**Command:** XPLODER

**Alias** XPL

### **Located:**

1. **Pull-down Menu:** File– Drawing Xplorer
2. **Top Function Bar**
3. **Or Enter Command or Alias at the Command Line**

The Drawing Xplorer offers you fast and uncomplicated access to important information on the currently open drawing(s). Furthermore, you can use the Xplorer to alter various properties of the current drawings.



More precisely, the Drawing Xplorer is a tool to manage the named objects in the drawing database tables. With it, you can manage the tables and table entries of layers, linetypes, textstyles, views, user coordinate systems, and blocks.

Type in the command `XPLORER` to activate this tool or select *Drawing Xplorer* from the *File* menu.

The *Xplorer* window displays the preview bitmap of the currently open drawings in the area underneath the menu bar. You switch to another drawing table, click the appropriate bitmap. A ToolTip over the corresponding bitmap displays the drawing name including drive and path.

The left-hand list box titled *Table* serves to select and access a specific drawing database table. The table entries with all its named objects and characteristics will then be displayed in the right-hand list box.

In detail, the drawing Xplorer permits the following:

- Delete unused, not referenced table entries (allowed only for the active drawing)
- Rename table entries (allowed only for the active drawing)
- Set the current layer, current linetype, current textstyle, current user coordinate system, or current named view (allowed only for the active drawing) by double clicking to one of the table entries
- In case the *Block* table is the current one, you can insert a block by double clicking to its name.

Clicking the *Tool* button will invoke the main corresponding function associated to the active table.

For example, if *Layer* is the Drawing Navigators current table, you will call the Layer Manager dialog window by clicking the *Tool* button. This way, you might create new layer entries.

## Externally Referenced Drawings



**Command: XLINK**

**Alias XL**

### **Located:**

- 1. Pull-down Menu: Parts – External References**
- 2. Palette: Parts**
- 3. Or Enter Command or Alias Directly at the Command Line**

Externally Referenced Drawings (XREF's) serve especially to

13. Create assembly drawings from different files
14. Insert detail drawings into a finished one
15. Temporarily reference another drawing in the current drawing

Often, the biggest benefit of using externally referenced drawings is realized within a running project where a workgroup uses XREF's to create several drawings and details in a network.

Another advantage is that within the current drawing, only a link to a referenced drawing is stored but not the entire entity and symbol database information of the referenced file. This keeps the file size small and as such it is easier to work with. In this way, very complex drawings can be created from a group of relatively small files. In these instances, XREF's have an advantage over the use of blocks.

### **XLINK command: Control externally referenced drawings**

The XLINK command allows you to create a link to an external drawing and to control externally referenced drawings (also called Xrefs) in the current drawing.

### **Differences to Part Insertions**

Although the treatment of linked drawings is basically similar to those of part insertions (block insertions), some fundamental differences include:

An Xref establishes a link to another drawing file, but it does not become a permanent part of the current drawing.

### **Dependent Symbols**

When attaching an external file, those symbol tables are loaded into current drawing that are required to display the referenced coincident to its original: layers, linetypes, text styles, dimension styles, and block definitions.

The program utilizes the following scheme to name dependent symbols in the current drawing:

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For example, the layer names of an externally referenced drawing attached with the logical name SHAFT are listed as:

SHAFT|CONSTRUCTION

SHAFT|DETAILS

SHAFT|DIMENSION

Other dependent symbol table entries like linetypes or blocks will carry the same naming convention.

Note that you cannot redefine or rename dependent symbols. Also, dependent blocks cannot be inserted and a dependent layer cannot be made the current layer.

But the dependent symbol conventions allow you to control the visibility, the colors, and the linetypes of

externally referenced drawings. The layer dialog box displays the names of the dependent layers and you can apply the options On/Off, Thaw/Freeze, Linetype, and Color on them.

The logical name prefix makes it easy to distinguish the dependent layers from the layers originally defined in the current drawing.

To keep the visibility and display settings over subsequent sessions, make sure to set the system variable **VISRETAIN** to 1. By default VISRETAIN is set to a value of 0 that means that visibility and display modifications made to the dependent layers are not maintained when you leave the current drawing.

### **The Dialog Box *Externally Referenced Drawings***

The dialog box displays a list box and a column of option buttons.

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<b>Attach</b>	Insert an external drawing file into the drawing as block
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These options are described in detail in the following sections.

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#### **To attach (set up a link and insert the contents) to an external drawing file:**

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2. Within the dialog box *Attach External Drawing*, a unique Logical Name (block name) needs to be determined for the associated referenced file. The program suggests using the name of the file. However, in many cases it is recommended to use a short logical name, because the logical name will be used as prefix for the names of dependant symbol names for layers, linetypes, etc.
3. The other options of the dialog box let you specify preferences for the insertion and are identical to those found in the dialog box of the INSERT command (insertion point, scale factor, and rotation angle), as described previously.

The drawing attached is based on the most recently saved version of the drawing file.

#### **Reload**

The option *Reload* allows you to update and external reference any time during a current drawing session. Any Reload reflects any subsequent modification made to the external file.

Note that when you open a drawing that contains externally referenced drawings, these are displayed automatically in their current state. When working on a project in a workgroup over a network, it might be necessary to update one or more external references while you are working on a current drawing. The program will reload the specified drawing(s) in the shape as they have been saved most recently.

#### **To reload externally referenced drawings in a current drawing session,**



1. select the external reference to be updated from the list box
2. click the *Reload* button

#### **Path (Relocate or alternate source of the referenced file)**

Occasionally the storage location of an externally referenced drawing might have been changed or the referenced file has been renamed. Also, it might happen that, when sending the project files including referenced drawings to someone else, the path to the referenced files needs to be updated.

In such a situation the option *Path* of the dialog box allows you to re-establish the path to the referenced drawing, to reload a renamed referenced file, or to replace a referenced drawing by another file.

#### **To edit or relocate the path to a previously linked reference drawing,**

1. select the external reference to be relocated from the list box
2. click the *Path* button

From now on, the link to the referenced file will be maintained as specified and the data of the referenced file will be updated from that source.

#### **Bind**

In certain situations it is useful to make an externally referenced drawing a permanent part of a drawing, for example when you need to send the drawing to someone else or if it has been finalized and is going to be archived. Binding the referenced drawing to the current drawing has the advantage that there is no need to gather an assembly of drawings within an archive or mail.

The option *Bind* of the XLINK command allows you to transform an externally referenced drawing to an ordinary block in the drawing. Once you bind an externally referenced drawing, the link to it is broken and you will no longer see any modifications made in the original referenced file.

Bind also incorporates the layers, linetype definition, text styles, and dimension styles into the current drawing. Within each corresponding dependent symbol name, the vertical bar symbol (|) is replaced by a new three character symbol \$n\$, where n is a number that is usually 0. For example, the layer name HOUSE|BASEMENT will be altered to HOUSE\$0\$BASEMENT.

#### **To bind one or more external references to the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Bind* button

#### **Detach**

Externally referenced drawings that are not needed any longer in the current drawing can be removed from the base drawing with the *Detach* option. This will remove all of its dependent layer, linetype, and block references as well.

Note: Although you can erase an externally referenced drawing with the DELETE command, it is recommended to use the Detach option of the XLINK command, as this operation will also remove the Block definition from the drawing database.

#### **To remove one or more external references from the current drawing,**

1. select the corresponding entries in the list box of the dialog box
2. click the *Detach* button

#### **Open**

As the Multiple Document Interface allows you to display up to four drawings on the desktop, the option *Open* of the dialog box allows you to open an externally referenced drawing simultaneously on the CAD desktop. This allows you to modify a referenced drawing, for example, if you have detected discrepancies or inconsistencies in the linked drawing that needs to be fixed.

**To open a referenced drawing in another drawing window (viewport),**

1. select the external reference you want to open from the list box
2. click to the *Open* button

Once you have modified a referenced drawing, save the drawing (and, if the modification has been finished, close it). To display the revised external reference in the master drawing, call the command XLINK and reload the corresponding file (see above: *Reload*).

**Insert**

The option *Insert* allows you to insert another copy of the Xref chosen from the list box into the current drawing. You may then define the insertion point, the scale factor and the rotation angle for that additional instance of the external drawing in the current drawing.

You can move, scale, rotate, and copy externally referenced drawings. But, you cannot explode an external reference.

**Note:** You should not remove an Xref from the current drawing with the DELETE command, because it does not purge the Xref block definition from the drawing database and the associated symbols will be maintained. To remove an external reference correctly from the current drawing, it is recommended you use the Detach option of the XLINK command.

The Insert option, however, can serve to reinsert an external reference that accidentally has been deleted.

---

**See also:**

XINSERT

## Command X,Y



### Command X,Y

**Located:**

1. **Left Side Bar**
2. **Or Enter Command Directly at the Command Line**

The X,Y command is an alternate method used for 2-D point entry, where you may want to specify an existing point in the drawing that has the same x or y coordinate value. Whenever you are prompted by the program to enter a point, you can issue the X,Y command and you will be prompted separately for x and y input values. At the prompt, you can enter these values directly by typing them at the command line, or else you can select an object snap and select a point in the drawing that has the correct value.

See also: Command X,Y,Z  
Coordinate Point Filters

## Command X,Y,Z



### Command X,Y

**Located:**

1. **Left Side Bar**
2. **Or Enter Command Directly at the Command Line**

The X,Y,Z command is an alternate method used for 3-D point entry, where you may want to specify an existing point in the drawing that has the same x, y or z coordinate value. Whenever you are prompted by the program to enter a point, you can issue the X,Y,Z command and you will be prompted separately for x, y and z input values. At the prompt, you can enter these values directly by typing them at the command line, or else you can select an object snap and select a point in the drawing that has the correct value.

See also: Command X,Y  
Coordinate Point Filters

## "Y-Z"

### Zoom

The ZOOM commands change the viewing scale of the drawing, or of the portion of the drawing chosen by the user. The absolute ratios of scale of the entire drawing to the included objects chosen remains unchanged.

These variations on the ZOOM command are possible:

Zoom Window	ZOOMWIN
Zoom Back	ZOOMBACK
Zoom Fit	ZOOMFIT
Zoom In	ZOOMIN
Zoom Out	ZOOMOUT
Zoom Factor	ZOOMFAC

The ZOOM commands, like most other commands for view control, are transparent commands that can be called up and executed at the same time another command is being executed.

For example, it is possible to execute the command ZOOMOUT while at the same time entering points when drawing an element. The screen plot will be enlarged by a pre-determined factor.

```
> LINE
From point: P1
To point: P2
To point: 'ZOOMOUT
To point: P3
To point:
```

---

#### See also:

DZOOM command

PAN command

### Zoom Back



**Command: ZOOMBACK**

**Alias ZX**

**Located:**

1. **Pull-down Menu: View – Zoom Back**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The command ZOOMBACK allows you to undo the last ZOOM command and recreates the prior drawing portion. The deleted scale is saved as the last used scale and may now be called again.

Using this command allows the user to switch quickly between two definite portions in order to view an object as a whole, or to see its details more clearly.

It is still possible to recreate the last scale if other operations have been performed.

---

**See also:**

ZOOM command

Zoom Factor

**Command: ZOOMFAC**

**Alias ZF**

**Located:**

1. **Pull-down Menu: View – Zoom Factor also**  
**View – Zoom In/Out/Pan Direct – Zoom 4.00 and Zoom 0.25**
2. **Or Enter Command or Alias at the Command Line**

The command ZOOMFAC allows the user to zoom with the specified scale factor. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

Zoom to Fit



**Command: ZOOMFIT**

**Alias ZE**

**Located:**

1. **Pull-down Menu: View – Zoom Fit**
2. **Palette: Zoom**
3. **Or Enter Command or Alias at the Command Line**

The command ZOOMFIT allows the user to show the drawing with all its elements as large as possible on the viewing area.

The user should remember that points of deleted or modified elements that may still be saved in the

drawing can influence the view resulting from this command. These points will be completely deleted when a drawing is regenerated. Thus, before zooming to the largest possible drawing size, the user should consider doing a REGEN.

---

**See also:**

ZOOM command

### **Zoom In - Nearer to Objects**



**Command: ZOOMIN**

**Alias ZN**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom In**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMIN function enlarges the viewing window contents with each successive use by a factor determined by a given value. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

### **Zoom Out - Away from Objects**



**Command: ZOOMOUT**

**Alias ZT**

**Located:**

1. **Pull-down Menu: View – Zoom In/Out/Pan Direct – Zoom Out**
2. **Left SideBar**
3. **Palette: Zoom**
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The ZOOMOUT function is the opposite of the zoom-in function. It reduces the size of viewing window contents with each successive use by a factor determined by a given value. The default value for this factor can be changed using (Command VIEWPAR).

---

**See also:**

ZOOM command

## Command ZOOMPAGE

**Command:** ZOOMPAGE

**Alias** ZP

### **Located:**

1. **Pull-down Menu:** View – Zoom In/Out/Pan Direct – Zoom Page Limits
2. **Or Enter Command or Alias at the Command Line**

The **ZOOMPAGE** command zooms out to the page size defined for your printer (or plotter).

## **Zoom Window**



**Command:** ZOOMWIN

**Alias** ZW

### **Located:**

1. **Pull-down Menu:** View – Zoom Window
2. **Left SideBar**
3. **Palette:** Zoom
4. **Left Sidebar**
5. **Or Enter Command or Alias at the Command Line**

The **ZOOMWIN** command allows a chosen portion of a drawing to be shown in the largest possible scale. The drawing portion is specified by a box. Identify opposite corner points with the cursor or enter the coordinates manually.

```
> ZOOMWIN  
First corner: P1  
Second corner: P2
```

---

### **See also:**

ZOOM command

# Chapter 4 System Variables

## System Variable Chart

Note: Local Variable settings are drawing-specific and are stored in the drawing file. Global variables, however, are stored with the program and their current settings are in effect whenever the program is running. To apply other than default local variable settings to all new drawing files, the use of a template drawing is recommended in which the required setting changes has been made.

AllenCad Sysvars	Global / Local	Description
ACTDB	(G)	Returns the ID number of the drawing database currently in use.
AFLAGS	(L)	Sets attribute flags for ATTDEF bit-code
ANGBASE	(L)	Sets the base angle 0 with respect to the current UCS
ANGDIR	(L)	Sets the positive angle direction from angle 0 with respect to the current UCS
AREA	(G)	Stores the last area computed by AREA, LIST, or DBLIST
ARLINEDISP	(G)	Controls line and hatch display in the ARENDER command.
AROUNDTRIP	(G)	Controls ALLENCAD/s handling of unsupported ACAD 2000 entities
ATTDIA	(L)	Controls whether -INSERT uses a dialog box for attribute value entry
ATTMODE	(L)	Controls display of attributes
ATTREQ	(L)	Determines whether INSERT uses default attribute settings during insertion of blocks
AUNITS	(L)	Sets units for angles
AUPREC	(L)	Sets the number of decimal places for angular units
BACKZ	(L)	Stores the back clipping plane offset from the target plane for the current viewport
BLIPMODE	(L)	Controls whether marker blips are visible
BPOLYPREC	(L)	Controls boundary closure proximities for boundary polylines and boundary hatches
CECOLOR	(L)	Sets the color of new objects
CELTSCALE	(L)	Sets the current object linetype scaling factor
CELTYPE	(L)	Sets the linetype of new objects
CHAMFERA	(L)	Sets the first chamfer distance
CHAMFERB	(L)	Sets the second chamfer distance
CHAMFERC	(L)	Sets the chamfer length
CHAMFERD	(L)	Sets the chamfer angle
CHAMMODE	(L)	Sets the input method by which the program creates chamfers
CIRCLERAD	(G)	Sets the default circle radius
CIRCLERES	(G)	Circle resolution - determines the number of line segments used to display a circle
CLAYER	(L)	Sets the current layer
CMDACTIVE	(G)	Stores the bit-code that indicates whether an ordinary command, transparent command, script, or dialog box is active
CMDECHO	(G)	Controls whether the program echoes prompts and input during the LISP (command) function
CMDNAMES	(L)	Displays the names of the active and transparent commands

CURSORSIZE	(G)	Determines the size of the crosshairs as a percentage of the screen size
CVPORT	(L)	Sets the identification number of the current viewport
DATE	(G)	Stores the current date and time
DBMOD	(L)	Indicates the drawing modification status using bit-code
DEFANGBASE	(G)	Sets the default direction for zero degrees when template drawings are not used.
DEFANGDIR	(G)	Sets the default angular direction (clockwise or anticlockwise) when template drawings are not used
DEFAUNITS	(G)	Sets default angular unit format when template drawings are not used.
DEFAUPREC	(G)	Sets default angular unit accuracy when template drawings are not used.
DEFDIMZIN	(G)	Sets default dimension display mode accuracy when template drawings are not used.
DEFLUNITS	(G)	Sets default unit format when template drawings are not used.
DEFLUPREC	(G)	Sets default unit precision when template drawings are not used.
DIASAT	(G)	Stores the exit method of the most recently used dialog box
DIMALT	(L)	Controls the display of alternate units in dimensions
DIMALTD	(L)	Controls the number of decimal places in alternate units
DIMALTF	(L)	Controls the multiplier for alternate units
DIMAPOST	(L)	Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular
DIMASO	(L)	Controls the "associativity" of dimension objects
DIMASZ	(L)	Controls the size of dimension line and leader line arrowheads
DIMBLK	(L)	Sets the arrowhead block displayed at the ends of dimension lines or leader lines
DIMBLK1	(L)	Sets the arrowhead for the first end of the dimension line when DIMSAH is on
DIMBLK2	(L)	Sets the arrowhead for the second end of the dimension line when DIMSAH is on
DIMCEN	(L)	Controls drawing of circle or arc center marks and centerlines by DIMCENTER, DIMDIAMETER, and DIMRADIUS
DIMCLRDL	(L)	Assigns colors to dimension lines, arrowheads, and dimension leader lines
DIMCLRE	(L)	Assigns colors to dimension extension lines
DIMCLRT	(L)	Assigns colors to dimension text
DIMDLE	(L)	Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads
DIMDLI	(L)	Controls the spacing of dimension lines in baseline dimensions
DIMEXE	(L)	Specifies how far to extend the extension line beyond the dimension line
DIMEXO	(L)	Specifies how far extension lines are offset from origin points
DIMGAP	(L)	Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text
DIMLAYER	(L)	Sets the layer dimensions are placed on
DIMLFAC	(L)	Sets a scale factor for linear dimension measurements
DIMLIM	(L)	Generates dimension limits as the default text
DIMLINE	(L)	Determines whether the inner dimension line of linear and



		circle/arc dimensioning is forced to be drawn as single line (on) or not (off).
DIMPOST	(L)	Specifies a text prefix or suffix (or both) to the dimension measurement
DIMRND	(L)	Rounds all dimensioning distances to the specified value
DIMSAH	(L)	Controls the display of dimension line arrowhead blocks
DIMSCALE	(L)	Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets
DIMSE1	(L)	Suppresses display of the first extension line
DIMSE2	(L)	Suppresses display of the second extension line
DIMSHO	(L)	Controls redefinition of dimension objects while dragging
DIMSOXD	(L)	Suppresses drawing of dimension lines outside the extension lines
DIMSTYLE	(L)	Shows the current dimension style
DIMTAD	(L)	Controls the vertical position of text in relation to the dimension line
DIMTFAC	(L)	Sets a scale factor used to calculate the height of text for dimension fractions and tolerances
DIMTIH	(L)	Controls the position of dimension text inside the extension lines for all dimension types except ordinate
DIMTIX	(L)	Draws text between extension lines
DIMTM	(L)	When DIMTOL or DIMLIM is on, sets the minimum (or lower) tolerance limit for dimension text
DIMTMSTR	(L)	Determines an additional string for the negative value of a plus/minus tolerance to be displayed behind the dimension text
DIMTOFL	(L)	Controls whether a dimension line is drawn between the extension lines even when the text is placed outside
DIMTOH	(L)	Controls the position of dimension text outside the extension lines
DIMTOL	(L)	Appends tolerances to dimension text
DIMTP	(L)	When DIMTOL or DIMLIM is on, sets the maximum (or upper) tolerance limit for dimension text
DIMTPSTR	(L)	Determines an additional string for the +/- tolerance to be displayed behind the dimension text
DIMTSTYLE	(L)	Determines the dimension text style to be used when dimension text is created or modified
DIMTSZ	(L)	Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning
DIMTVP	(L)	Controls the vertical position of dimension text above or below the dimension line
DIMTXT	(L)	Specifies the height of dimension text, unless the current text style has a fixed height
DIMZIN	(L)	Controls the suppression of zeroes in the primary unit value
DISTANCE	(G)	Stores the distance computed by DIST
DRAGMODE	(L)	Controls the display of objects being dragged
DTCMDLINE	(G)	Controls command line display and location
DTOPTIONS	(G)	Controls display format of command options
DTPROPBAR	(G)	Controls display (on/off) of properties bar
DTSTATUSLINE	(G)	Controls display (on/off) of status line
DTTOOLAREA	(G)	Controls display (on/off) of left sidebar
DTTOOLAREASTYLE	(G)	Controls display options of left sidebar
DWGNAME	(L)	Stores the drawing name as entered by the user

DWGPREFIX	(L)	Stores the drive/directory prefix for the drawing
DWGWRITE	(L)	Specifies access mode (read-write / read only) for .dwg drawing files
DYNASNAP	(G)	Controls dynasnap symbol and tooltip display
DYNASNAPCOLOR	(G)	Controls dynasnap symbol and tooltip color
ELEVATION	(L)	Stores the current elevation relative to the current UCS for the current viewport in the current space
ENTMAKERMODE	(G)	Controls drawing regeneration
ERRNO	(G)	May contain an error number if a built-in API function could not be executed error-free
EXPERT	(G)	Controls whether certain prompts are issued
FCMULTISEL	(G)	Controls multiple object selection options
FCTEMPLATE	(G)	Used to set default template filename and path
FCVERSION	(G)	Returns the version number of the GDE based application
FILEDIA	(G)	Suppresses display of the file dialog boxes
FILLETRAD	(L)	Stores the current fillet radius
FILLMODE	(L)	Specifies whether multilines, traces, solids, all hatches (including solid-fill), and wide polylines are filled in
FONTALT	(G)	Specifies the alternate font to be used when the specified font file cannot be located
FRONTZ	(L)	Stores the front clipping plane offset from the target plane for the current viewport
GRIDMODE	(L)	Specifies whether the grid is turned on or off
GRIDUNIT	(L)	Specifies the grid spacing (X and Y) for the current viewport
GROUPSEL	(L)	Controls group selectability
HIGHLIGHT	(G)	Controls object highlighting; does not affect objects selected with grips
HLDOTTED	(G)	Determines whether highlighted objects are drawn with dotted lines during object selection or not
HPANG	(L)	Specifies the hatch pattern angle
HPASO	(L)	Turns associative hatching on/off
HPDELASO	(L)	Determines whether editing commands maintain the "associativity" of hatches
HPDOUBLE	(L)	Specifies hatch pattern doubling for user-defined patterns
HPFILE	(L)	Determines the current complete filename for the hatch pattern file.
HPNAME	(L)	Sets a default hatch pattern name
HPSCALE	(L)	Specifies the hatch pattern scale factor
HPUSRANG	(L)	Determines hatch angle of a built-in quick hatch pattern
IMPORTMSG	(G)	Provides notification of the presence of non-displaying or unsupported entities in opened, inserted or XReferenced DWG files.
INITGETFLAG	(G)	Determines whether or not INITGET requires an argument for display of keywords
INSBASE	(L)	Stores insertion base point set by BASE
INSNAME	(L)	Sets a default block name for INSERT
LANGUAGE	(G)	Determines the language used by the CAD kernel GDE
LASTPOINT	(L)	Stores the last point entered
LASTVAR	(G)	Allows the developer to retrieve the most recently called system variable
LENSLENGTH	(L)	Stores the length of the lens (in millimeters) used in perspective viewing for the current viewport
LIMCHECK	(L)	Controls the creation of objects outside the drawing limits

LIMMAX	(L)	Stores the upper-right drawing limits for the current space
LIMMIN	(L)	Stores the lower-left drawing limits for the current space
LOGFILEMODE	(G)	Specifies whether the contents of the text window are written to a log file
LOGFILENAME	(G)	Specifies the path and name of the log file
LSPALOAD	(L)	Determines a lisp filename to load automatically when a drawing is opened
LTSCALE	(L)	Sets the global linetype scale factor
LUNITS	(L)	Sets linear units
LUPREC	(L)	Sets the number of decimal places displayed for linear units
MATRIXMODE	(G)	Determines the default structure of the transformation matrix when applied in subsequent LISP functions
MIRRHATCH	(L)	Controls hatch orientation while using MIRROR or FLIP
MIRRTEXT	(L)	Controls how MIRROR reflects text
MSTOPONERR	(G)	Controls program response to errors in running macros
OFFSETDIST	(G)	Sets the default offset distance
OPENFORMATVERSION	(L)	Reports Open Format Version compatibility
ORTHOMODE	(L)	Constrains cursor movement to the perpendicular
OSMODE	(L)	Sets running object snap modes using bitcodes
PALETTE1	(G)	Returns the filename of a palette number 1 currently loaded and displayed.
PALETTE10	(G)	Returns the filename of a palette number 10 currently loaded and displayed
PALETTE2	(G)	Returns the filename of a palette number 2 currently loaded and displayed
PALETTE3	(G)	Returns the filename of a palette number 3 currently loaded and displayed
PALETTE4	(G)	Returns the filename of a palette number 4 currently loaded and displayed
PALETTE5	(G)	Returns the filename of a palette number 5 currently loaded and displayed
PALETTE6	(G)	Returns the filename of a palette number 6 currently loaded and displayed
PALETTE7	(G)	Returns the filename of a palette number 7 currently loaded and displayed
PALETTE8	(G)	Returns the filename of a palette number 8 currently loaded and displayed
PALETTE9	(G)	Returns the filename of a palette number 9 currently loaded and displayed
PANSCALE	(G)	Specifies the movement factor applied to PAN commands
PARAMETRIC	(G)	For future use – currently inactive
PARAMMODELACTIVE	(G)	For future use – currently inactive
PDMENUNAME	(G)	Returns the filename of the currently used pull-down menu
PDMODE	(L)	Controls how point objects are displayed
PDSIZE	(L)	Sets the display size for point objects
PERIMETER	(G)	Stores the last perimeter value computed by AREA, LIST, or DBLIST
PICKFIRST	(G)	Controls whether you select objects before or after you issue a command
PICKSTYLE	(L)	Controls whether groups and/or hatches are selectable
PLATFORM	(G)	Indicates which platform of the program is in use
PLINETYPE	(G)	Specifies whether the program uses optimized 2D polylines
PLINEWID	(L)	Stores the default polyline width

POLYSIDES	(G)	Sets the default number of sides for POLYGON
PREVCMD	(L)	Returns the last executed command name - even if cancelled
PSLTSCALE	(L)	Controls paper space linetype scaling
QTEXTMODE	(L)	Controls how text is displayed
REGENMODE	(L)	Controls automatic regeneration of the drawing
RINGDIA1	(G)	Specifies current default for inner diameter for the RING command
RINGDIA2	(G)	Specifies current default for outer diameter for the RING command
SAVETIME	(G)	Sets the automatic save interval, in minutes
SCREENMODE	(G)	Stores a bitcode indicating the graphics/text state of the program display
SCREENSIZE	(G)	Stores current viewport size in pixels (X and Y)
SELECTBOX	(G)	Permits sizing the of the cursor select box in pixels
SERNUMBER	(G)	Returns the serial number of the user's program license
SNAPANG	(L)	Sets snap and grid rotation angle for the current viewport
SNAPBASE	(L)	Sets the snap and grid origin point for the current viewport relative to the current UCS
SNAPBOX	(G)	Sets the dimension in pixels of the cursor snap box.
SNAPISOPAIR	(L)	Controls the isometric plane for the current viewport
SNAPMODE	(L)	Turns Snap mode on and off
SNAPSTYL	(L)	Sets snap style for the current viewport
SNAPUNIT	(L)	Sets the snap spacing for the current viewport
SPLFRAME	(L)	Controls the display of splines and spline-fit polylines
SURFTAB1	(L)	Sets the number of tabulations to be generated for RULESURF and TABSURF
SURFTAB2	(L)	Sets the mesh density in the N direction for REVSURF and EDGESURF
SURFU	(L)	Sets the surface density for PEDIT Smooth in the M direction
SURFV	(L)	Sets the surface density for PEDIT Smooth in the N direction
TABMENUNAME	(G)	Returns the filename and path of the currently used tablet menu
TEXTMINDISP	(G)	Controls the character display of text onscreen as determined by text height (in pixels)
TEXTSIZE	(L)	Sets the default height for new text objects drawn with the current text style
TEXTSTYLE	(L)	Sets the name of the current text style
THICKNESS	(L)	Sets the current 3D solid thickness
TILEMODE	(L)	Makes the Model tab or the last layout tab current
TRACEWID	(L)	Sets the default trace width
TRIMMODE	(L)	Controls whether the program trims selected edges for chamfers and fillets
UCSFOLLOW	(L)	Generates a plan view whenever you change from one UCS to another
UCSICON	(L)	Displays the UCS icon for the current viewport
UCSNAME	(L)	Stores the name of the current coordinate system for the current viewport in the current space
UCSORG	(L)	Stores the origin point of the current coordinate system for the current viewport in the current space
UCSXDIR	(L)	Stores the X direction of the current UCS for the current viewport in the current space
UCSYDIR	(L)	Stores the Y direction of the current UCS for the current viewport in the current space

UNDOCTL	(G)	Stores a bitcode indicating the state of the Auto and Control options of the UNDO command
UNITMODE	(L)	Controls the display format for units
USER1-15	(L)	Stores and retrieves integer values
USERR1-5	(L)	Stores and retrieves real numbers
USERS1-5	(L)	Stores and retrieves text string data
VIEWCTR	(L)	Stores the center of view in the current viewport
VIEWDIR	(L)	Stores the viewing direction in the current viewport
VIEWSIZE	(L)	Stores the height of the view in the current viewport
VIEWTWIST	(L)	Stores the view twist angle for the current viewport
VISRETAIN	(L)	Controls the visibility, color, linetype, lineweight, and plot styles (if PSTYLEPOLICY is set to 0) of xref-dependent layers and specifies whether nested xref path changes are saved
WORLDUCS	(L)	Indicates whether the UCS is the same as the WCS
ZINSCALE	(G)	(Zoom in scale) specifies the zoom factor for ZOOMIN command
ZOUTSCALE	(G)	(Zoom out scale) specifies the zoom factor for ZOOMOUT command

## Changing System Variables

System variables can be changed, either by entering the variable name at the command prompt and entering a new value to replace the current value indicated as default, or through the SETVAR command. Please click on the link for a detailed explanation of the latter command.

Please refer to the section "System Variables" in this Help file for an explanation of each individual system variable and its use in the program.

Below, you will find an example of the command sequence used when changing a variable value directly. For this example, the variable FILEDIA is used and we will change its value from the current 1 to value 0.

```
> FILEDIA
New value for FILEDIA <1>: 0
>
```

## "A"

### ACTDB

The variable ACTDB (Active Database) returns the ID number of the currently active drawing database that is currently active. Because up to four drawings may be opened simultaneously, this variable may contain one of the values as follows:

#### Values

-1	No drawing database opened
0	First drawing database active
1	Second drawing database active

- 2 Third drawing database active
- 3 Fourth drawing database active

#### Properties

<i>Type</i>	Integer
<i>Range</i>	-1 ... 3
<i>Default</i>	-1, because normally no drawing is opened when the program starts up.
<i>Status</i>	Read-Only

#### Commands

The variable is reset if one of the commands NEW, OPEN, or CLOSE has been executed successfully.

#### See also

Lisp functions (**flxnames ...**) and (**getactvport ...**);  
Local system variable CVPORT (Current viewpoint).

## AFLAGS

Sets bit-coded attribute flags for the ATTDEF and QATTDEF command.  
Sum of the following:

#### Bit-Codes

0	No Attribute mode set
1	Invisible
2	Constant
4	Verify
8	Preset

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0, 1, 2, 4, 8 (bit-coded)
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	Not saved

#### Commands

ATTDEF, QATTDEF.

Note: The Verify flag cannot be set within the ATTDEF command.

## ANGBASE

Determines a default value for the zero degree angle direction (Default Angle Base). As reference point for the direction serves the positive X-axis of the current user coordinate system.

#### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write

## ANGDIR

Determines the default direction when angles are specified by the user as follows:

#### Values

0	Counter-clockwise
1	Clockwise

#### Properties

<i>Type</i>	Integer
<i>Default</i>	0

*Status*            Read / Write

**Command**  
UNITSETUP

## AREA

This variable returns the area computed by the most recently executed AREA command in the current session with the program. Note: The variable is not drawing sensitive.

### Properties

*Type*            Real  
*Default*        0.0  
*Status*         Read-Only

### Commands

The variable is reset by the usage of the command AREA.

### See also

Global system variables PERIMETER and DISTANCE.

## ARLINEDISP

This global variable Controls line and hatch display in the ARENDER command.

### Values

0	Lines are not displayed in the ARENDER command
1	Lines (e.g. of hatches) are displayed in the ARENDER command

0 = Lines are not displayed in the ARENDER command

1 = Lines (E.g. of hatches) are displayed in the ARENDER command

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write

## AROUNDTRIP

Controls the switch in the OPEN dialog to preserve AutoCAD 2000 entities.

Two values for this variable are supported, where 1 = preserve AC2000 entities and 0 = off (does not preserve them).

The variable AROUNDTRIP is also evaluated in the command-line commands QOPEN, QINSERT and the INSERT command.

The default value is 1 and is stored in the cfg-file.

Note: As of the PCAD5 release date, the SAVEAS function does not fully support the correct handling of unsupported ACAD 2000 entities when "saving as" pre-AutoCAD 2000 version formats. If you experience difficulties, you may be best to use the native AutoCAD "SAVEAS" function, either to provide PCAD input, or to convert ACAD 2000-compatible PCAD5 output. Full functionality in this regard is planned for a later service release of the program.

## ATTDIA

Controls the mode for attribute requests when inserting blocks.

### Values

0	Attribute request at command line
1	Attribute request in a dialog box

### Properties

<i>Type</i>	Integer
<i>Default</i>	1
<i>Status</i>	Read / Write

### Command

The variable influences the command QINSERT that is used by LISP programmers in functions to insert parts or by advanced users to integrate part insertions into menus and palettes.

## ATTMODE

Controls the visibility of attributes in the drawing (Attribute Mode). Valid values are:

### Values

0	Hide all attributes (set to invisible)
1	Visibility as determined within the attribute definition (visible   invisible)
2	Display all attributes (visible)

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1   2
<i>Default</i>	1
<i>Status</i>	Read / Write

### Command

DRAWMODE. See also command ATTDEF for variable value 1.

## ATTREQ

Determines the mode of requests of attribute values when parts (that contain attribute definitions) are inserted. Valid values are:

### Values

0	Attribute value requests are suppressed
1	Attribute value requests performed normally

### Properties

<i>Type</i>	Integer
<i>Default</i>	1
<i>Status</i>	Read / Write

### Command

INSERT

## AUNITS

Determines the unit format for input or measurement of angles as follows:

### Values

0	Decimal Degrees
1	Degrees/Minutes/Seconds
2	Grads



3	Radians
4	Surveyor's Units

#### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read / Write

#### Command

UNITSETUP

## AUPREC

Determines the number of decimal places for angles. The angular unit precision is specified by an integer value between 0 (no decimal places) and 8 (eight decimal places. For example, 0.000000001)

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 8
<i>Default</i>	4
<i>Status</i>	Read / Write

#### Command

UNITSETUP

## "B"

## BACKZ

Back clipping plane offset. Reserved for future program releases.

#### Properties

<i>Type</i>	Real
<i>Default</i>	0.00
<i>Status</i>	Read-only
<i>Saved</i>	In the drawing file

#### See also

System variable FRONTZ

## BLIPMODE

Controls marker blip visibility. Blips are markers left on the drawing editor in the locations where left-mouse clicks have been used for selection. The flag can be set to:

#### Values

0	Marker blips are invisible (off)
1	Marker blips are visible (on)

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

## BPOLYPREC

Establishes maximum distance between boundary entities endpoints to establish a boundary contour. Used

in the command BPOLY and BHATCH.

For a variable value of 1, the maximum bridged distance between two endpoints may be 0.1 or 10 percent of the enclosing contour length.

For a variable value of 7 (maximum value), the maximum bridged distance between two endpoints may be 0.000001 or 0.00001 percent of the enclosing contour length.

## "C"

### CDATE

The variables CDATE (Current Date) return the current date and time. The calendar date returned is as follows:

YYYYMMDD.HHMMSSmm  
= Year Month Day . Hour Minute Second Millisecond

#### Properties

<i>Type</i>	Real
<i>Value</i>	Current date and time of the computer system
<i>Example</i>	19980907.12055239
<i>Status</i>	Read-Only

### CECOLOR

#### Variable CECOLOR

##### Located:

1. **Pull-down Menu: Tools – Set Color - Set Explicit Color**
2. **Or Enter Command or Alias at the Command Line**

This variable determines the current color when entities are drawn. This color assignment over-rides layer color assignments. To return to Layer control of color, use the CECOLOR – "bylayer" option.

#### Properties

<i>Type</i>	Integer
<i>Default</i>	"BYLAYER" (= 256)
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Command

'SETCOLOR

### CELTSCALE

This local variable sets the current linetype scaling factor. The default value is 1

### CELTYPE

#### Variable CELTYPE

##### Located:

3. **Pull-down Menu: Tools – Set Linetype - Set Layer's Linetype**

#### 4. Or Enter Command or Alias at the Command Line

This variable specifies the current linetype when drawing entities. The argument of the variable contains the name of the linetype.

##### Properties

<i>Type</i>	String
<i>Default</i>	"BYLAYER"
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

##### Commands

'SETLINETYPE. See also command LINETYPE.

## CHAMFERA

Determines the default value of the trim distance, in world units, associated with the first entity the user clicks on when using the edit command CHAMFER. The variable is updated when the user specifies another distance when being prompted.

##### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

##### Command

CHAMFER

##### See also

System variables CHAMFERB, CHAMFERC, CHAMFERD, CHAMMODE

## CHAMFERB

Determines the default value of the trim distance, in world units, associated with the second entity the user clicks on when using the edit command CHAMFER. The variable is updated when the user specifies another distance when being prompted.

##### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

##### Command

CHAMFER

##### See also

System variables CHAMFERB, CHAMFERC, CHAMFERD, CHAMMODE

## CHAMFERC

Stores the distance of the chamfer associated with the Angle method.

##### Properties

<i>Type</i>	Real
<i>Range</i>	0.00 or greater
<i>Default</i>	1.0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

**Command**  
CHAMFER

**See also**  
System variables CHAMFERB, CHAMFERC, CHAMFERD, CHAMMODE

## CHAMFERD

Stores the angle of the chamfer associated with the Angle method.

### Properties

<i>Type</i>	Real
<i>Range</i>	0.00 ... 359.99
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

**Command**  
CHAMFER

**See also**  
System variables CHAMFERB, CHAMFERC, CHAMFERD, CHAMMODE

## CHAMMODE

Determines whether the chamfer is created based on the input of two distances, or an angle and a distance.

### Values

0	Requires two chamfer distances (defaults retrieved from the system variables CHAMFERA and CHAMFERB)
1	Requires one chamfer distance and an angle (defaults retrieved from the system variables CHAMFERC and CHAMFERD)

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	Not saved

**Command**  
CHAMFER

**See also**  
System variables CHAMFERA, CHAMFERB, CHAMFERC, [CHAMFERD](#)

## CIRCLERAD

This variable permits setting the default value for the radius requested by the command CIRCLE (Circle Radius).

### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write
<i>Saved</i>	not saved

### Command

If the variable is set to 0.0, the command CIRCLE does not display a default value when the user is

prompted for a radius or a diameter. Any other value is displayed as default that might be accepted by the user by pressing RETURN. The variable is influenced by the command CIRCLE itself. As soon as the user has specified a radius or a diameter the variable is set to that value.

## CIRCLERES

Determines the number of line segments used to display circles and arcs in the drawings (Circle Resolution). Valid integer values lie in the range between 8 and 1024.

### Note

1. The higher the value, the more ideal the displaying of arcs and circles.
2. The lower the value, the better the performance when dynamically dragging arched objects during entity generation or selection and when redrawing or regenerating a drawing viewport.

### Viewing a circle and its segments

For technical reasons, arcs and circles cannot be shown with one curved line. They are shown as line segments adjoining each other.

CIRCLERES determines the number of arc segments used to create arcs and circles. In this way, the variable controls the apparent onscreen smoothness of curved surfaces.

A low value for the amount of arc segments used will make the arc and circles look like polygons. A higher value will make them seem round. The standard value for arched segments is 256. The user may exchange this value for any value between 8 and 1024 arched segments in the edit box.

### Properties

<i>Type</i>	Integer
<i>Range</i>	8 ... 1024
<i>Default</i>	256
<i>Status</i>	Read / Write
<i>Saved</i>	Saved in the CFG file

### Command

The variable may be modified by the user via the dialog command DRAWMODE in the *Circle Segments* area.

## CLAYER

Determines the Current Layer. The argument of the variable contains the name of the layer.

### Properties

<i>Status</i>	Read / Write
<i>Type</i>	String
<i>Default</i>	"0"
<i>Saved</i>	In the drawing file

### Command

'SETLAYER, LAYER.

## CMDACTIVE

This variable (Command Active) allows the developer to evaluate the type of a currently active command. A bit code is returned with a meaning as follows:

### Values

Bit 1	Standard command
Bit 2	Standard and transparent command
Bit 4	Macro script

### Properties

<i>Type</i>	Integer (Bit-Code)
<i>Status</i>	Read-Only

## CMDECHO

This variable (Command Echo On/Off) allows the developer to set a preference for the mode a statement using the LISP function (**command ...**) is performed as follows:

0	The command sequences are not visible for the user
1	The command sequences are visible for the user

### Properties

<i>Type</i>	Integer
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Saved</i>	not saved

### Command

Not influenced by a built-in command.

Programming: See function (**command ...**).

## CMDNAMES

This variable (Command Names) allows the developer to retrieve in his routines the name(s) of the currently active command(s).

### Properties

<i>Type</i>	String
<i>Examples</i>	"LINE", "LINE 'ZOOM"
<i>Status</i>	Read-Only

### Example

> LINE

From point: 'ZOOM

Zoom scale factor: (progn (setq x (getvar "cmdnames")) (princ x)(princ))

LINE'ZOOM

## CURSORSIZE

Sets the size of the crosshairs as a percentage of the screen. By default it is set to 5 (5% of the screen). If the system variable is set to 100, the crosshairs cross the entire screen.

<i>Type</i>	Integer
<i>Range</i>	1 ... 100
<i>Default</i>	5
<i>Status</i>	Read / Write
<i>Saved</i>	Configuration (CFG)

## CVPORT

This variable allows the LISP programmer to retrieve the identification number of the Current Viewport.

### Properties

<i>Status</i>	Read-Only
<i>Type</i>	Integer
<i>Default</i>	1

**See also**

Lisp functions (**getactvport**) and (**setactvport**)

**"D:****DATE**

This variable returns the current date and the current time in the *Julian Date* format.

**Properties**

<i>Type</i>	Real
<i>Status</i>	Read-Only
<i>Example</i>	2449968.64956018

**See also**

Global system variable CDATE.

**DBMOD**

This variable allows the programmer to returns a bit-coded modification status of the drawing database.

**Properties**

<i>Status</i>	Read-Only
<i>Type</i>	Bit-Code

**DEFANGBASE**

If no template drawing is used when the user creates a new drawing, this variable (Default Angle Base) allows setting a default value for the zero degree angle direction. As reference point for the direction serves the positive X-axis of the current user coordinate system.

**Properties**

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write
<i>Saved</i>	Saved in the system's CFG file

**Command**

UNITSETUP

**DEFANGDIR**

If no template drawing is used when the user creates a new drawing, this variable (Default Angle Direction) sets the default direction when angle are specified by the user as follows:

**Values**

0	Counter-clockwise
1	Clockwise

**Properties**

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

**Command**

UNITSETUP

## DEFAUNITS

If no template drawing is used when the user creates a new drawing, this variable (Default Angular Units) allows the programmer to determine the unit format to enter or to measure angles as follows:

### Values

0	Decimal Degrees
1	Degrees/Minutes/Seconds
2	Grads
3	Radians
4	Surveyor's Units

### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 4
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	in the system's CFG file

### Command

UNITSETUP

## DEFAUPREC

If no template drawing is used when the user creates a new drawing, this variable (Default Angular Units Precision) determines the standard setting for the number of decimal places to be displayed when prompting the user for angle specifications. A setting of up to eight decimal places is valid.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 8
<i>Default</i>	4
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

### Command

UNITSETUP

## DEFDIMZIN

If no template drawing is used when the user creates a new drawing, this variable (Default Dimension Zero Inches) determines the display mode of dimensions.

### Note:

The meaning of the integer values (in the range between 0 and 16) is documented in the description of the local system variable DIMZIN.

### Properties

<i>Type</i>	Integer
<i>Type</i>	0 ... 16
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

## DEFLUNITS

If no template drawing is used when the user creates a new drawing, this variable (Default Linear Units) allows the programmer to determine the standard setting for linear unit format used in the drawing as follows:



1	Scientific
2	Decimal
3	English: Engineering
4	English: Architecture
5	Fraction
<i>/4Type</i>	Integer
<i>Range</i>	1 ... 5
<i>Default</i>	2
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

**Command**  
UNITSETUP

## DEFLUPREC

If no template drawing is used when the user creates a new drawing, this variable (Default Linear Units Precision) allows the programmer to determine the standard setting for the number of decimal places of linear units. A value up to 8 decimal places is accepted.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 8
<i>Default</i>	2
<i>Status</i>	Read / Write
<i>Saved</i>	in the system's CFG file

**Command**  
UNITSETUP

## DIASTAT

This variable (Dialog Status) allows the programmer to retrieve the status, how the most recently dialog box of a built-in GDE command called by the user has been exited. Allows you to retrieve the exit status of a dialog when inside a LISP program a built-in dialog-command has been called.

### Values

0	Dialog has been terminated by OK
1	Dialog has been terminated by Cancel

### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read-Only

### See also

Global system variables CMDNAMES, CMDACTIVE, LASTVAR.

## DIMALT

0 = OFF (default) | 1 = ON

Disables or enables alternate unit dimensioning.

## DIMALTD

0 .. 8 (Default: 2)

Determines the number of decimal places for alternate unit dimensioning if the variable DIMALT is set to 1 (see above).

## DIMALTF

Real (Default: 1.0)

Determines the scale factor for alternate unit linear dimensioning if the variable DIMALT is set to 1 (see above).

## DIMAPOST

String (Default: "")

Determines a string appended to the measurement of an alternate unit dimension if the variable DIMALT is set to 1 (see above). Valid for all dimensions except angular.

## DIMASO

0 = OFF | 1 = ON (default)

Disables or enables associative dimensioning.

If associative dimensioning is turned on, all dimensions are created as anonymous block and can be modified and updated as a unique object. Also, the dimension is associated to its defining points in the geometry.

If associative dimensioning is turned off, all entities of the dimension are created as single lines, arrows, text, etc. An association to the geometry is not maintained.

## DIMASZ

Real (Default: ...)

Determines the size of arrows at the end of dimension lines.

## DIMBLK

String (Default: "")

Determines the name of an arrow symbol block replacing the normal arrow at the ends of dimension lines.

## DIMBLK1

String (Default: "")

Determines the name of an arrow symbol block replacing the normal arrow at the starting point of a dimension line, if DIMSAH is set to 1 (see below).

## DIMBLK2

String (Default: "")

Determines the name of an arrow symbol block replacing the normal arrow at the ending point of a dimension line, if DIMSAH is set to 1 (see below).

## DIMCEN

Real (Default: ...)

Determines the size (in drawing units) of center marks when circles or arcs are dimensioned with one of the commands DIMCENT, DIMRAD, or DIMDIA. Positive values determine that only a center mark is drawn. Negative values determine that centerlines are drawn.

## DIMCLRD

0 .. 256

Determines the color number assigned to dimension lines, arrowheads, and dimension line leaders.

## **DIMCLRE**

0 .. 256

Determines the color number assigned to dimension extension lines.

## **DIMCLRT**

0 .. 256

Determines the color number assigned to dimension text.

## **DIMDLE**

Real

Determines the length of the extension of the dimension line past the extension line(s) when ticks are replacing the normal arrowheads.

## **DIMDLI**

Real

Determines the offset distance of dimension lines when base line dimensioning is applied.

## **DIMEXE**

Real

Determines a value for the length the end of extension lines exceeds the dimension line.

## **DIMEXO**

Real

Determines an offset of extension line starting point from the measured entity.

## **DIMGAP**

Real

Determines and offset distances around the dimension text to be maintained when the dimension label is located in between the dimension line.

## **DIMLAYER**

String (Default: "")

Determines the layer dimensions will be placed on, if the variable contains a string matching an existing layer name. A null string ("" ) or non-entry specification determines that the current layer will be used for placing dimension objects.

## **DIMLFAC**

Real (Default: 0.00 = not used)

Forces all linear dimensioning measurements (including diameter, radius, and ordinate dimensioning) to be multiplied with a factor specified by this dimension variable and generates dimension text containing the result of that multiplication. DIMLFAC has no effect on angular dimensioning. The factor is used as a

multiplier only in the case that DIMLFAC is non-zero. The DIMLFAC variable is especially applicable when dimensioning scaled details.

## Linear Dimensioning

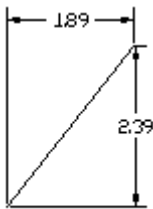


Command: DIMLIN

Alias DL

### Located:

1. Pull-down Menu: Dim – Linear
2. Palette: Dim
3. Or Enter Command or Alias Directly at the Command Line

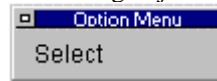


Linear

The DIMLIN command creates a horizontal or vertical dimensioning depending on the position of the dimension line. This command is a combination between both orthogonal modes horizontal and vertical.

To use this command, define two points of measurement (DEFPOINTS).

The program also offers the user the chance to select a specific dimensioning object function from the



Options Bar. Activate this function by choosing the Select button. Lines, polylines, circles and arcs may now be selected. Point to the desired object with the selection cursor. The start and end points, or the diameter of the object, will be interpreted automatically and used for the measurement calculations.

Once the points of measurement have been defined, the user will be asked to choose a position for the dimensioning line.

The distance between the two points is measured along the X-axis for horizontal and Y-axis for vertical dimensioning and displayed.

Pressing the RETURN key will verify the calculated value and insert it into the drawing.

```
> DIMLIN
1. Point: P1
2. Point: P2
Dimension line positioning: P3
Dimension text <2.50>: Return
```

or

```
> DIMLIN
1. Point: Select
Select line, polyline, circle, arc: P1
Dimension line positioning: P2
Dimension text <2.50>: Return
```

## DIMLINE

0 = OFF (default) | 1 = ON

Determines whether the inner dimension line of linear and circle/arc dimensioning is forced to be drawn as single line (on) or not (off). The DIMLINE variable is especially applicable when dimensioning diameters and radii.

## DIMPOST

String (Default: "")

Determines a text prefix, suffix or both, for dimension label created with following dimension commands. These strings surround the dimension measurement value computed. The string <> used in the variable setting serves to separate prefix and suffix specification; otherwise the string supplied is used as suffix. Example: A variable value of "Approx. <> m" might result in the dimension text *Approx. 12 m*

## DIMRND

Real (Default: 0.00)

Determines rounding of all dimension measurements to the setting of this variable. For example, a DIMRND value set to 0.5 results to round all dimensions represented in the dimension text to the next 0.5 value.

## DIMSAH

0 = OFF (default) | 1 = ON

Determines whether separate arrowhead symbol blocks are allowed (on) or not (off). The arrow blocks used are specified by the variable settings of DIMBLK1 and DIMBLK2 (see above).

## DIMSCALE

Real (Default: 1.0)

Determines a general scale factor applied on all scalar dimension variables (e.g. text size, arrow size, tick size, extension line offset, ...).

**Note:** This variable does *not* take effect on the measured values (lengths, angles, or coordinates) by the linear, radial, angular, and ordinate dimensioning commands.

## DIMSE1

0 = OFF | 1 = ON (default)

Determines whether the first extension line is displayed (1) or not (0).

## DIMSE2

0 = OFF | 1 = ON (default)

Determines whether the second extension line is displayed (1) or not (0).

## DIMSHO

0 = OFF | 1 = ON (default)

Determines whether the dynamic dragging mode is used when creating or modifying dimensions. Note: This variable is not stored in a dimension style.

## DIMSOXD

0 = OFF (default) | 1 = ON

Suppresses dimension lines drawn outside the extension lines.

## DIMSTYLE

Contains the name of the current dimension style that can be set with the DIMTYPE command.

## DIMTAD

0 = OFF (default) | 1 = ON

Determines, if set to 1 (on), to place dimension text above the dimension line. Then a single solid dimension line is drawn beneath the dimension text.

If activated, the variable takes effect,

1. If the dimension text is drawn between the extension line (see DIMTIX) and at the same angle as the dimension line; or
2. If the dimension text is placed outside extension lines.

If DIMTAD is set to zero (off) the vertical location of dimension text in relation to the dimension line is controlled by the DIMTVP variable (see below).

## DIMTFAC

Real

Determines a scale factor for the text height of tolerance values relative to the general dimension text height.

## DIMTIH

0 = OFF | 1 = ON (default)

Forces dimension text to be drawn horizontally in linear, diameter, or radius dimensioning, when the dimension text is placed between the extension lines. When set to 0 (off), the angle of the dimension text takes the angle of the dimension line.

## DIMTIX

0 = OFF (default) | 1 = ON

Forces dimension text to be drawn in between the extension lines.

## DIMTM

String

Determines a value for the negative value of a plus/minus tolerance to be drawn. The tolerance value is only drawn if either DIMTOL or DIMLIM is set to 1(on).

## DIMTMSTR

String

Determines an additional string for the negative value of a plus/minus tolerance to be displayed behind the dimension text. See also: DIMTPSTR.

## DIMTOFL

0 = OFF | 1 = ON

Forces, if set to 1 (on), a dimension line to be drawn between the extension lines even when the dimension text is placed outside the extension line.

For dimension and radius dimensioning the variable has an additional meaning. If at the same time the variable DIMTIX is turned off (0) and DIMTOFL is turned on (1), the dimension line and the arrowheads are drawn inside the circle or arc, but the leader line and the dimension text are placed outside the circle or arc.

## DIMTOH

0 = OFF | 1 = ON (default)

Forces dimension text to be drawn horizontally in linear, diameter, or radius dimensioning, when the dimension text is located outside the extension lines. When set to 0 (off), the angle of the text outside of extension lines takes the angle of the dimension line.

## DIMTOL

When switched on (value 1) this local variable shows dimension tolerances. It works in conjunction with DIMTP (the positive tolerance), and DIMTM (the minus tolerance) settings  
See also DIMTPSTR and DIMTMSTR for addition of string characters.

## DIMTP

String

Determines a value for the positive value of a plus/minus tolerance to be drawn. The tolerance value is only drawn if either DIMTOL or DIMLIM is set to 1(on).

## DIMTPSTR

String

Determines an additional string for the positive value of a plus/minus tolerance to be displayed behind the dimension text. See also: DIMTMSTR.

## DIMTSTYLE

Determines the text style used when dimension text is created or modified. The string must contain a valid text style name already stored in the STYLE table of the drawing database. A text style can be loaded, defined, or re-defined with the command FONT. If the variable is set to a null string (""), the current text style specified in the local variable TEXTSTYLE (see above) is used.

## DIMTSZ

Real (Default: 0.0)

Determines the length of ticks, if tick symbols are replacing arrowheads.

If the variable setting of DIMTSZ is 0, arrows are drawn.

If the variable value is > 0 ticks are drawn with the size resulting from the product of BEMSLG\*BEMFKTR.

## DIMTVP

Real

Permits adjustment of the vertical offset of the dimension text in relation to the dimension line. The dimension text can be placed above or below the dimension line. The magnitude of the vertical offset of the dimension text is the product of text height and the current variable setting: DIMTXT\*DIMTVP.

The DIMTVP value is only used, if DIMTAD is set to off (0).

## DIMTXT

Real

Determines the height of dimension text in dimension labels by a floating-point number greater than zero, unless the current dimension text style has a fixed height.

## DIMZIN

0, 1, 2, 3, 4, 8, 12

Determines the display mode of dimension text to suppress leading and/or trailing zeros in dimensions. The integer value affects dimension text as follows:

- 0 = Suppress zero feet and suppress precisely zero inches
- 1 = Display zero feet and display precisely zero inches
- 2 = Display zero feet, but suppress zero inches
- 3 = Suppress zero feet, but display zero inches
- 4 = Suppress leading zeros in all decimal dimensions
- 8 = Suppress trailing zeros in all decimal dimensions
- 12 = Suppress leading and trailing zeros in all decimal dimensions

**Note:** The values 0 through 3 affect feet/inch dimensioning only. The values 4, 8, and 12 are applied on decimal dimensions.

## DISTANCE

This variable (Distance) returns the value of a length respectively distance measurement most recently performed by a call of the command 'DIST.

### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read-Only

### Commands

The variable is reset if the user has performed the command 'DIST.

Note: This variable is drawing-sensitive.

## DRAGMODE

Controls the display of entities as they are dragged.

### Values

When set to 1 or 2, the program displays an outline of the entities as they are dragged.

When set to 0, dragging is turned off.

### Properties

<i>Type</i>	Integer
<i>Default</i>	2
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Command

'SETTINGS

## DTCMDLINE

The global system variable DTCmdLine controls the desktop position of the command line window.

**Note**



Dependencies on the FelixCAD initialization-file and TopArea entry:  
If you switch the DTCmdLine off (0), the TopArea entry in the initialization-file is set to 0 (off). If you select top (1) or bottom (2) display, the TopArea entry is automatically switched to 1 (on).

#### Values

0 Off  
1 Top  
2 Bottom

#### Properties

*Type* Integer

*Range* 0 ... 2

*Default* Depends on the FelixCAD initialization-file setting of CmdLine

*Status* Read / Write (saved in the system's INI file)

*Command* DESKCONF

*See also* The global system variables DTOptions, DTPropBar, DTToolArea, DTToolAreaStyle and DTStatusLine

## DTOPTIONS

The DTOptions global system variable allows display control of input options for commands. You have the choice between no option display, an option bar and an option menu.

#### Values

0 No option bar and option menu support  
1 Command option menu  
2 Command option bar

#### Properties

*Type* Integer

*Range* 0 ... 2

*Default* Depends on the AllenCad initialization file setting of OptionMenu

*Status* Read / Write (saved in the system's INI file)

*Command* DESKCONF

*See also* The global system variables DTCmdLine, DTPropBar, DTToolArea, DTToolAreaStyle and DTStatusLine

## DTPROPBAR

The global system variable DTPropBar controls display of the property information bar. The property bar contains the current layer name, color, linetype and the drawing aid toggles for orthogonal, snap and grid mode.

#### Values

0 Off  
1 On

#### Properties

*Type* Integer

*Range* 0 ... 1

*Default* Depends on the AllenCad initialization-file setting of PropBar

*Status* Read / Write (saved in the system's INI file)

*Command* DESKCONF

*See also* The global system variables DTCmdLine, DTOptions, DTToolArea, DTToolAreaStyle and DTStatusLine

## DTSTATUSLINE

The status line at the bottom of the drawing window can be controlled with the global system variable `DTStatusLine`. The status line displays the current coordinate system in X-, Y- and Z-coordinates as well as the process bar for opening and saving of drawings.

### Values

0 Off  
1 On

### Properties

*Type* Integer

*Range* 0 ... 1

*Default* Depends on the AllenCad initialization-file setting of `StatusLine`

*Status* Read / Write (saved in the system's INI file)

*Command* DESKCONF

*See also* The global system variables `DTCmdLine`, `DTOptions`, `DTPropBar`, `DTToolArea` and `DTToolAreaStyle`

## DTTOOLAREA

The global system variable `DTToolArea` controls the tool area on the left-hand side of the drawing window (the Left Sidebar). If the `DTToolArea` variable is switched on, the left-hand tool area will be displayed using the last mode set in the `DTToolAreaStyle` setting.

### Values

0 Off  
1 On

### Properties

*Type* Integer

*Range* 0 ... 1

*Default* Depends on the AllenCad initialization-file setting of `ToolArea`

*Status* Read / Write (saved in the system's INI file)

*Command* DESKCONF

*See also* The global system variables `DTCmdLine`, `DTOptions`, `DTPropBar`, `DTToolAreaStyle` and `DTStatusLine`

## DTTOOLAREASTYLE

The global system variable `DTToolAreaStyle` switches between five different display modes available for the Left Sidebar. The `DTToolAreaStyle` variable depends on the setting of `DTToolArea`, which controls the display of the left tool area.

### Values

0 0 Column  
1 1 Column  
2 2Columns  
3 3Columns  
4 3Columns (Old tool area style of FelixCAD 3.1 and earlier)

### Properties

*Type* Integer

*Range* 0 ... 4

*Default* Depends on the AllenCad initialization-file setting of `ToolAreaStyle`

*Status* Read / Write (saved in the system's INI file)

*Command*            DESKCONF

*See also* The global system variables DTCmdLine, DTOptions, DTPropBar, DTToolArea and DTStatusLine

## DWGNAME

Returns the complete name of the current drawing as string.

### Properties

<i>Status</i>	Read-Only
<i>Type</i>	String
<i>Default</i>	"C:\SUPPORT\noname_0"

## DWGPREFIX

Returns the drive and the path of the current drawing.

### Properties

<i>Status</i>	Read-Only
<i>Type</i>	String
<i>Default</i>	"C:\SUPPORT\"

## DWGWRITE

Specifies the access mode to (.DWG) drawings, and allows protection of DWG files when these should only be displayed. Valid values are:

### Values

0	Read-Only
1	Read/Write

### Properties

<i>Type</i>	Integer
<i>Status</i>	Read / Write

## DYNASNAP

Controls Dynasnap mode.

Type: Integer

Saved in: cfg-file

Default value: 3

Controls DynaSnap marker and tooltip. The system variable value is the sum of the following bit values:

- |   |   |
|---|---|
| 0 | Turns off the DynaSnap marker and tooltips. |
| 1 | Turns on the DynaSnap marker                |
| 2 | Turns on the DynaSnap tooltips              |
| 3 | Turns on marker and tooltips                |

Note: The values 4, 8, 16 and 32 are not supported and only used for AutoCAD compatibility purposes.

## DYNASNAPCOLOR

This variable controls the color displayed for Dynasnap symbols and tooltips.

## "E"

### ELEVATION

Specifies the value for an elevation (Z axis) in the current user coordinate system. This might be used as temporary construction plane.

#### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write

#### See also

System variable THICKNESS and INSBASE

### ENTMAKERMODE

The global system variable ENTMAKERMODE suppresses regeneration of the drawing, when entities are created with the LISP- or FDT- entmake-function. After the creation of entities, the developer has to call the regen command to make all changes visible in the drawing. The global system variable ENTMAKERMODE is not saved and is set at the beginning of a drawing session to 1.

#### Note

The regen / regenall command sets this system variable back to the default value 1

#### Values

0	Off
1	On

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 1
<i>Default</i>	1 (On)
<i>Status</i>	Read / Write (not saved)
<i>Command</i>	LISP : entmake FDT : fdt_entmake

*See also* The C-API function fdt\_entmake and the LISP-function entmake

### ERRNO

This variable (Error Number) may contain an error number set by the system if a built-in API function could not be executed error-free.

#### Properties

<i>Status</i>	Read-Only
<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read-Only

### EXPERT

This variable allows the developer to set on or off a so-called Expert Mode for his file operations.

#### Values

0	Requests may display warnings or security prompts in some cases
1	Requests do not display warnings or security prompts

#### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	Not saved

## "F"

### FCMULTISEL

Supports the previous style of entity selection as used in Release 3.1 and earlier. AllenCad commands like erase, move, etc. select more than one entity if the global system variable is set to 1. The current value of the system variable FCMULTISEL is saved in the ALLENCAD.CFG file. If FCMULTISEL is switched on, all found entities at the pick point will be selected.

#### Values

0	Off
1	On

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 1
<i>Default</i>	0 (Off) Only single selection. Compatible to the AutoCAD selection mode.
<i>Status</i>	Read / Write (saved in the AllenCad configuration file AllenCad.CFG)
<i>Command</i>	All commands, which select entities like ERASE, MOVE etc.

### FCTEMPLATE

This variable allows the programmer to set the default name (including drive and path) for the template drawing used when the user executes the command NEW. No template drawing is pre-set, if a null string ("") is specified.

#### Properties

<i>Type</i>	String
<i>Example</i>	"C:\SUPPORT\template.flx"
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

#### Command

Note: The user has the possibility to specify another template drawing or to disable the usage of a template drawing with the command NEW. The user's specification is written to the system variable FCTEMPLATE.

### FCVERSION

This variable returns the version number of the Graphic Developer's Engine based application.

#### Properties

<i>Type</i>	String
<i>Example</i>	"4.0"

*Status*            Read-Only

## FILEDIA

This variable (File Dialog) may be set or evaluated by the developer to determine, if for certain commands a dialog box or a command line request sequence is used.

### Values

0	Dialog Box version of the command is used
1	Command prompts parameters at the command line

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Status</i>	Not saved

## FILLETRAD

Determines the default value for the Fillet Radius when the command FILLET is used.

### Properties

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write

### Command

FILLET

## FILLMODE

Controls the fill mode for Polyline and 2DFACE entities. Valid values are:

### Values

0	Display entities not filled
1	Display entities filled

### Properties

<i>Status</i>	Read / Write
<i>Type</i>	Integer
<i>Default</i>	1

### Commands

2DFACE, POLYLINE, CHAIN, NGON, RING

## FONTALT

This global variable stores the name of the font to be used whenever a drawing file calls for an unknown font.

Note: If no font is specified for this variable, no substitution will be made for fonts specified in a drawing file that cannot be found on the current computer. Such text will not be visible onscreen.

## FRONTZ

Front clipping plane offset. Reserved for future program releases.

**Properties**

<i>Type</i>	Real
<i>Default</i>	0.00
<i>Status</i>	Read-only
<i>Saved</i>	In the drawing file

**See also**

System variable BACKZ

## "G"

## GRIDMODE

Turns grid display on or off. The dots of the grid are displayed in the XY distances as specified by the system variable GRIDUNIT and in the area specified by the system variables LIMMIN and LIMMAX.

**Values**

0	Off
1	On

**Properties**

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

**Command**

'PRECPAR

## GRIDUNIT

Specifies the horizontal and vertical distances (x, y) between the grid dots when grid display is turned on (see system variable GRIDMODE).

**Properties**

<i>Type</i>	2D point
<i>Default</i>	0.0,0.0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

**Command**

'PRECPAR

**See also**

System variable [GRIDMODE](#)

## GROUPSEL

Determines whether groups are selectable or not. The system variable changes the group's selectable status as follows:

**Values**

0	Enables temporarily editing on entities in a group.
1	Disables the editing of grouped entities. Returns the selectable status to work with the entire group again.

**Properties**

<i>Type</i>	Integer
-------------	---------

<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file (unique to PCAD)

**Command**  
GROUP

## "H"

### HIGHLIGHT

This variable is used to determine if selected drawing objects are marked, by highlighting them.

#### Values

0	Off
1	On

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Status</i>	Not saved

### HLDOTTED

Determines whether highlighted objects are drawn with dotted lines during object selection or not.

#### Values

0	Highlighted objects are drawn with continuous lines
1	Highlighted objects are drawn with dotted lines

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Writeable
<i>Saved</i>	Configuration (CFG)

#### Note:

The system variable is applied in any object selection.

### HPANG

Determines the hatch angle of a pattern file based hatch pattern. See the variables HPFILE, HPNAME, and HPSCALE.

#### Properties

<i>Type</i>	Real
<i>Default</i>	0
<i>Status</i>	Read / Write

**Command**  
HATCH

### HPASO



Turns associative hatching on and off.

**Values**

0	Off
1	On

**Properties**

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

**Commands**

HATCH, QHATCH, BHATCH

**See also**

System variable HPDELASO

## HPDELASO

Determines whether editing commands maintain the "associativity" of hatches.

**Values**

0	The "associativity" is maintained.
1	The "associativity" gets lost when an entity that belongs to a contour defining a hatch boundary is displaced.

**Properties**

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file (unique to AllenCad)

**Commands**

HATCH, QHATCH, BHATCH

Editing commands MOVE, ROTATE, SCALE, etc.

**See also**

System variable HPASO

## HPDOUBLE

Determines if the built-in quick hatch pattern is drawn with single lines or with crossing lines. Valid values for this flag are:

**Values**

0	Single line hatch pattern
1	Double line hatch pattern (cross)

**Properties**

<i>Status</i>	Read / Write
<i>Type</i>	Integer
<i>Default</i>	0

**Command**

HATCH

## HPFILE

Determines the current complete filename for the hatch pattern file. If the variable contains a null string ("") no hatch pattern file is set to allow pattern selection in the Hatch dialog box.

**Properties**

<i>Type</i>	String
<i>Example</i>	"C:\SUPPORT\ansi.pat"
<i>Status</i>	Read / Write

**Command**

[HATCH](#)

## HPNAME

Determines the default name of a hatch pattern based on a hatch pattern file. If the variable contains a null string ("") the hatching commands uses the built-in quick pattern for hatching (see HPSPACE, HPANGLE, and HPDOUBLE).

**Properties**

<i>Type</i>	String
<i>Default</i>	""
<i>Status</i>	Read / Write

**Command**

[HATCH](#)

## HPSCALE

Determines the scale factor for a hatch pattern based on a hatch pattern description file.

**Properties**

<i>Type</i>	Real
<i>Default</i>	1.0
<i>Status</i>	Read / Write

**Command**

[HATCH](#)

## HPSPACE

Determines the distance between the lines of the standard quick pattern. See also HPNAME, HPDOUBLE, HPUSRANG.

**Properties**

<i>Type</i>	Real
<i>Default</i>	1.0
<i>Status</i>	Read / Write

**Command**

[HATCH](#)

## HPUSRANG

Determines the hatch angle of a built-in quick hatch pattern (single line or cross pattern; see also the system variable HPDOUBLE).

**Properties**

<i>Type</i>	Real
<i>Default</i>	0
<i>Status</i>	Read / Write

**Command**

[HATCH](#)

# "IJK"

## IMPORTMSG

The global IMPORTMSG variable if set to 1 (on) will display an alert box listing all non-displayed and unsupported entities that are present in an opened, inserted, or XReferenced DWG drawing file. It works in conjunction with the global AROUNDTRIP variable, which must also be set to 1 (on) for this notification to be given.

Besides the on-screen alert box notification, a report file (importdwg.rpt) is generated in the Windows\Temp directory. This file will be re-written each time a new alert is issued. Switching the IMPORTMSG value to 0 (off) will disable these notifications.

## INITGETFLAG

Determines whether the Lisp function *initget* requires the bit-coded flag value 256 as argument for initialization of the display of keywords in the context bar, or not.

### Values

0	The bit value 256 must be provided in the flag argument explicitly to have the options defined in the keyword string argument being displayed in the context bar respectively the context menu (default).
1	The keywords specified as second argument of <i>initget</i> are displayed automatically in the context bar respectively the context menu.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	Not saved

### Related

Lisp function **initget**.  
FDT function **fdt\_initget()**.

### Note

The system variable has been added for the reason that an application wants to maintain one source code applicable in various CAD programs, but also wants the feature of utilizing the context bar for keywords activated. If you want so, set the global variable INITGETFLAG to 1.

## Command INSBASE

Applies a constant offset to the base (insertion) point for parts.

Notes:

3. A value for INSBASE other than 0,0,0 will apply the designated coordinate adjustments to the insertion base point specified when inserting a part.
4. The command PARTEXP does not request a base point when creating a drawing file representing a part when the entity selection method is used.

### Properties

<i>Type</i>	3D point
<i>Default</i>	0.00,0.00,0.00

*Status*            Read / Write

#### **Command**

PARTEXP. See also PARTDEF and ELEVATION

## INSNAME

Determines a default part name for the INSERT command. The variable is updated when the user specifies another part in the INSERT dialog. If the variable contains a null string (""), no default part name is set.

#### **Properties**

*Type*            String  
*Default*        ""  
*Status*           Read / Write

#### **Command**

INSERT

## "L"

## LANGUAGE

This variable enables the determination of the language used by the CAD kernel Graphic Developer's Engine (GDE). Currently two values are accepted:

#### **Values**

1                German  
2                English

#### **Properties**

*Type*            Integer  
*Default*        1 (read from INI file)  
*Status*           Read / Write  
*Saved*          Not saved

## LASTPOINT

Returns the coordinates of the point specified by the last point input within a drawing or editing command function. May also be set.

#### **Properties**

*Type*            3D Point  
*Default*        x, y, z  
*Status*           Read / Write

## LASTVAR

This variable (Last Variable) allows the developer to retrieve the most recently called system variable.

#### **Properties**

*Type*            String  
*Example*        "PDMENUNAME"  
*Status*           Read-Only

## LENSLENGTH

This local variable stores the focal length of the lens (in millimeters) used in perspective viewing for the current drawing. In this, it acts much as a viewing zoom factor.

## LIMCHECK

Limit Checking.

This system variable is not used in AllenCad by built-in commands or functions. Applications may use it. The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Note

In other CAD programs utilizing the *.dwg* and *.dxf* drawing file format this variable may control whether entities can be created outside the drawing limits specified by LIMMIN and LIMMAX.

### See also

System variables LIMMIN and LIMMAX.

## LIMMAX

Determines the upper right XY coordinates for the display of a grid in the drawing. May also, in some cases, be interpreted as drawing sheet corner.

### Properties

<i>Type</i>	2D Point
<i>Default</i>	12.0, 9.0
<i>Status</i>	Read / Write

### Command

---

## LIMMIN

Determines the lower left XY coordinates for the display of a grid in the drawing. May also, in some cases, be interpreted as drawing sheet corner.

### Properties

<i>Type</i>	2D Point
<i>Default</i>	0.0, 0.0
<i>Status</i>	Read / Write

### Command

---

## LOGFILEMODE

This system variable controls log file generation mode in AllenCad. After typing in LOGFILEMODE at the command prompt, the user can respond to the program prompt with a value of either 1 to enable logging mode or 0 to disable logging mode.

#### Values

0	No log file active. (off)
1	File logging (LOGFILEON) is active. (on)

## LOGFILENAME

This system variable establishes the default file name and path for log files. After typing in LOGFILENAME at the command prompt, the user can enter a new file name and path for recorded log files.

## LSPALOAD

Determines the file name of a LISP file that is loaded automatically when the drawing is opened. Recommended for use especially in template drawings to perform certain application specific tasks (e.g. evaluation of application specific configuration files).

#### Properties

<i>Type</i>	String
<i>Default</i>	""
<i>Status</i>	Read / Write

## LTSCALE

Determines a common scale factor for linetypes. This permits display adjustment of dashed, dash-dotted, or dotted lines to the drawing and plotting scale.

The dash lengths specified in the linetype definition (in drawing units) are multiplied with the factor specified in the variable LTSCALE. The variable accepts floating-point values that must be greater than zero.

#### Properties

<i>Type</i>	Real
<i>Default</i>	1.0
<i>Status</i>	Read / Write

#### Command

'DRAWMODE. See also: LAYER, LINETYPE, 'SETLINETYPE.

## LUNITS

Determines the mode for the linear unit format used in the drawing. Valid values are:

#### Values

1	Scientific
2	Decimal
3	English: Engineering
4	English: Architecture
5	Fraction

#### Properties

<i>Type</i>	Integer
<i>Range</i>	1 ... 5
<i>Default</i>	2
<i>Status</i>	Read / Write

**Command**  
UNITSETUP

## LUPREC

This variable (Linear Units Precision) determines the number of decimal places of linear units. A value up to 8 decimal places is accepted.

### Properties

<i>Type</i>	Integer
<i>Default</i>	2
<i>Status</i>	Read / Write

**Command**  
UNITSETUP

## "MN"

## MATRIXMODE

Determines the default structure of the transformation matrix when applied in subsequent Lisp functions.

### Values

0	Standard matrix structure (default)
1	Alternate matrix structure (AutoCAD-like)

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read/Write (not saved)

### Related

Lisp functions **nentselp** and **nentsel**.  
FDT function **fdt\_xformss()**, **fdt\_draggen()**, **fdt\_grvecs()**, **fdt\_nentsel()**, **fdt\_nentselp()**.

## MIRR Hatch

By default, the MIRROR and FLIP commands mirror selected hatches. Set this system variable to 0 to keep the hatch in its original orientation.

### Values

0	Retain hatch direction when mirrored
1	Mirror hatch

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file (unique to PCAD)

**Command**  
MIRROR, FLIP

**See also**  
System variable MIRRTEXT

## MIRRTEXT

This variable (Mirror Text) determines, if text is reflected or retains direction when selected by the MIRROR or FLIP command.

**Values**

0	Mirror text
1	Retain text direction when mirrored

**Properties**

<i>Type</i>	Integer
<i>Default</i>	1
<i>Status</i>	Read / Write

**Command**

MIRROR, FLIP

**See also**

System variable MIRR Hatch

## MSTOPONERR

This variable controls how the program responds to errors encountered when running macros. Valid values are 0 and 1, where 1 will halt the macro when an error is encountered and zero will allow the macro to continue, but with an error message being sent to the text screen.

## "O"

## OFFSETDIST

This variable (Offset Distance) sets the default value for the command OFFSET when prompting the user for the distance between the original objects (contour) and the copy of the objects.

**Properties**

<i>Type</i>	Real
<i>Default</i>	0.0
<i>Status</i>	Read / Write
<i>Saved</i>	Not saved

**Command**

The variable is re-written when the user executes the command OFFSET.

## OPENFORMATVERSION

This local variable simply returns the version number of the Open Drawing Alliance Library that is currently being used by AllenCad.

## ORTHOMODE

This variable turns the orthogonal drawing mode on or off.

**Values**

0	Off
1	On

**Properties**

<i>Type</i>	Integer
-------------	---------



*Default*        0  
*Status*        Read / Write

#### Commands

PRECPAR, TORTH0

## OSMODE

This variable (Object Snap Mode) sets an explicit object snap mode for drawing and editing. To set more than one object snap mode explicitly, the sum of the following valid value must be specified. Valid values for the bit-coded argument of the system variable are:

Value	Meaning
0	No explicit object snap
1	End point
2	Mid point
4	Center point of arcs and circles
8	Insertion point of text, blocs, attribute definitions and anonymous blocks like dimensioning and hatching
16	Quadrant of arcs and circles
32	Intersection point
64	Insertion point
128	Perpendicular to
256	Tangent to
512	Next point on nearest entity

#### Properties

*Status*        Read / Write  
*Type*         Integer  
*Default*       0

#### Command

[PRECPAR](#)

## "P"

### PALETTE1 ... PALETTE $n$

These variables return the filename of a palette number  $n$  currently loaded and displayed on the desktop. The variable returns in a string the filename including drive and path. If no palette has been loaded indexed as  $n$ , the variable returns a null string ("").

#### Properties

*Type*         String  
*Default*       ""  
*Status*        Read-Only  
*Example*       "C:\APX\AP\_DRAW.MNP"

#### Commands

PALETTE, PALMAN, MENU

## PANSCALE

This variable (Pan Scale) specifies the factor for the PAN commands 'PANRIGHT, 'PANLEFT, 'PANUP, and 'PANDOWN, that move the view of the current viewport up or down or to the left or right.

#### Properties

*Type*         Real

<i>Default</i>	0.5
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

#### **Commands**

This variable may be set by the user with the DISPPAR command.

## PARAMETRIC

This variable is intended for future use in the program and is inactive in AllenCad

## PARAMMODELACTIVE

This variable is intended for future use in the program and is inactive in AllenCad

## PDMENUNAME

This variable (Pull-down Menu Name) returns the filename including drive and path of the currently used pull-down menu.

#### **Properties**

<i>Type</i>	String
<i>Status</i>	Read-Only
<i>Example</i>	"C:\SUPPORT\SUPPORT.mnu"

#### **Command**

MENU

## PDMODE

Determines the symbol used to display point entities in the drawing.

#### **Properties**

<i>Type</i>	Integer
<i>Range</i>	0 ... 20
<i>Default</i>	0
<i>Status</i>	Read / Write

#### **Command**

DRAWMODE

#### **See also**

System variable PDSIZE

## PDSIZE

Determines the size of point entities in the drawing.

#### **Properties**

<i>Status</i>	Read / Write
<i>Type</i>	Real
<i>Default</i>	0.0

#### **Command**

DRAWMODE

#### **See also**

System variable PDMODE

## PERIMETER

This variable returns the value of the perimeter computed by the most recently usage of the command AREA.

### Properties

<i>Type</i>	Real
<i>Initial</i>	0.0
<i>Value</i>	
<i>Status</i>	Read-Only

### Command

AREA

### See also

Global system variable AREA.

## PICKFIRST

With the global variable PICKFIRST system variable, you specify whether you wish to select objects before or after you issue a command that uses the object selection set. Values of zero and one are valid for this variable.

0 - Turns off Variable PICKFIRST; you select objects after you issue a command

1 - Turns on Variable PICKFIRST; you select objects before you issue a command

## PICKSTYLE

This local variable controls whether groups and/or hatches are selectable

Values

0	Neither are selectable
1	Groups only are selectable
2	Associative hatches only are selectable
3	Both groups and hatches are selectable

## PLATFORM

This variable returns a string to identify the operating system platform of the Graphic Developer's Engine (GDE).

### Properties

<i>Type</i>	String
<i>Value</i>	"Win32"
<i>Status</i>	Read-Only

## PLINETYPE

This system variable controls polyline format, both in the creation of new polylines with the PLINE command, as well as conversion of existing polylines. Both standard format and lightweight (LWPOLYLINE) format polylines are supported. Valid settings for this variable are as follows:

- 0 Polylines in older drawings are not converted when opened; PLINE creates old format polylines
- 1 Polylines in older drawings are not converted when opened; PLINE creates optimized polylines
- 2 Polylines in older drawings are converted when opened; PLINE creates optimized polylines

This command is entered at the command line and the program will prompt with the currently set value for the variable. You may enter a new value or accept the current value by simply pressing the Enter key.

## PLINEWID

Determines the default value for the width when polylines are drawn with one of the commands POLYLINE, CHAIN, NGON, or RECTANGLE.

### Properties

<i>Status</i>	Read / Write
<i>Type</i>	Real
<i>Default</i>	0.00

### Commands

POLYLINE, CHAIN, NGON, RECTANGLE

## POLYSIDES

This variable (Polygon Sides) allows setting the default value for the command NGON when prompting the user to enter the number of sides of the regular polygon to be drawn. The value of the integer can lie in the range between 3 and 1024.

### Properties

<i>Type</i>	Integer
<i>Range</i>	3 ... 1024
<i>Default</i>	4
<i>Status</i>	Read / Write

### Command

The setting is updated when the command NGON is used.

## PREVCMD

Returns the most recently called or executed command in a string even if the command has been canceled. The variable returns both a built-in command of the graphics engine and a command provided by a so-called "c:" function defined within the user interface. If the most recently called command has been a lisp expression, like (setq a 4.7), or an unknown command, the variable contains the string "PREVCMD". The variable is of interest only for programming.

### Properties

<i>Status</i>	Read-Only
<i>Type</i>	String
<i>Example</i>	"_line"

## PSLTSCALE

Paper Space Linetype Scaling.

This system variable is not used in AllenCad by built-in commands or functions. Applications may use it. The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Note

In other CAD programs that use the *.dwg* and *.dxf* drawing file format, this variable may control whether viewport scaling governs linetype scaling (1) or no special linetype scaling (0) is applied in paper space.

## "Q"

### QTEXTMODE

#### Quick Text Mode

This system variable is not used in AllenCad by built-in commands or functions. Applications may use it. The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Note

In other CAD programs using the *.dwg* and *.dxf* drawing file format, this variable may toggle Quick Text Mode on and off: When the Quick Text value is set to 1 (on), text entities are replaced by rectangular boxes. When the Quick Text Mode value is set to off 0 (off), text displays normally.

## "R"

### REGENMODE

Toggles on and off automatic regeneration of the drawing.

#### Values

0	When the value is set to on, the drawing automatically regenerates.
1	When the value is set to off, the drawing will not regenerate automatically.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### See also

Lisp functions **tblmake**, **tblmod**

## RINGDIA1

This variable (Ring Diameter 1) specifies for the built-in command RING the preference value for the inner diameter.

### Properties

<i>Type</i>	Real
<i>Default</i>	0.5
<i>Status</i>	Read / Write
<i>Saved</i>	not saved

### Command

The variable is reset if the command RING has been used.

## RINGDIA2

This variable (Ring Diameter 2) specifies for the built-in command RING the preference value for the outer diameter.

### Properties

<i>Type</i>	Real
<i>Default</i>	1.0
<i>Status</i>	Read / Write
<i>Saved</i>	Not saved

### Command

The variable is probably reset if the command RING has been used.

## "S"

## SAVETIME

Sets the time for auto-saving of drawings in minutes.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0 ... 60
<i>Default</i>	30
<i>Status</i>	Read / Write
<i>Saved</i>	In INI file

### Command

CONFIG > *Common* tab

## SCREENMODE

This variable returns a flag to evaluate, if the *Lisp/History* window is active.

### Values

0	Lisp/History text window is active
1	Lisp/History text window is closed

### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read-Only

### Note

The text window is not displayed "on top" when a new session with the program is started.

### Command

The command ``TSCREEN` re-sets this variable. The variable is only of interest for LISP programming, that allows setting the History/Lisp text window on top with the function (**textscr**) and to remove the text window with the functions (**graphscr**) to not overlap the drawing windows.

## SCREENSIZE

This variable (Screen Size) returns the size of the current viewport of the current drawing in pixels (X, Y). The value 0,0 indicates, that no drawing is opened.

### Properties

<i>Type</i>	Screen coordinates (in pixels)
<i>Example</i>	550,283
<i>Status</i>	Read-Only

### Note

The commands ``WTILE` and ``WCASCADE` as well as the Windows operations Full Screen, Maximize, Minimize, etc. lead to a reset of the system variable.

## SELECTBOX

This variable allows retrieving and setting the size (in pixels) of the Select Box cursor that is used when prompting the user to select objects.

### Properties

<i>Type</i>	Integer
<i>Default</i>	5
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

### Commands

The transparent command `'PRECPAR` allows the user to specify an appropriate value for his current needs.

## SERNUMBER

This variable returns the serial number of the user's program license. The serial number is also displayed in the dialog displayed by the ``INFO` command.

### Properties

<i>Status</i>	Read-Only
<i>Type</i>	String

## SNAPANG

Determines the rotation angle for the snap grid for the current viewport. The rotation angle is set relative to the current UCS.

### Properties

<i>Type</i>	Real
<i>Range</i>	0.00 ... 360.00
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Command

`'PRECPAR`. `'SETTINGS > Snap/Grid` tab.

### See also

System variables SNAPMODE, SNAPBASE, SNAPSTYL, GRIDMODE

## SNAPBASE

Determines the origin (x,y) for the snap grid.

### Properties

<i>Status</i>	Read / Write
<i>Type</i>	2D Point
<i>Default</i>	0.00,0.00

### Command

'PRECPAR

## SNAPBOX

This variable (Snap Cursor Box) sets the size of the snap area in pixels for object snap functions.

### Properties

<i>Type</i>	Integer
<i>Default</i>	5
<i>Status</i>	Read / Write
<i>Saved</i>	In the CFG file

### Commands

The transparent command 'PRECPAR allows the user to specify an appropriate value for his current needs.

## SNAPISOPAIR

Sets the current isometric plane for the isometric drawing style. Has effect only if SNAPSTYL is set to isometric (see below).

### Values

0	Sets the crosshairs to the left plane (default)
1	Sets the crosshairs to the top plane
2	Sets the crosshairs to the right plane

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1   2
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

'PRECPAR 'SETTINGS > Snap/Grid tab.

### See also

System variables [SNAPSTYL](#), [SNAPMODE](#), [SNAPBASE](#), [GRIDMODE](#)

## SNAPMODE

Determines the snap mode. If snap mode is turned on, the cursor movements and the coordinates identified are locked to the nearest point on the snap grid. If turned on, this mechanism is disabled.

### Values

0	Off
1	On



### Properties

<i>Status</i>	Read / Write
<i>Type</i>	Integer
<i>Default</i>	0

### Commands

[PRECPAR](#), 'TSNAP

## SNAPSTYL

Toggles the snap style for the current viewport between standard and isometric drawing style. The style setting can be:

### Values

0	Standard (default)
1	Isometric

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

[PRECPAR](#) ['SETTINGS > Snap/Grid tab.](#)

### See also

System variables SNAPISOPAIR, SNAPMODE, SNAPUNIT, SNAPBASE, GRIDMODE

## SNAPUNIT

Determines the horizontal and vertical distance of the points on the snap grid.

### Properties

<i>Type</i>	2D Point
<i>Default</i>	1.00, 1.00
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Command

[PRECPAR](#)

## SPLFRAME

Determines how to display the edges of 3DFACE entities, that have been set to invisible are currently displayed. The value may be as follows:

### Values

0	Edges of 3D Faces are displayed as defined (visible or invisible)
1	All edges of 3D Faces are displayed visible

### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read / Write

### Commands

'VIEWPAR. See also: 3DFACE.

## SURFTAB1

Sets the number of tabulations used in the RULESURF and TABSURF commands and the M-direction mesh density for the REVSURF command.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	2 ... 32766
<i>Default</i>	6
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Commands

REVSURF, RULESURF, TABSURF

## SURFTAB2

Sets the N-direction mesh density for the REVSURF command.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	2 ... 32766
<i>Default</i>	6
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

#### Commands

RULESURF,

## SURFU

Sets the default surface density in the M direction (primary direction) for polyface mesh editing.

This system variable is not used in AllenCad by built-in commands or functions. Application may use it.

The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	2 ... 200
<i>Default</i>	6
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

## SURFV

Sets the default surface density in the N direction (secondary direction) for polyface mesh editing.

This system variable is not used in AllenCad by built-in commands or functions. Applications may use it.

The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

#### Properties

<i>Type</i>	Integer
<i>Range</i>	2 ... 200
<i>Default</i>	6
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

## "T"

## TABMENUNAME

This variable (Tablet Menu Name) returns the filename including drive and path of the currently used tablet menu. It returns a null string ("" ) if no tablet menu is used.

### Properties

<i>Type</i>	String
<i>Status</i>	Read-Only
<i>Example</i>	"C:\SUPPORT\aptable.mnt"

### Commands

MENU, TABLET

## TEXTMINDISP

This global variable controls the character display of text onscreen as determined by text height (in pixels)

### Values

0	Displays all text, regardless of onscreen text height
1 to 20	Sets the upper limit in pixels at which text is represented onscreen by a straight line

### Properties

<i>Type</i>	Integer
<i>Range</i>	0 to 20
<i>Default</i>	0
<i>Status</i>	Read / Write

## TEXTSIZE

Determines the default value of the text size proposed by the command TEXT or ATTDEF for entity creation.

### Properties

<i>Status</i>	Read / Write
<i>Type</i>	Real
<i>Default</i>	0.2

### Commands

TEXT, ATTDEF

## TEXTSTYLE

Determines the default value for the text style proposed by the command TEXT or ATTDEF for entity creation. The string must contain a valid text style name already stored in the STYLE table of the drawing database. A text style can be loaded, defined, or re-defined with the command FONT.

### Properties

<i>Type</i>	String
<i>Default</i>	"STANDARD"
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

TEXT, ATTDEF. See also: FONT.

## THICKNESS

**System Variable - Thickness**

## Alias TH

### Located:

1. **Pull-down Menu: Tools – Entity Extrusion - Set Thickness or No Thickness**
2. **Or Enter Command or Alias at the Command Line**

Sets the value for the default thickness (extrusion). Specifying the value 0.00 means that no thickness is applied.

### Properties

<i>Type</i>	Real
<i>Range</i>	Any positive or negative number
<i>Default</i>	0.00
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### See also

System variable ELEVATION

## TILEMODE

Toggles on or off tilemode, allowing you to access paper space and model space with floating viewports. When set to a value of 1 (on), you have access to model space with tiled viewports.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

In Paper Space (Layout Mode): Command VIEWPORT

## TRACEWID

Sets the value for the default trace width. This system variable is not used in AllenCad by built-in commands or functions. Applications may use it. The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

### Properties

<i>Type</i>	Real
<i>Range</i>	0.00 or greater
<i>Default</i>	0.05
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Properties

AllenCad has neither a TRACE entity nor a TRACE command. When reading *.dwg* or *.dxf* files that contain TRACE entities, AllenCad converts them to 2DFACE entities, that are identical to traces.

## TRIMMODE

Determines the trim mode used when executing the command FILLET or CHAMFER as follows

### Values

0	OFF: The selected lines to be filleted or chamfered are not modified
1	ON: The selected lines are trimmed when filleted or chamfered

**Properties**

<i>Type</i>	Integer (0   1)
<i>Default</i>	1
<i>Status</i>	Read / Write

**Command**

EDITPAR, FILLET, CHAMFER

## "U"

### UCSFOLLOW

**Variable UCSFOLLOW****Alias UF****Located:**

1. **Pull-down Menu: Tools – Let View Follow UCS**
2. **Or Enter Command or Command Alias Directly at the Command Line**

Determines, if the alteration of the user coordinates system (UCS) causes an automatic change to the plan view of the new UCS.

**Values**

0	Off: A new UCS does not change the view
1	On: Plan view follows automatically to an UCS alteration

**Properties**

<i>Type</i>	Integer (0   1)
<i>Default</i>	0
<i>Status</i>	Read / Write

**Command**

'VIEWPAR

## UCSICON

This global variable controls the visibility of the UCS icon at the lower left side of your drawing editor. Valid values for this variable are 0 and 1. When the variable is set to 0 the icon is not displayed whereas with 1 it is displayed.

## UCSNAME

Returns the name of the current user coordinate system (UCS). If the variable is a null string (""), the UCS is unnamed.

**Properties**

<i>Type</i>	String
<i>Default</i>	""
<i>Status</i>	Read-Only

## UCSORG

Returns the coordinates of the origin of the current user coordinate system (UCS).

**Properties**

<i>Type</i>	3D Point
<i>Default</i>	0.00,0.00,0.00
<i>Status</i>	Read-Only

## UCSXDIR

Returns the coordinates for the direction of the X axis of the current user coordinate system.

**Properties**

<i>Type</i>	3D Point
<i>Default</i>	1.00,0.00,0.00
<i>Status</i>	Read-Only

## UCSYDIR

Returns the coordinates for the direction of the Y axis of the current user coordinate system.

**Properties**

<i>Type</i>	3D Point
<i>Default</i>	0.00,1.00,0.00
<i>Status</i>	Read-Only

## UNDOCTL

This variable sets the status for the function to undo steps performed by a function or command. Also, it returns the current default value. The setting is bit-coded as follows:

**Bit Codes**

Bit 1	Undo disabled
Bit 2	Undo all elements
Bit 3	Undo the last element or the last group
Bit 4	Undo Messages On/Off

**Properties**

<i>Type</i>	Integer (bit coded)
<i>Default</i>	2
<i>Status</i>	Read / Write

## UNITMODE

Toggles on and off the display of certain formats for units (fractional, feet-and-inches, and surveyor's angles) in the status line.

This system variable is not used in AllenCad by built-in commands or functions. Applications may use it. The system variable has been added for compatibility reasons. The system variable will be written to *.dwg* and *.dxf* files as is currently set.

**Properties**

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

## USERI1 ... USERI5

Stores a user-defined integer in the one of the variables USERI1 through USERI5.

### Properties

<i>Type</i>	Integer
<i>Default</i>	0
<i>Status</i>	Read / Write

## USERR1 ... USERR5

Stores a user-defined floating-point number in the one of the variables USERR1 through USERR5.

### Properties

<i>Type</i>	Real
<i>Default</i>	0.00
<i>Status</i>	Read / Write

## USERS1 ... USERS5

Stores a user-defined string number in the one of the variables USERS1 through USERS5.

### Properties

<i>Type</i>	String
<i>Default</i>	""
<i>Status</i>	Read / Write

## "V"

## VIEWCTR

Returns the coordinates (of the current user coordinate system) of the center point of the current viewport.

### Properties

<i>Type</i>	2D Point
<i>Default</i>	x, y
<i>Status</i>	Read-Only

## VIEWDIR

Returns the coordinates (of the current user coordinate system) containing the view direction of the current viewport.

### Properties

<i>Type</i>	3D Point
<i>Default</i>	0.00,0.00,1.00
<i>Status</i>	Read-Only

## VIEWSIZE

Returns the height in drawing units of the visible drawing portion (view) in the current viewport.

### Properties

<i>Type</i>	Real
-------------	------

<i>Default</i>	9.00
<i>Status</i>	Read-Only

## VIEWTWIST

Returns the view angle to the current viewport.

### Properties

<i>Status</i>	Read-Only
<i>Type</i>	Real
<i>Default</i>	0.0

## VISRETAIN

Controls layer properties of externally referenced drawings (XREFs) as follows:

### Values

0	Externally referenced drawings take on the layer settings (on or off, thawed or frozen, etc.) in the current drawing.
1	The layer settings of externally referenced drawings take precedence. This implies that these settings persist each time you open a drawing with the Xref.

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read / Write
<i>Saved</i>	In the drawing file

### Commands

Commands XLINK and QXLINK

## "W"

## WORLDUCS

Indicates, if the current user coordinate system (UCS) is identical with the world coordinate system (WCS). In this case the variable contains the integer value 1; otherwise 0.

### Values

0	UCS does not correspond to WCS
1	UCS is identical to the WCS

### Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	1
<i>Status</i>	Read-Only

## "XYZ"

## ZINSCALE



This variable (Zoom In Scale) specifies the zoom factor for the transparent command ZOOMIN.

**Properties**

<i>Type</i>	Real
<i>Default</i>	0.5
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

**Commands**

The user may set a new value by using the command VIEWPAR.

## ZOUTSCALE

This variable (Zoom Out Scale) specifies the zoom factor for the transparent command ZOOMOUT.

**Properties**

<i>Type</i>	Real
<i>Default</i>	2.0
<i>Status</i>	Read / Write
<i>Saved</i>	In the system's CFG file

**Commands**

The user may set a new value by using the command VIEWPAR.

# Chapter 5 AllenCad for AutoCAD Users

## Introducing AllenCad

For more information on purchasing Power products, or to enroll in the Power Developer Network™, call (800) 258-6360 (no technical support available at this number) or visit [www.ADSI.com](http://www.ADSI.com).

### Introduction

Based on core FCAD® technology, AllenCad® has evolved through several major releases to become one of the world's best values in flexible, powerful, and industry-compatible computer-aided design processing. From its inception in 1994, AllenCad™ has been designed to offer exceptionally compatibility with AutoCAD® by Autodesk Inc.

The objective of AllenCad Pro™ is to deliver nearly 100% of the functionality of industry leader AutoCAD in as much as 90% of use cases; for less the 20% of the price. AllenCad Pro utilizes the latest OpenDWG® file exchange libraries allowing direct operation with billions of existing drawings saved in industry standard AutoCAD DWG® or DXF® formats.

AllenCad Pro offers extensive customization and programming extension capabilities. AllenCad Pro is available as the affordable and completely scalable PowerENGINE™ Software Development System, whereby all, or selected components can be embedded into custom solutions of your own design. Solutions being built using PowerENGINE range from traditional AEC, GIS, and manufacturing applications to non-traditional business engineering applications such as automated sales and wireless stock trading systems.

By default, AllenCad installs with its own user interface, desktop layout, commands, and command aliases. Each of these components can be substituted with optional AutoCAD emulation components to enhance system operation for experienced AutoCAD users. In addition, AllenCad includes a number of utilities to edit or extend each user interface and desktop layout component to best suit your needs.

Many millions of dollars have been spent to ensure unparalleled compatibility with AutoCAD® in these crucial areas:

File exchange

Functionality

User interface

Desktop layout

Command execution

Customization and programming techniques

The sections that follow describe significant AutoCAD compliance and compatibility issues in these specific areas, and are intended primarily for AutoCAD-familiar operators and/or programmers.

## AllenCad/ AutoCAD Functionality Comparison

### AllenCad / AutoCAD Functionality Comparison

AllenCad Pro supports over 300 compatible AutoCAD commands and variables. By design, AllenCad supports the most commonly used AutoCAD function and commands. In addition to support for most common AutoCAD functions, AllenCad adds numerous additional commands and functions to improve productivity. By design, we have concentrated, on improving basic functionality as it applies to most CAD applications, with the needs of the vast majority of our users foremost in mind.

**Primary AutoCAD functions not presently supported include:**

AllenCad R5 does not yet support ACIS™ solids object creation or editing, although all conventional 3D "primitives" are supported (eg. 3D faces, polylines, etc.). A solids modeling extension is planned for future releases.

AllenCad R5 does not support AutoCAD 3DOrbit® functions, although a number of advanced 3D visualization tools are include (eg. Auto-render, Render, and Dview).

AllenCad R5 does not yet support direct import or export of 3D Studio® files. Users are asked to use the DWG and/or DXF drawing format is using 3D Studio.

AllenCad R5 does not support AutoCAD ASE® (AutoCAD SQL Extension). Users wishing to utilize external database linking must use PowerENGINE, the developer-specific version of AllenCad which allows sophisticated external database linking and handling using SQL2-ODBC.

Detailed charts outlining supported, unsupported, and equivalent AutoCAD functions, commands, and system variables are provided at the end of this section.

## Equivalent AutoCAD 2000 / AllenCad Pro Commands

<b>AutoCAD 2000 Command</b>	<b>AllenCad Pro Command</b>	<b>Alias</b>	<b>Command Description</b>
3D	3DOBJECTS (Application)	3O	Creates three-dimensional polygon mesh objects
3DFACE	3DFACE	3F	Creates a three-dimensional face
3DMESH	3DMESH	3M	Creates a free-form polygon mesh
3DPOLY	POLY3D	3P	Creates a polyline of straight line segments in three-dimensional space
'ABOUT	'ABOUT	AB	Displays information about the Program
'APPLOAD	'APPLOAD (Application)	AP	Loads and unloads applications and defines which applications to load at startup
ARC	ARC	A	Creates an arc
AREA	AREA	AA	Calculates the area and perimeter of objects or of defined areas
ARRAY	ARRAY	AR or RA	Creates multiple copies of objects in a pattern
ATTDEF	ATTDEF	AD	Creates an attribute definition
ATTEDIT	ATTEDIT	ATT	Modify part attributes, one by one
ATTEDIT	QATTEDIT	ATE or QAE	Changes attribute information independent of its block definition (Command line version)
ATTEXT	'ATTEXP (Application)	ATX	Extracts attribute data

BHATCH	BHATCH	BH	Fills an enclosed area with a hatch pattern
BLOCK	PARTDEF	B or PDF	Define a part (block)
BMAKE	BMAKE	BK	Defines a block using a dialog box
BMPOUT	BMPOUT	BPO	Saves selected objects to a file in device-independent bitmap format
BOUNDARY	BOUNDARY	BD or BO	Creates a boundary polyline (command line version of BPOLY)
BOX	3DBOX	3B	Creates a three-dimensional box
BPOLY	BPOLY	BP	Creates a region or a polyline from an enclosed area
BPOLY	PCONTOUR	PC	Creates a boundary polyline on top of other entities
BREAK	BREAK	BR	Erases parts of objects or splits an object in two
BREAK	DELPARTIAL	DPL	Delete part of an entity
CHAMFER	CHAMFER	CHA	Bevels the edges of objects
CHANGE	CHANGE	C or CV	Changes the properties of existing objects
CHANGE	CHANGE	C or CV	Changes the properties of existing objects
CHANGE	CHPROP	CHP	Changes the properties of existing objects
CHANGE	PROPEdit	PED	Changes the properties of existing objects
CHPROP	CHPROP	CHP	Changes the color, layer, linetype, linetype scale factor, and thickness of an object
CIRCLE	CIRCLE	CI	Creates a circle
CLOSE	CLOSE	CL	Exit the drawing but not close the program
CLOSE	WCLOSE	WC	Closes the current viewport
COMPILE	See DWFCOMPILE, FCOMPILE, OEMFCOMPILE		Compiles shape files and PostScript font files (Compiles FelixCAD fnt font files and shp font files)
CONE	3DCONE (APPLICATION)	3DC	Creates a three-dimensional solid cone
CONFIG	CONFIG	CFG	Configure program operating parameters
CONFIG	DESKCONF	DSK	Configures the desktop options
COPY	COPY	C or CO	Duplicates objects
COPYCLIP	COPYSELECT	CST	Copies objects to the Clipboard
CUTCLIP	CUTSELECT	CT	Copies objects to the Clipboard and erases the objects from the drawing
CYLINDER	3DCONE	3DC	Creates a three-dimensional solid cylinder
DDATTE	ATTXEDIT	AX	Edits the variable attributes of a block
DDATTEXT	ATTEXP	ATX	Extracts attribute data
DDCHPROP	PROPEdit	PED	Changes the color, layer, linetype, and thickness of an object
DDCOLOR	'SETCOLOR	STC	Sets the color for new objects
DDEDIT	TEXTEDIT	ED or TE	Edits text and attribute definitions
DDIM	DIMDRAW (Application)	DDR	Creates and modifies dimension styles
DDIM	DIMTYPE	DTP	Creates and modifies dimension styles
DDINSERT	INSERT	I	Inserts a block or another drawing
'DDPTYPE	DRAWMODE	DMD	Specifies the display mode and size of point objects
'DDPTYPE	SETTINGS	OP, RM, PR ST or DMD	Specifies the display mode and size of point objects
'DDRModes	SETTINGS	OP, RM, PR or ST	Sets drawing aids
'DDSELECT	SETTINGS	OP, RM, PR	Sets object selection modes

DDUCS	UCS_C	or ST	
'DDUNITS	SETUP	UCC	Manages defined user coordinate systems
	(Application)	US	Sets up drawing units
'DDUNITS	UNITSETUP	UN or US	Sets up drawing units
DDVIEW	VIEW	V	Creates and restores views
DDVPOINT	3DVIEW	3V	Sets the three-dimensional viewing direction
DIM (Option)	DIMHOR	DH	Horizontal dimensioning
DIM (Option)	DIMROT	DRO	Rotate existing dimensions
DIM (Option)	DIMUPD	DU	Updates dimension
DIM (Option)	DIMVER	DVR	Labels dimensions along the Y (vertical dimension) axis
DIMALIGNED	DIMALI	DAL	Creates an aligned linear dimension
DIMANGULAR	DIMA3P	D3P	Creates an angular dimension
DIMANGULAR	DIMA4P	D4P	Creates an angular dimension out of four definition points
DIMBASELINE	DIMBAS	DBA	Continues a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension
DIMCENTER	DIMCENT	CM or DCE	Creates the center mark or the center lines of circles and arcs
DIMCONTINUE	DIMCON	DCN or DCO	Continues a linear, angular, or ordinate dimension from the second extension line of the previous or a selected dimension
DIMDIAMETER	DIMDIA	DDA or DDI	Creates diameter dimensions for circles and arcs
DIMEDIT	DIMEDIT	DED	Edits dimensions
	(Application)		
DIMEDIT (Option H)	DIMTHOME	DTH	Undoes the movement of dimension text
DIMEDIT (Option)	DIMOBL	DOB	Modifies the orientation of existing linear dimensioning objects.
DIMLINEAR	DIMLIN	DL or DLI	Creates linear dimensions
DIMORDINATE	DIMORD	DOD or DOR	Creates ordinate point dimensions
DIMRADIUS	DIMRAD	DRA	Creates radial dimensions for circles and arcs
DIMSTYLE	DIMTYPE	DTP	Creates and modifies dimension styles on the command line
_DIMSTYLE	DIMREST	DRE	Reinstate a stored style file so that it can be reinstated as the current style
_DIMSTYLE	DIMSAVE	DS	Allows the user to combine the individual parameters and store them in a named dimensioning style
DIMTEDIT	See DIMEDIT		Moves and rotates dimension text
	DIMTROT & DIMTMOVE		
DIMTEDIT (Option A)	DIMTROT	DTR	Rotate dimension text
'DIST	'DIST	D or DI	Measures the distance and angle between two points
DIVIDE	DIVIDE	DIV	Places evenly spaced point objects or blocks along the length or perimeter of an object
	(Application)		
DONUT	RING	DO or RI	Draws filled circles and rings
DONUT	RING	DO or RI	Draws filled circles and rings
DRAWORDER	DRAWORDER	DR	Changes the display order of images and other objects

DSETTINGS	SETTINGS	OP, RM, PR or ST	Specifies settings for Snap mode, grid, and polar and object snap tracking
DTEXT	DTEXT	DT	Displays text on screen as it is entered
DVIEW	DVIEW	DV	Defines parallel projection or perspective views
DXFIN	DXFIN	DXN	Imports a drawing interchange file
DXFOUT	DXFOUT	DXO	Creates a drawing interchange file of the current drawing
EDGE	EDGE (Application)	EG	Changes the visibility of three-dimensional face edges
'ELEV	'ELEVATION	ELE	Sets elevation properties of new objects
'ELEV (Sysvar)	See 'THICKNESS (Sysvar)	TH	Sets extrusion thickness properties of new objects
ELLIPSE	ELLIPS	EL	Creates an ellipse or an elliptical arc
ERASE	DELETE and ERASE	DEL	Removes objects from a drawing
ERASE	ERASE	E	Removes objects from a drawing
EXIT	EXIT	Q or EXIT	Exit the program
EXPLODE	EXPLODE	X	Breaks a compound object into its component objects
EXPLODE	XPLODE Application)	X	Breaks up complex objects into component entities
EXPORT	See SAVEAS BMPOUT WMFOUT PARTEXP BATPROC		Saves objects to other file formats
EXTEND	EXPAND	EX, ET or EXP	Extends an object to meet another object
FILLET	FILLET	F	Rounds and fillets the edges of objects
'GRAPHSCR	'GRAPHSCR	GS	Switches from the text window to the drawing area
'GRID	PRECPAR	OS or PCP	Displays a dot grid in the current viewport
GRID – Option On/Off	'TGRID	TG	Toggles grid display on/off
GROUP	GROUP	G	Creates a named selection set of objects
HATCH	HATCH	H	Fills a specified boundary with a pattern
HATCHEDIT	HPEDIT	HE or HED	Modifies an existing hatch object
'HELP (F1)	HELP (F1)	?	Displays online help
HIDE	FHIDE	FHD	Creates a view in the current window.
'ID	'ID	ID	Displays the coordinate values of a location
IMAGE	RLOAD	RL	Inserts images in many formats into an AutoCAD drawing file
IMAGEFRAME	RBORDER	RB	Controls whether the image frame is displayed on the screen or hidden from view
IMPORT	See OPEN		Imports various file formats into AutoCAD
INSERT	INSERT	I	Places a named block or drawing into the current drawing
INTERSECT	INTERSECT	IN	Selects a location at the theoretical intersection of two linear entities
'LAYER	LAYER	LA	Manages layers and layer properties
LENGTHEN	LENGTHEN	LEN	Lengthens an object
LIGHT	LIGHT	LIT	Manages lights and lighting effects
LIGHT (Option Modify)	LIGHTEDIT	LE	Modify light source settings

'LIMITS	'LIMITS (Application)	LM	Sets and controls the drawing boundaries
LINE	LINE	L	Creates straight line segments
'LINETYPE	LINETYPE,	LT or LTYPE	Creates, loads, and sets linetypes
'LINETYPE	LOADLTYPE	LLT	Loads linetypes
'LINETYPE	LOADLTYPE	LLT	Creates, loads, and sets linetypes
LIST	EINFO (Application)	LI or LS	Displays database information for selected objects
LOGFILEOFF	LOGFILEOFF	LOF	Closes the log file opened by LOGFILEON
LOGFILEON	LOGFILEON	LON	Writes the text window contents to a file
LTYPE	LINETYPE,	LT or LTYPE	Creates, loads, and sets linetypes
LTYPE	LOADLTYPE	LLT	Loads linetypes
'MATCHPROP	COPYPROP (Application)	CP or MA	Change object properties to that of selected entity
MEASURE	MEASURE (Application)	ME or MR	Places point objects or blocks at measured intervals on an object
MENU	MENU	MU	Loads a menu file
MENU	PULLDOWN	MU or PN	Loads a menu file
MENULOAD	PULLDOWN	MU or PN	Loads partial menu files
MIRROR	MIRROR	MI	Creates a mirror image copy of objects
MIRROR (Option Delete source object)	FLIP	FL	Mirror and delete original entity
MIRROR3D	3DMIR (Application)	M3 OR 3MR	Creates a mirror image of objects about a plane
MODEL	TLMODE	MS, PS or TM	Switches from a layout tab to the Model tab and makes it current
MOVE	MOVE	M	Displaces objects a specified distance in a specified direction
MSLIDE	MSLIDE	SLD	Save current view of drawing as a slide
MSLIDE	MSLIDE	SLD	Creates a slide file of the current viewport
MSLIDE	SLDOUT	SLD	Creates a slide file of the current viewport
MSPACE	See TLMODE	MS, PS or TM	Switches from paper space to model space
MTEDIT	MTEXTEDIT	MTE	Edit multiline text / Import text files
MTEXT	MTEXT	T or MT	Creates multi-line text
MVIEW	VIEWPORT	MV or VP	Creates floating viewports and turns on existing floating viewports
#NAME?	XLINK	XL	Controls external references to drawing files (Command line version)
NEW	NEW	N	Creates a new drawing file
NEW - Template Option	TEMPLATE	TP	Defines drawing file to be used as example in initial drawing setup
OFFSET	DOFFSET	DOF or O	Creates concentric circles, parallel lines, and parallel curves
OOPS	OOPS	OOPS	Restores erased objects
OOPS	UNDELETE	UD	Restores erased objects
OPEN	OPEN	OPN	Opens an existing drawing file
OPTIONS	SETTINGS	OP, RM, PR or ST	Customizes the Drawing settings
OPTIONS	VIEWPAR	VR	Selects view parameters
'ORTHO	'TORTHO	TORT	Constrains cursor movement
OSNAP	OSNAP	OS	Calls for manual snap setting
'OSNAP	'PRECPAR	OS or PCP	Sets running object snap modes and

'OSNAP	SETTINGS	OS or PCP	changes the target box size Sets running object snap modes and changes the target box size
PAN	DPAN	DP	Pan dynamically
'PAN	'PAN	P	Moves the drawing display in the current viewport
PASTECLIP	PASTE	PST	Inserts data from the Clipboard
PEDIT	POLYEDIT	PE	Edits polylines and three-dimensional polygon meshes
PEDIT	VTXEDIT	VTX or VX	Edits polyline vertices)
PEDIT	VTXEDIT	VTX or VX	Edits polylines and three-dimensional polygon meshes (VTXEDIT edits polyline vertices)
PFACE	PFACE	PF	Creates a three-dimensional polyface mesh vertex by vertex
PLAN	PLAN	PLN	Displays the plan view of a user coordinate system
PLAN - Option Current	PLANVIEW	PV	Returns a 3d view to being planar to current UCS
PLINE	PLINE	PL	Creates two-dimensional polylines
PLINE	POLYLINE	PL	Creates two-dimensional polylines
PLOT	PRINT	PT or PRINT	Plots a drawing to a plotter, printer, or file
PLOT – Settings	PSETUP	PSTP	Sets up print devices
POINT	POINT	PO or POI	Creates a point object
POLYGON	NGON	NG or POL	Creates an equilateral closed polyline
POLYGON	NGON	NG or POL	Creates an equilateral closed polyline
POLYGON	NGON	PG or NG	Creates an equilateral closed polyline
PREFERENCES	CONFIG	PCP	Customizes the program settings
PREFERENCES	DESKCONF	DSK	Customizes the program settings
PREFERENCES	PRECPAR	OS or PCP	Customizes the program settings
PREFERENCES	SETTINGS	OP, RM, PR or ST	Selects view parameters
PREFERENCES	'SETTINGS	OP, RM, PR or ST	Customizes the program settings
PSPACE	See TLMODE	MS, PS or TM	Switches from model space to paper space
PURGE	PURGE	PU or PRG	Removes unused named objects, such as blocks or layers, from the database
QSAVE	QSAVE	QS	Saves the current drawing
RECOVER	FLXRECOVER	REC	Repairs a damaged drawing
RECTANG	RECTANG	RT	Draws a rectangular polyline
RECTANG	RECTANGLE	RT	Draws a rectangular polyline
REDEFINE	REDEFINE	RED	Restores AutoCAD internal commands overridden by UNDEFINE
REDO	REDO	RD	Reverses the effects of the previous UNDO or U command
'REDRAW	REDRAW	R	Refreshes the display of the current viewport
REDRAWALL	REDRAWALL	RA	Refreshes the display in all viewports
REGEN	REGEN	RE or RG	Regenerates the drawing and refreshes the current viewport
REGENALL	REGENALL	REA or RGA	Regenerates the drawing and refreshes all viewports
RENAME	RENAME	REN or RNM	Changes the names of objects
RENDER	RENDER	SHA or RR	Creates a realistically shaded image of a three-dimensional wireframe or solid



REVSURF	REVSURF	RS	model Creates a rotated surface about a selected axis
ROTATE	ROTATE	RO	Moves objects about a base point
ROTATE3D	3DROT (Application)	3R or R3	Moves objects about a three-dimensional axis
RPREF	RCONFIG	RPR or RC	Sets rendering preferences
RULESURF	RULESURF	RSF	Creates a ruled surface between two curves
SAVE	SAVE	SAV	Saves the drawing under the current file name or a specified name
SAVEAS	SAVEAS	SA	Saves an unnamed drawing with a file name or renames the current drawing
SCALE	SCALE	SC	Enlarges or reduces selected objects equally in the X, Y, and Z directions
SCRIPT	RUNMACRO	RMAC	Calls and executes a macro
'SCRIPT	_SCRIPT	MAC or SCR	Executes a sequence of commands from a script
'SCRIPT	'MACRO	MAC or SCR	Executes a sequence of commands from a script
SELECT	SELECT	SL	Places selected objects in the Previous selection set
SETLINETYPE	SETLINETYPE	SLT	Sets linetypes
'SETVAR	'SETVAR	SET or SV	Lists or changes the values of system variables
SHADE	RENDER	SHA or RR	Displays a flat-shaded image of the drawing in the current viewport
'SNAP	'TSNAP	SN or TS	Restricts cursor movement to specified intervals. Toggles snap modes on/off
SOLID	2DFACE	2F or SO	Creates solid-filled polygons
SOLID	SOLID	SO	Creates solid-filled polygons
SPHERE	3DSPHER (Application)	3S or 3DS	Creates three-dimensional solid sphere
SPLINE	SPLINE (Application)	SP or SPL	Creates quadratic or cubic spline (NURBS) curve
SPLINEDIT	SPLINE;D	SPD or SPE	Edits a pline object
STRETCH	STRETCH	S or SR	Moves or stretches objects
STYLE	_STYLE	FT	Loads text styles from the specified font file and customize it to the individual requirement.
STYLE	FONT	FT	Loads text styles from the specified font file and customize it to the individual requirement.
'STYLE	QFONT	QF	Creates or modifies named styles and sets the current style for text in your drawing
TABLET	DIGICONFIG	DC	Digitizer configuration
TABLET	DIGIOFF	DGF	Switch digitizer mode off
TABLET	DIGION	DGN	Switch Digitizer mode on
TABLET	TABLET	TA	Calibrates, configures, and turns on and off an attached digitizing tablet
TABLET - CONFIG	TABCONFIG	TC	Configure tablet menu areas
TABSURF	TABSURF	TF	Creates a tabulated surface from a path curve and a direction vector
TEXT	TEXT	TX	Creates a single line of text

'TEXTSCR	'TEXTSCR	TN	Opens the text window
'TEXTSCR	'TSCREEN	TN	Opens the text window
TOOLBAR	PALETTE	PA, or PLT	Places a palette on the desktop
TOOLBAR	PALMAN	PM	Opens the palette manager
TOOLBAR	PALOPEN	PLO	Opens a new palette
TORUS	3DTORUS	3T or TOR	Creates a donut-shaped solid
TRIM	TRIM	TR	Trims objects at a cutting edge defined by other objects
U	U	U	Reverses the most recent operation
UCS	UCS	UC	Manages coordinate systems
UCSICON (Sysvar)	TICON (Sysvar)	TI	Controls the visibility and placement of the UCS icon
UCSMAN	UCS_C	UCC	Manages defined user coordinate systems
UNDEFINE	UNDEFINE (Application)	UF	Allows an application-defined command to override an internal AutoCAD command
UNDO	UNDO	U	Reverses the effect of commands
'UNITS	'UNITSETUP	UN or US	Sets coordinate and angle display formats and precision
'VIEW	VIEW	V	Creates and restores views
VIEWPORT	WOPEN	WO	Opens new viewport
VPLAYER	VIEWPORT option LAYER	VPL	Sets layer visibility within viewports
VPOINT	3DVIEW	3V	Sets the viewing direction for a three-dimensional visualization of the drawing
VPOINT	SETVIEWDIR	STV	Sets the viewing direction for a three-dimensional visualization of the drawing
VPORTS	VIEWPORT	MV or VP	Divides the graphics area into multiple tiled viewports
VSLIDE	VSLIDE	VS	Displays a raster image slide file in the current viewport
WBLOCK	PARTEXP	PX or W	Writes objects to a new drawing file
WEDGE	3DWEDGE (Application)	3W or WE	Creates 3D solid with a sloped face tapering along the X axis
WMFOUT	WMFOUT	WM	Saves objects to a Windows metafile
XATTACH	XINSERT	XI	Attaches an external reference (Xref)
XBIND	XLINK	XL	Binds dependent symbols of an xref to a drawing
XREF	XLINK	XL	Controls external references to drawing files
_XREF (Command line version)	XLINK	XL	Controls external references to drawing files
'ZOOM	'ZOOM	Z	Increase or decreases the apparent size of objects in the current viewport
ZOOM – (Option Realtime)	DZOOM	DZ	Dynamic zooming
Zoom - Extents	ZOOMPAGE (Application)	ZP	Zoom to drawing page limits
Zoom - Factor	'ZOOMIN	ZN	Zoom into the drawing
Zoom - Factor	'ZOOMOUT	ZT	Zoom out from the drawing
Zoom - Previous	'ZOOMBACK	ZX	Zoom to previous view
Zoom - Scale	'ZOOMFAC	ZF	Zoom by a factor
Zoom - Window	'ZOOMWIN	ZW	Zoom to a window
Zoom Extents	'ZOOMFIT	ZE	Zoom to extents

## Unique AllenCad5 Pro Commands

AllenCad Pro Command	Alias	Command Description
3DELLIPS	3L	Creates a 3d ellipsoid
ARENDER	AE	Animated rendering
ATTDEDIT	ATD	Modify existing attribute definition
'ATTMOVE (Application)	ATM	Move attribute text
'ATTROT (Application)	ATR	Rotate attribute text
ATTVALUE	AV	Edit attribute text value
BMPEDIT (Application)	BE	Calls external BMP into Windows Paintbrush program for editing
BATPROC (Application)	BT	Batch processing - Plots and conversion
CHAIN	CN	Draws single entity continuous lines
DINSERT	DIN	Loads but doesn't insert the part in the drawing
DELFLK	DLK	Deletes FelixCAD lock files (*.flk)
DLGEDIT	DM	Dialog and Menu editor
DIMTNEW	DN or DOV	Changes dimension text value
DIMSER	DSR	Adds multiple additional dimension labels in the same orientation
DIMTMOVE	DTM	Moves dimension text
EDITPAR	EP	Dialog to change edit parameters
FCOMPILE	FC	Compile a shape file to a binary font file
HIDE	HI or HD	Creates a second drawing file with a hidden line view
HPUPDATE	HPU	Updates hatch pattern parameters to new settings
ICONMENU	IM	Creates menu selection with graphics using .mni files
INSERTQ	IQ	Inserts an internal part into the drawing
LIBRARY	LB	Manages collections of symbols and parts
LAYCONF	LC	Save and restore layer settings
LISTFILE	LF	View contents of an external file
'PANLEFT	LFT	Pans left
PARTLIB	LIB	Calls the part library manager
LOADLISP	LL	Loads a LISP file for use
LAYREF	LRF	Change object layer properties to match reference object
MERGE	MG	Import an external .flx drawing file
OFFSET	O	Offset linear objects in parallel orientation
'PANDOWN	PD	Pans down
PLBSETUP	PLB	Sets up or modifies existing part (block) libraries
PALCLOSE	PLC	Removes a palette from the desktop
PROPLAYER (application)	PP	Moves objects to selected layer
'PANUP	PUP	Pans up
QATTDEF	QAD	Command line version of ATTDEF command
QBMPOUT	QB	Command line version of BMPOUT command
QFONT	QF	Command line version of FONT command
QHATCH	QH	Command line version of HATCH command
QHPEDIT	QHE	Command line version of HPEDIT command
QINSERT	QI	Command line version of INSERT command
QLAYER	QL	Command line version of LAYER command
QMTEXT	QM	Command line version of MTEXT

QOPEN	QO	Command line version of OPEN command
QPRINT	QP	Command line version of PRINT command
QPROPEdit	QPR	Command line version of PROPEdit command
QARRAY	QR	Command line version of ARRAY command
QSLDOUT	QSL	Command line version of SLDOUT command
QTEXT	QT	Command line version of TEXT command
QVIEW	QV	Command line version of VIEW command
QVIEWPORT	QVP	Command line version of VIEWPORT command
QWOPEN	QW	Command line version of WOPEN command
QWMFOUT	QWO	Command line version of WMFOUT command
QXLINK	QX	Command line version of XLINK command
'PANRIGHT	RGT	Pans right
REJOIN (application)	RJ	Creates single entity from co-linear objects (lines and arcs)
RTEXT (Application)	RX	Writes new text with same properties as a selected text entity
SAVEALL	SAL	Saves all open drawings
SENDFLX	SD	Sends the current drawing via Microsoft Exchange
SENDALL	SDA	Sends all drawings via Microsoft Exchange
TABSECTION	SEC	Replaces a section of the tablet menu
SETLIB (Application)	SLB	Displays current part library dialog
'SETLAYER	STL	Change current layer to selection
TABSERVICE	TB	Changes the names of named objects
TABLES (Application)	TBL	Documents and controls database information and drawing parameters
TCORRECT	TCOR	Corrects text entities and attribute definitions
'TSCREENFULL	TSC	Toggles command line window on/off (full screen)
TRAPEZOID (Application)	TZ	Creates a 3D trapezoid
WCASCADE	WD	Arrange windows in cascade format
WTILEHOR	WH	Arranges Windows in a horizontally tiled format
WICONARR	WI	Arranges all drawing windows
WTILE	WT	Arranges Windows in a tiled format
WTILEVER	WV	Arranges Windows in a vertically tiled format
EXPLODE	X	Breaks a compound object into its component objects
XPLORER	XP	Accesses and allows alteration of the current drawings properties

## Unique AutoCAD 2000 Commands

AutoCAD 2000 Command	Command Description
3DARRAY	Creates a three-dimensional array
3DCLIP	Invokes the interactive 3D view and opens the Adjust Clipping Planes window
3DCORBIT	Invokes the interactive 3D view and enables you to set the objects in the 3D view into continuous motion
3DDISTANCE	Invokes the interactive 3D view and makes objects appear closer or farther away
3DORBIT	Controls the interactive viewing of objects in 3D
3DPAN	Invokes the interactive 3D view and enables you to drag the view horizontally and vertically
3DSIN	Imports a 3D Studio File
3DSOUT	Exports to a 3D Studio file

3DSWIVEL	Invokes the interactive 3D view and simulates the effect of turning the camera
3DZOOM	Invokes the interactive 3D view so you can zoom in and out on the view
ACISIN	Imports an ACIS file
ACISOUT	Exports AutoCAD solid objects to an ACIS file
ADCCLOSE	Closes AutoCAD DesignCenter
ADCENTER	Manages content
ADCNAVIGATE	Directs the Desktop in AutoCAD DesignCenter to the file name, directory location, or network path you specify
ALIGN	Moves and rotates objects to align with other objects
AMECONVERT	Converts AME solid models to AutoCAD solid objects
ARX	Loads, unloads, and provides information about ARX applications
ASEADMIN	Performs administrative functions for external database commands
ASEEXPORT	Exports link information for selected objects
ASELINKS	Manages links between objects and an external database
ASEROWS	Displays and edits table data and creates links and selection sets
ASESELECT	Creates a selection set from rows linked to textual selection sets and graphic selection sets
ASESQLED	Executes Structured Query Language (SQL) statements
ATTREDEF	Redefines a block and updates associated attributes
AUDIT	Evaluates the integrity of a drawing
BACKGROUND	Sets up the background for your scene
BLOCKICON	Generates preview images for blocks created with Release 14 or earlier
BROWSER	Launches the default Web browser defined in your system registry
'CAL	Evaluates mathematical and geometric expressions
CAMERA	Sets a different camera and target location
CONVERT	Converts 2D polylines and associative hatches to the optimized Release 14 format
COPYBASE	Copies objects with a specified base point
COPYHIST	Copies the text in the command line history to the Clipboard
COPYLINK	Copies the current view to the Clipboard for linking to other OLE applications
DBCCLCLOSE	Closes the dbConnect Manager
DBCONNECT	Provides an AutoCAD interface to external database tables
DBLIST	Lists database information for each object in the drawing
'DDGRIPS	Turns on grips and sets their color
DDMODIFY	Controls properties of existing objects
DDUCSP	Selects a preset user coordinate system
'DELAY	Provides a timed pause within a script
DIM/DIM1	Both Access Dimensioning mode
DSVIEWER	Opens the Aerial View window
DWFOUT	Exports a drawing web format file
DWGPPOPS	Sets and displays the properties of the current drawing
DXBIN	Imports specially coded binary files
EDGESURF	Creates a three-dimensional polygon mesh
EXPRESSTOOLS	Activates the installed AutoCAD Express Tools if currently unavailable
EXTRUDE	Creates unique solid primitives by extruding existing two-dimensional objects
'FILTER	Creates lists to select objects based on properties
FIND	Finds, replaces, selects, or zooms to specified text
FOG	Provides visual cues for the apparent distance of objects
HYPERLINK	Attaches a hyperlink to a graphical object or modifies an existing hyperlink
HYPERLINKOPTIONS	Controls the visibility of the hyperlink cursor and the display of

IMAGEADJUST	hyperlink tooltips
IMAGEATTACH	Controls the brightness, contrast and fade values of the selected image
IMAGECLIP	Attaches a new image object and definition
INSERTOBJ	Creates new clipping boundaries for single image objects
INTERFERE	Inserts a linked or embedded object
	Finds the interference of two or more three-dimensional solids and creates a composite 3D solid from their common volume
'ISOPLANE	Specifies the current isometric plane
LAYOUT	Creates a new layout and renames, copies, saves, or deletes an existing layout
LAYOUTWIZARD	Starts the Layout wizard, in which you can designate page and plot settings for a new layout
LEADER	Creates a line that connects annotation to a feature
LSEDT	Lets you edit a landscape object
LSLIB	Lets you maintain libraries of landscape objects
LSNEW	Lets you add realistic landscape items, such as trees and bushes, to your drawings
LWEIGHT	Sets the current lineweight, lineweight display options, and lineweight units
MASSPROP	Calculates and displays the mass properties of regions or solids
MATLIB	Imports and exports materials to and from a library of materials
MENUUNLOAD	Unloads partial menu files
MINsert	Inserts multiple instances of a block in a rectangular array
MLEDIT	Edits multiple parallel lines
MLINE	Creates multiple parallel lines
MLSTYLE	Defines a style for multiple parallel lines
MRMRBLOCK	Creates a block definition from selected objects
MULTIPLE	Repeats the next command until canceled
MVSETUP	Sets up the specifications of a drawing
OLELINKS	Updates, changes, and cancels existing OLE links
OLESCALE	Displays the OLE Properties dialog box
PAGESETUP	Specifies the layout page, plotting device, paper size, and settings for each new layout
PARTIALLOAD	Loads additional geometry into a partially opened drawing
PARTIALOPEN	Loads geometry from a selected view or layer into a drawing
PASTEBLOCK	Pastes a copied block into a new drawing
PASTEORIG	Pastes a copied object in a new drawing using the coordinates from the original drawing
PASTESPEC	Inserts data from the Clipboard and controls the format of the data
PCINWIZARD	Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout
PLOTSTYLE	Sets the current plot style for new objects, or the assigned plot style for selected objects
PLOTTERMANAGER	Displays the Plotter Manager, where you can launch the Add-a-Plotter wizard and the Plotter Configuration Editor
PREVIEW	Shows how the drawing will look when it is printed or plotted
PROPERTIES	Controls properties of existing objects
PROPERTIESCLOSE	Closes the Properties window
PSDRAG	Controls the appearance of a PostScript image as it is dragged into position with PSIN
PSETUPIN	Imports a user-defined page setup into a new drawing layout
PSFILL	Fills a two-dimensional polyline outline with a PostScript pattern
PSIN	Imports a PostScript file
PSOUT	Creates an Encapsulated PostScript file
QDIM	Quickly creates a dimension
QLEADER	Quickly creates a leader and leader annotation

QSELECT	Quickly creates selection sets based on filtering criteria
RAY	Creates a semi-infinite line
REFCLOSE	Saves back or discards changes made during in-place editing of a reference (an xref or a block)
REFEDIT	Selects a reference for editing
REFSET	Adds or removes objects from a working set during in-place editing of a reference (an xref or a block)
REGION	Creates a region object from a selection set of existing objects
REINIT	Reinitializes the digitizer, digitizer input/output port, and program parameters file
RENDSCR	Redisplays the last rendering created with the RENDER command
REPLAY	Displays a BMP, TGA, or TIFF image
'RESUME	Continues an interrupted script
REVOLVE	Creates a solid by revolving a two-dimensional object about an axis
RMAT	Manages rendering materials
RSCRIPT	Creates a script that repeats continuously
SAVEIMG	Saves a rendered image to a file
SCENE	Manages scenes in model space
SECTION	Uses the intersection of a plane and solids to create a region
SETUV	Lets you map materials onto geometry
SHADEMODE	Shades the objects in the current viewport
SHAPE	Inserts a shape
SHELL	Accesses operating system commands
SHOWMAT	Lists the material type and attachment method for a selected object
SKETCH	Creates a series of freehand line segments
SLICE	Slices a set of solids with a plane
SOLDRAW	Generate profiles and sections in viewports created with SOLVIEW
SOLIDEDIT	Edits faces and edges of 3D solid objects
SOLPROF	Creates profile images of three-dimensional solids
SOLVIEW	Creates floating viewports using orthographic projection to lay out multi- and sectional view drawings of 3D solid and body objects
'SPELL	Checks spelling in a drawing
STATS	Displays rendering statistics
'STATUS	Displays drawing statistics, modes, and extents
STLOUT	Stores a solid in an ASCII or binary file
STYLESMANAGER	Displays the Plot Style Manager
SUBTRACT	Creates a composite region or solid by subtraction
SYSWINDOWS	Arranges windows
'TIME	Displays the date and time statistics of a drawing
TOLERANCE	Creates isometric tolerances
TRACE	Creates solid lines
TRANSPARENCY	Controls whether background pixels in an image are transparent or opaque
'TREESTAT	Displays information about the drawing's current spatial index
UNION	Creates a composite region or solid
VBAIDE	Displays the Visual Basic Editor
VBALOAD	Loads a global VBA project into the current AutoCAD session
VBAMAN	Loads, unloads, saves, creates, embeds, and extracts VBA projects
VBARUN	Runs a VBA macro
VBASTMT	Executes a VBA statement on the AutoCAD command line
VBAUNLOAD	Unloads a global VBA project
VLISP	Displays the Visual LISP interactive development environment (IDE)
VPCLIP	Clips viewport objects
WHOHAS	Displays ownership information for opened drawing files
WMFIN	Imports a Windows metafile
WMFOPTS	Sets options for WMFIN

XCLIP	Defines an xref clipping boundary and sets the front or back clipping planes
XLIN	Creates an infinite line

## Equivalent AutoCAD 2000 / AllenCad Pro Systems Variables

<b>AutoCAD 2000 Sysvars</b>	<b>AllenCad Pro Sysvars</b>	<b>Local / Global</b>	<b>Function</b>
ACADVER	FCVERSION		Stores the PROGRAM version number
AFLAGS	AFLAGS	(L)	Sets attribute flags for ATTDEF bit-code
ANGBASE	ANGBASE	(L)	Sets the base angle 0 with respect to the current UCS
ANGDIR	ANGDIR	(L)	Sets the positive angle direction from angle 0 with respect to the current UCS
APERTURE	SNAPBOX		Sets object snap target height, in pixels
AREA	AREA	(G)	Stores the last area computed by AREA, LIST, or DBLIST
ATTDIA	ATTDIA	(L)	Controls whether -INSERT uses a dialog box for attribute value entry
ATTMODE	ATTMODE	(L)	Controls display of attributes
ATTREQ	ATTREQ	(L)	Determines whether INSERT uses default attribute settings during insertion of blocks
AUNITS	AUNITS	(L)	Sets units for angles
AUPREC	AUPREC	(L)	Sets the number of decimal places for angular units
AUTOSNAP	DYNASNAP		Controls Snap marker, tooltip, and magnet
AUTOSNAP	DYNASNAP	(G)	Controls dynasnap symbol and tooltip display
BACKZ	BACKZ	(L)	Stores the back clipping plane offset from the target plane for the current viewport
BLIPMODE	BLIPMODE	(L)	Controls whether marker blips are visible
CDATE	CDATE	(G)	Sets calendar date and time
CECOLOR	CECOLOR	(L)	Sets the color of new objects
CELTSCALE	CELTSCALE	(L)	Sets the current object linetype scaling factor
CELTYPE	CELTYPE	(L)	Sets the linetype of new objects
CHAMFERA	CHAMFERA	(L)	Sets the first chamfer distance
CHAMFERB	CHAMFERB	(L)	Sets the second chamfer distance
CHAMFERC	CHAMFERC	(L)	Sets the chamfer length
CHAMFERD	CHAMFERD	(L)	Sets the chamfer angle
CHAMMODE	CHAMMODE	(L)	Sets the input method by which The program creates chamfers
CIRCLERAD	CIRCLERAD	(G)	Sets the default circle radius
CLAYER	CLAYER	(L)	Sets the current layer
CMDACTIVE	CMDACTIVE	(G)	Stores the bit-code that indicates whether an ordinary command, transparent command, script, or dialog box is active
CMDECHO	CMDECHO	(G)	Controls whether The program echoes prompts and input during the LISP (command) function
CMDNAMES	CMDNAMES	(L)	Displays the names of the active and transparent commands
COORDS	COORDS		Controls when coordinates are updated on the status line
CURSORSIZE	CURSORSIZE	(G)	Determines the size of the crosshairs as a percentage of the screen size



CVPORT	CVPORT	(L)	Sets the identification number of the current viewport
DATE	DATE	(G)	Stores the current date and time
DBMOD	DBMOD	(L)	Indicates the drawing modification status using bit-code
DIASAT	DIASAT	(G)	Stores the exit method of the most recently used dialog box
DIMALT	DIMALT	(L)	Controls the display of alternate units in dimensions
DIMALTD	DIMALTD	(L)	Controls the number of decimal places in alternate units
DIMALTF	DIMALTF	(L)	Controls the multiplier for alternate units
DIMAPOST	DIMAPOST	(L)	Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular
DIMASO	DIMASO	(L)	Controls the associativity of dimension objects
DIMASZ	DIMASZ	(L)	Controls the size of dimension line and leader line arrowheads
DIMBLK	DIMBLK	(L)	Sets the arrowhead block displayed at the ends of dimension lines or leader lines
DIMBLK1	DIMBLK1	(L)	Sets the arrowhead for the first end of the dimension line when DIMSAH is on
DIMBLK2	DIMBLK2	(L)	Sets the arrowhead for the second end of the dimension line when DIMSAH is on
DIMCEN	DIMCEN	(L)	Controls drawing of circle or arc center marks and centerlines by DIMCENTER, DIMDIAMETER, and DIMRADIUS
DIMCLRD	DIMCLRD	(L)	Assigns colors to dimension lines, arrowheads, and dimension leader lines
DIMCLRE	DIMCLRE	(L)	Assigns colors to dimension extension lines
DIMCLRT	DIMCLRT	(L)	Assigns colors to dimension text
DIMDLE	DIMDLE	(L)	Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn instead of arrowheads
DIMDLI	DIMDLI	(L)	Controls the spacing of dimension lines in baseline dimensions
DIMEXE	DIMEXE	(L)	Specifies how far to extend the extension line beyond the dimension line
DIMEXO	DIMEXO	(L)	Specifies how far extension lines are offset from origin points
DIMGAP	DIMGAP	(L)	Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text
DIMLFAC	DIMLFAC	(L)	Sets a scale factor for linear dimension measurements
DIMLIM	DIMLIM	(L)	Generates dimension limits as the default text
DIMPOST	DIMPOST	(L)	Specifies a text prefix or suffix (or both) to the dimension measurement
DIMRND	DIMRND	(L)	Rounds all dimensioning distances to the specified value
DIMSAH	DIMSAH	(L)	Controls the display of dimension line arrowhead blocks
DIMSCALE	DIMSCALE	(L)	Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets
DIMSE1	DIMSE1	(L)	Suppresses display of the first extension line
DIMSE2	DIMSE2	(L)	Suppresses display of the second extension line
DIMSHO	DIMSHO	(L)	Controls redefinition of dimension objects while

DIMSOXD	DIMSOXD	(L)	dragging Suppresses drawing of dimension lines outside the extension lines
DIMSTYLE	DIMSTYLE	(L)	Shows the current dimension style
DIMTAD	DIMTAD	(L)	Controls the vertical position of text in relation to the dimension line
DIMTFAC	DIMTFAC	(L)	Sets a scale factor used to calculate the height of text for dimension fractions and tolerances
DIMTIH	DIMTIH	(L)	Controls the position of dimension text inside the extension lines for all dimension types except ordinate
DIMTIX	DIMTIX	(L)	Draws text between extension lines
DIMTM	DIMTM	(L)	When DIMTOL or DIMLIM is on, sets the minimum (or lower) tolerance limit for dimension text
DIMTOFL	DIMTOFL	(L)	Controls whether a dimension line is drawn between the extension lines even when the text is placed outside
DIMTOH	DIMTOH	(L)	Controls the position of dimension text outside the extension lines
DIMTOL	DIMTOL	(L)	Appends tolerances to dimension text
DIMTP	DIMTP	(L)	When DIMTOL or DIMLIM is on, sets the maximum (or upper) tolerance limit for dimension text
DIMTSZ	DIMTSZ	(L)	Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning
DIMTVP	DIMTVP	(L)	Controls the vertical position of dimension text above or below the dimension line
DIMTXT	DIMTXT	(L)	Specifies the height of dimension text, unless the current text style has a fixed height
DIMZIN	DIMZIN	(L)	Controls the suppression of zeroes in the primary unit value
DISTANCE	DISTANCE	(G)	Stores the distance computed by DIST
DONUTID	RINGDIA1		Sets the default for the inside diameter of a donut
DONUTOD	RINGDIA2		Sets the default for the outside diameter of a donut
DRAGMODE	DRAGMODE	(L)	Controls the display of objects being dragged
DWGNAME	DWGNAME	(L)	Stores the drawing name as entered by the user
DWGPREFIX	DWGPREFIX	(L)	Stores the drive/directory prefix for the drawing
DWGTITLED	DWGTITLED		Indicates whether the current drawing has been named
ELEVATION	ELEVATION	(L)	Stores the current elevation relative to the current UCS for the current viewport in the current space
EXPERT	EXPERT	(G)	Controls whether certain prompts are issued
EXTMAX	EXTMAX		Stores the upper-right point of the drawing extents
EXTMIN	EXTMIN		Stores the lower-left point of the drawing extents
FILEDIA	FILEDIA	(G)	Suppresses display of the file dialog boxes
FILLETRAD	FILLETRAD	(L)	Stores the current fillet radius
FILLMODE	FILLMODE	(L)	Specifies whether multilines, traces, solids, all hatches (including solid-fill), and wide polylines are filled in
FONTALT	FONTALT	(G)	Specifies the alternate font to be used when the specified font file cannot be located
FRONTZ	FRONTZ	(L)	Stores the front clipping plane offset from the target plane for the current viewport
GRIDMODE	GRIDMODE	(L)	Specifies whether the grid is turned on or off

GRIDUNIT	GRIDUNIT	(L)	Specifies the grid spacing (X and Y) for the current viewport
HIGHLIGHT	HIGHLIGHT	(G)	Controls object highlighting; does not affect objects selected with grips
HPANG	HPANG	(L)	Specifies the hatch pattern angle
HPDOUBLE	HPDOUBLE	(L)	Specifies hatch pattern doubling for user-defined patterns
HPSCALE	HPSCALE	(L)	Specifies the hatch pattern scale factor
HPSPACE	HPSPACE	(L)	Specifies the hatch pattern line spacing for user-defined simple patterns
INSBASE	INSBASE	(L)	Stores insertion base point set by BASE
INSNAME	INSNAME	(L)	Sets a default block name for INSERT
LASTANGLE	LASTANGLE		Stores the end angle of the last arc entered
LASTPOINT	LASTPOINT	(L)	Stores the last point entered
LENSLENGTH	LENSLENGTH (for future use)	(L)	Stores the length of the lens (in millimeters) used in perspective viewing for the current viewport
LIMCHECK	LIMCHECK	(L)	Controls the creation of objects outside the drawing limits
LIMMAX	LIMMAX	(L)	Stores the upper-right drawing limits for the current space
LIMMIN	LIMMIN	(L)	Stores the lower-left drawing limits for the current space
LOGFILEMODE	LOGFILEMODE	(G)	Specifies whether the contents of the text window are written to a log file
LOGFILENAME	LOGFILENAME	(G)	Specifies the path and name of the log file
LTSCALE	LTSCALE	(L)	Sets the global linetype scale factor
LUNITS	LUNITS	(L)	Sets linear units
LUPREC	LUPREC	(L)	Sets the number of decimal places displayed for linear units
MIRRTEXT	MIRRTEXT	(L)	Controls how MIRROR reflects text
OFFSETDIST	OFFSETDIST	(G)	Sets the default offset distance
ORTHOMODE	ORTHOMODE	(L)	Constrains cursor movement to the perpendicular
OSMODE	OSMODE	(L)	Sets running object snap modes using bitcodes
PDMODE	PDMODE	(L)	Controls how point objects are displayed
PDSIZE	PDSIZE	(L)	Sets the display size for point objects
PERIMETER	PERIMETER	(G)	Stores the last perimeter value computed by AREA, LIST, or DBLIST
PICKBOX	SELECTBOX		Sets object selection target height
PICKBOX	SELECTBOX	(G)	Permits sizing the of the cursor select box in pixels
PICKFIRST	PICKFIRST	(G)	Controls whether you select objects before or after you issue a command
PICKSTYLE	GROUPSEL	(L)	Controls group selectability
PICKSTYLE	PICKSTYLE	(L)	Controls the use of group selection and associative hatch selection
PLATFORM	PLATFORM	(G)	Indicates which platform of the program is in use
PLINEGEN	PLINEGEN (for future use)	(L)	Sets how linetype patterns generate around the vertices of a 2D polyline
PLINETYPE	PLINETYPE	(G)	Specifies whether the program uses optimized 2D polylines
PLINEWID	PLINEWID	(L)	Stores the default polyline width
POLYSIDES	POLYSIDES	(G)	Sets the default number of sides for POLYGON
PSLTSCALE	PSLTSCALE	(L)	Controls paper space linetype scaling
QTEXTMODE	QTEXTMODE	(L)	Controls how text is displayed
REGENMODE	REGENMODE	(L)	Controls automatic regeneration of the drawing
SAVETIME	SAVETIME	(G)	Sets the automatic save interval, in minutes
SCREENMODE	SCREENMODE	(G)	Stores a bitcode indicating the graphics/text state of

SCREENSIZE	SCREENSIZE	(G)	the The program display
See VIEWRES	CIRCLERES	(G)	Stores current viewport size in pixels (X and Y)
SNAPANG	SNAPANG	(L)	Circle resolution - determines the number of line segments used to display a circle
SNAPBASE	SNAPBASE	(L)	Sets snap and grid rotation angle for the current viewport
SNAPISOPAIR	SNAPISOPAIR	(L)	Sets the snap and grid origin point for the current viewport relative to the current UCS
SNAPMODE	SNAPMODE	(L)	Controls the isometric plane for the current viewport
SNAPSTYL	SNAPSTYL	(L)	Turns Snap mode on and off
SNAPUNIT	SNAPUNIT	(L)	Sets snap style for the current viewport
SPLFRAME	SPLFRAME	(L)	Sets the snap spacing for the current viewport
SURFTAB1	SURFTAB1	(L)	Controls the display of splines and spline-fit polylines
SURFTAB2	SURFTAB2	(L)	Sets the number of tabulations to be generated for RULESURF and TABSURF
SURFU	SURFU	(L)	Sets the mesh density in the N direction for REVSURF and EDGESURF
SURFV	SURFV	(L)	Sets the surface density for PEDIT Smooth in the M direction
TEXTSIZE	TEXTSIZE	(L)	Sets the surface density for PEDIT Smooth in the N direction
TEXTSTYLE	TEXTSTYLE	(L)	Sets the default height for new text objects drawn with the current text style
THICKNESS	THICKNESS	(L)	Sets the name of the current text style
TILEMODE	TILEMODE	(L)	Sets the current 3D solid thickness
TRACEWID	TRACEWID	(L)	Makes the Model tab or the last layout tab current
TRIMMODE	TRIMMODE	(L)	Sets the default trace width
UCSFOLLOW	UCSFOLLOW	(L)	Controls whether The program trims selected edges for chamfers and fillets
UCSICON	UCSICON	(L)	Generates a plan view whenever you change from one UCS to another
UCSNAME	UCSNAME	(L)	Displays the UCS icon for the current viewport
UCSORG	UCSORG	(L)	Stores the name of the current coordinate system for the current viewport in the current space
UCSXDIR	UCSXDIR	(L)	Stores the origin point of the current coordinate system for the current viewport in the current space
UCSYDIR	UCSYDIR	(L)	Stores the X direction of the current UCS for the current viewport in the current space
UNDOCTL	UNDOCTL	(G)	Stores the Y direction of the current UCS for the current viewport in the current space
UNITMODE	UNITMODE	(L)	Stores a bitcode indicating the state of the Auto and Control options of the UNDO command
USERI1-5	USERI1-5	(L)	Controls the display format for units
USERR1-5	USERR1-5	(L)	Stores and retrieves integer values
USERS1-5	USERS1-5	(L)	Stores and retrieves real numbers
VIEWCTR	VIEWCTR	(L)	Stores and retrieves text string data
VIEWDIR	VIEWDIR	(L)	Stores the center of view in the current viewport
VIEWSIZE	VIEWSIZE	(L)	Stores the viewing direction in the current viewport
VIEWTWIST	VIEWTWIST	(L)	Stores the height of the view in the current viewport
VISRETAIN	VISRETAIN	(L)	Stores the view twist angle for the current viewport
		(L)	Controls the visibility, color, linetype, lineweight, and plot styles (if PSTYLEPOLICY is set to 0) of xref-dependent layers and specifies whether nested xref path changes are saved

WORLDUCS      WORLDUCS      (L)      Indicates whether the UCS is the same as the WCS

## Unique AllenCad Pro Systems Variables

AllenCad Pro Sysvars	Local / Global	Function
'ATTROT (Application)	ATR	Rotate attribute text
3DELLIPS	3L	Creates a 3d ellipsoid
ARENDER	AE	Animated rendering
ATTDEDIT	ATD	Modify existing attribute definition
'ATTMOVE	ATM	Move attribute text
(Application)		
ATTVALUE	AV	Edit attribute text value
BATPROC	BT	Batch processing - Plots and conversion
(Application)		
BMPEDIT	BE	Calls external BMP into Windows Paintbrush program for editing
(Application)		
CHAIN	CN	Draws single entity continuous lines
DELFLK	DLK	Deletes FelixCAD lock files (*.flk)
DIMSER	DSR	Adds multiple additional dimension labels in the same orientation
DIMTMOVE	DTM	Moves dimension text
DIMTNEW	DN or DOV	Changes dimension text value
DINSERT	DIN	Loads but doesn't insert the part in the drawing
DLGEDIT	DM	Dialog and Menu editor
EDITPAR	EP	Dialog to change edit parameters
EXPLODE	X	Breaks a compound object into its component objects
FCOMPILE	FC	Compile a shape file to a binary font file
HIDE	HI or HD	Creates a second drawing file with a hidden line view
HPUPDATE	HPU	Updates hatch pattern parameters to new settings
ICONMENU	IM	Creates menu selection with graphics using .mni files
INSERTQ	IQ	Inserts an internal part into the drawing
LAYCONF	LC	Save and restore layer settings
LAYREF	LRF	Change object layer properties to match reference object
LIBRARY	LB	Manages collections of symbols and parts
LISTFILE	LF	View contents of an external file
LOADLISP	LL	Loads a LISP file for use
MERGE	MG	Import an external .flx drawing file
OFFSET	O	Offset linear objects in parallel orientation
PALCLOSE	PLC	Removes a palette from the desktop
'PANDOWN	PD	Pans down
'PANLEFT	LFT	Pans left
'PANRIGHT	RGT	Pans right
'PANUP	PUP	Pans up
PARTLIB	LIB	Calls the part library manager
PLBSETUP	PLB	Sets up or modifies existing part (block) libraries
PROPLAYER	PP	Moves objects to selected layer
(application)		
QARRAY	QR	Command line version of ARRAY command
QATTDEF	QAD	Command line version of ATTDEF command
QBMPOUT	QB	Command line version of BMPOUT command
QFONT	QF	Command line version of FONT command
QHATCH	QH	Command line version of HATCH command
QHPEDIT	QHE	Command line version of HPEDIT command

QINSERT	QI	Command line version of INSERT command
QLAYER	QL	Command line version of LAYER command
QMTEXT	QM	Command line version of MTEXT
QOPEN	QO	Command line version of OPEN command
QPRINT	QP	Command line version of PRINT command
QPROPEdit	QPR	Command line version of PROPEdit command
QSLDOUT	QSL	Command line version of SLDOUT command
QTEXT	QT	Command line version of TEXT command
QVIEW	QV	Command line version of VIEW command
QVIEWPORT	QVP	Command line version of VIEWPORT command
QWMFOUT	QWO	Command line version of WMFOUT command
QWOPEN	QW	Command line version of WOPEN command
QXLINK	QX	Command line version of XLINK command
REJOIN (application)	RJ	Creates single entity from co-linear objects (lines and arcs)
RTEXT (Application)	RX	Writes new text with same properties as a selected text entity
SAVEALL	SAL	Saves all open drawings
SENDALL	SDA	Sends all drawings via Microsoft Exchange
SENDFLX	SD	Sends the current drawing via Microsoft Exchange
'SETLAYER	STL	Change current layer to selection
SETLIB (Application)	SLB	Displays current part library dialog
TABLES (Application)	TBL	Documents and controls database information and drawing parameters
TABSECTION	SEC	Replaces a section of the tablet menu
TABSERVICE	TB	Changes the names of named objects
TCORRECT	TCOR	Corrects text entities and attribute definitions
TRAPEZOID (Application)	TZ	Creates a 3D trapezoid
'TSCREENFULL	TSC	Toggles command line window on/off (full screen)
WCASCADE	WD	Arrange windows in cascade format
WICONARR	WI	Arranges all drawing windows
WTILE	WT	Arranges Windows in a tiled format
WTILEHOR	WH	Arranges Windows in a horizontally tiled format
WTILEVER	WV	Arranges Windows in a vertically tiled format
XPLORER	XP	Accesses and allows alteration of the current drawings properties

## Unique AutoCAD 2000 Systems Variables

<b>AutoCAD 2000</b>	<b>Function</b>
<b>Sysvars</b>	
3DARRAY	Creates a three-dimensional array
3DCLIP	Invokes the interactive 3D view and opens the Adjust Clipping Planes window
3DCORBIT	Invokes the interactive 3D view and enables you to set the objects in the 3D view into continuous motion
3DDISTANCE	Invokes the interactive 3D view and makes objects appear closer or farther away
3DORBIT	Controls the interactive viewing of objects in 3D
3DPAN	Invokes the interactive 3D view and enables you to drag the view horizontally and vertically
3DSIN	Imports a 3D Studio File
3DSOUT	Exports to a 3D Studio file

3DSWIVEL	Invokes the interactive 3D view and simulates the effect of turning the camera
3DZOOM	Invokes the interactive 3D view so you can zoom in and out on the view
ACISIN	Imports an ACIS file
ACISOUT	Exports AutoCAD solid objects to an ACIS file
ADCCLOSE	Closes AutoCAD DesignCenter
ADCENTER	Manages content
ADCNAVIGATE	Directs the Desktop in AutoCAD DesignCenter to the file name, directory location, or network path you specify
ALIGN	Moves and rotates objects to align with other objects
AMECONVERT	Converts AME solid models to AutoCAD solid objects
ARX	Loads, unloads, and provides information about ARX applications
ASEADMIN	Performs administrative functions for external database commands
ASEEXPORT	Exports link information for selected objects
ASELINKS	Manages links between objects and an external database
ASEROWS	Displays and edits table data and creates links and selection sets
ASESELECT	Creates a selection set from rows linked to textual selection sets and graphic selection sets
ASESQLED	Executes Structured Query Language (SQL) statements
ATTREDEF	Redefines a block and updates associated attributes
AUDIT	Evaluates the integrity of a drawing
BACKGROUND	Sets up the background for your scene
BLOCKICON	Generates preview images for blocks created with Release 14 or earlier
BROWSER	Launches the default Web browser defined in your system registry
'CAL	Evaluates mathematical and geometric expressions
CAMERA	Sets a different camera and target location
CONVERT	Converts 2D polylines and associative hatches to the optimized Release 14 format
COPYBASE	Copies objects with a specified base point
COPYHIST	Copies the text in the command line history to the Clipboard
COPYLINK	Copies the current view to the Clipboard for linking to other OLE applications
DBCCLCLOSE	Closes the dbConnect Manager
DBCONNECT	Provides an AutoCAD interface to external database tables
DBLIST	Lists database information for each object in the drawing
'DDGRIPS	Turns on grips and sets their color
DDMODIFY	Controls properties of existing objects
DDUCSP	Selects a preset user coordinate system
'DELAY	Provides a timed pause within a script
DIM/DIM1	Both Access Dimensioning mode
DSVIEWER	Opens the Aerial View window
DWFOUT	Exports a drawing web format file
DWGPROPS	Sets and displays the properties of the current drawing
DXBIN	Imports specially coded binary files
EDGESURF	Creates a three-dimensional polygon mesh
EXPRESSTOOLS	Activates the installed AutoCAD Express Tools if currently unavailable
EXTRUDE	Creates unique solid primitives by extruding existing two-dimensional objects
'FILTER	Creates lists to select objects based on properties
FIND	Finds, replaces, selects, or zooms to specified text
FOG	Provides visual cues for the apparent distance of objects
HYPERLINK	Attaches a hyperlink to a graphical object or modifies an existing hyperlink
HYPERLINKOPTIO NS	Controls the visibility of the hyperlink cursor and the display of hyperlink tooltips
IMAGEADJUST	Controls the brightness, contrast and fade values of the selected image
IMAGEATTACH	Attaches a new image object and definition
IMAGECLIP	Creates new clipping boundaries for single image objects
INSERTOBJ	Inserts a linked or embedded object
INTERFERE	Finds the interference of two or more three-dimensional solids and creates a

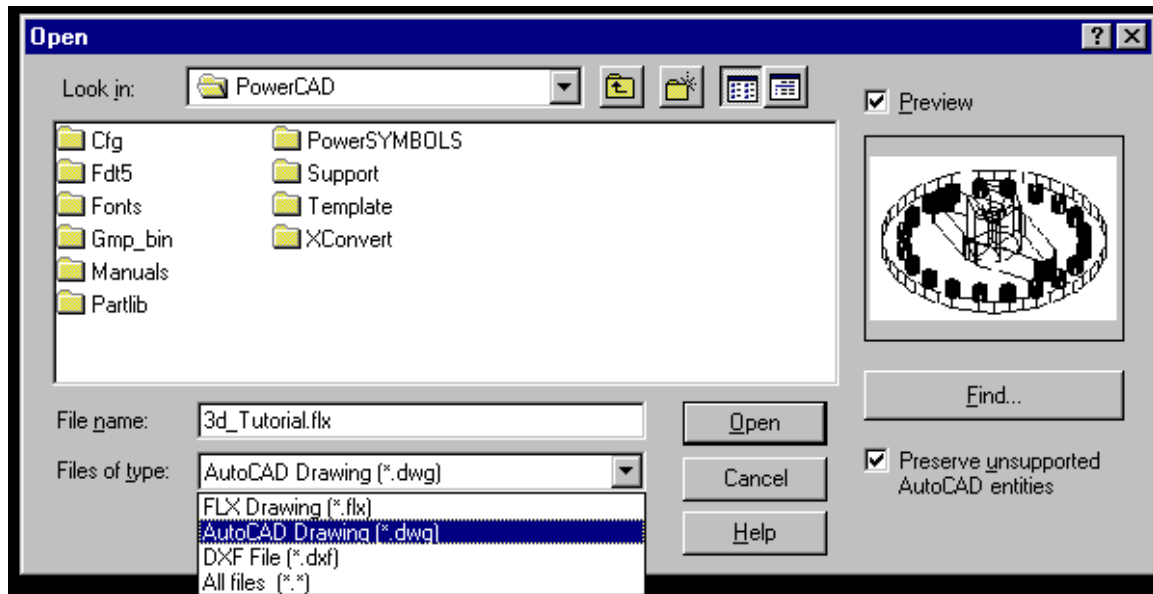
	composite 3D solid from their common volume
ISOPLANE	Specifies the current isometric plane
LAYOUT	Creates a new layout and renames, copies, saves, or deletes an existing layout
LAYOUTWIZARD	Starts the Layout wizard, in which you can designate page and plot settings for a new layout
LEADER	Creates a line that connects annotation to a feature
LSEEDIT	Lets you edit a landscape object
LSLIB	Lets you maintain libraries of landscape objects
LSNEW	Lets you add realistic landscape items, such as trees and bushes, to your drawings
LWEIGHT	Sets the current lineweight, lineweight display options, and lineweight units
MASSPROP	Calculates and displays the mass properties of regions or solids
MATLIB	Imports and exports materials to and from a library of materials
MENUUNLOAD	Unloads partial menu files
MININSERT	Inserts multiple instances of a block in a rectangular array
MLEDIT	Edits multiple parallel lines
MLINE	Creates multiple parallel lines
MLSTYLE	Defines a style for multiple parallel lines
MRMRBLOCK	Creates a block definition from selected objects
MULTIPLE	Repeats the next command until canceled
MVSETUP	Sets up the specifications of a drawing
OLELINKS	Updates, changes, and cancels existing OLE links
OLESCALE	Displays the OLE Properties dialog box
PAGESETUP	Specifies the layout page, plotting device, paper size, and settings for each new layout
PARTIALLOAD	Loads additional geometry into a partially opened drawing
PARTIALOPEN	Loads geometry from a selected view or layer into a drawing
PASTEBLOCK	Pastes a copied block into a new drawing
PASTEORIG	Pastes a copied object in a new drawing using the coordinates from the original drawing
PASTESPEC	Inserts data from the Clipboard and controls the format of the data
PCINWIZARD	Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout
PLOTSTYLE	Sets the current plot style for new objects, or the assigned plot style for selected objects
PLOTTERMANAGER	Displays the Plotter Manager, where you can launch the Add-a-Plotter wizard and the Plotter Configuration Editor
PREVIEW	Shows how the drawing will look when it is printed or plotted
PROPERTIES	Controls properties of existing objects
PROPERTIESCLOSE	Closes the Properties window
PSDRAG	Controls the appearance of a PostScript image as it is dragged into position with PSIN
PSETUPIN	Imports a user-defined page setup into a new drawing layout
PSFILL	Fills a two-dimensional polyline outline with a PostScript pattern
PSIN	Imports a PostScript file
PSOUT	Creates an Encapsulated PostScript file
QDIM	Quickly creates a dimension
QLEADER	Quickly creates a leader and leader annotation
QSELECT	Quickly creates selection sets based on filtering criteria
RAY	Creates a semi-infinite line
REFCLOSE	Saves back or discards changes made during in-place editing of a reference (an xref or a block)
REFEDIT	Selects a reference for editing
REFSET	Adds or removes objects from a working set during in-place editing of a reference (an xref or a block)
REGION	Creates a region object from a selection set of existing objects



REINIT	Reinitializes the digitizer, digitizer input/output port, and program parameters file
RENDSCR	Redisplays the last rendering created with the RENDER command
REPLAY	Displays a BMP, TGA, or TIFF image
'RESUME	Continues an interrupted script
REVOLVE	Creates a solid by revolving a two-dimensional object about an axis
RMAT	Manages rendering materials
RSCRIPT	Creates a script that repeats continuously
SAVEIMG	Saves a rendered image to a file
SCENE	Manages scenes in model space
SECTION	Uses the intersection of a plane and solids to create a region
SETUV	Lets you map materials onto geometry
SHADEMODE	Shades the objects in the current viewport
SHAPE	Inserts a shape
SHELL	Accesses operating system commands
SHOWMAT	Lists the material type and attachment method for a selected object
SKETCH	Creates a series of freehand line segments
SLICE	Slices a set of solids with a plane
SOLDRAW	Generate profiles and sections in viewports created with SOLVIEW
SOLIDEDIT	Edits faces and edges of 3D solid objects
SOLPROF	Creates profile images of three-dimensional solids
SOLVIEW	Creates floating viewports using orthographic projection to lay out multi- and sectional view drawings of 3D solid and body objects
'SPELL	Checks spelling in a drawing
STATS	Displays rendering statistics
'STATUS	Displays drawing statistics, modes, and extents
STLOUT	Stores a solid in an ASCII or binary file
STYLESMANAGER	Displays the Plot Style Manager
SUBTRACT	Creates a composite region or solid by subtraction
SYSWINDOWS	Arranges windows
'TIME	Displays the date and time statistics of a drawing
TOLERANCE	Creates isometric tolerances
TRACE	Creates solid lines
TRANSPARENCY	Controls whether background pixels in an image are transparent or opaque
'TREESTAT	Displays information about the drawing's current spatial index
UNION	Creates a composite region or solid
VBAIDE	Displays the Visual Basic Editor
VBALOAD	Loads a global VBA project into the current AutoCAD session
VBAMAN	Loads, unloads, saves, creates, embeds, and extracts VBA projects
VBARUN	Runs a VBA macro
VBASTMT	Executes a VBA statement on the AutoCAD command line
VBAUNLOAD	Unloads a global VBA project
VLISP	Displays the Visual LISP interactive development environment (IDE)
VPCLIP	Clips viewport objects
WHOHAS	Displays ownership information for opened drawing files
WMFIN	Imports a Windows metafile
WMFOPTS	Sets options for WMFIN
XCLIP	Defines an xref clipping boundary and sets the front or back clipping planes
XLINE	Creates an infinite line

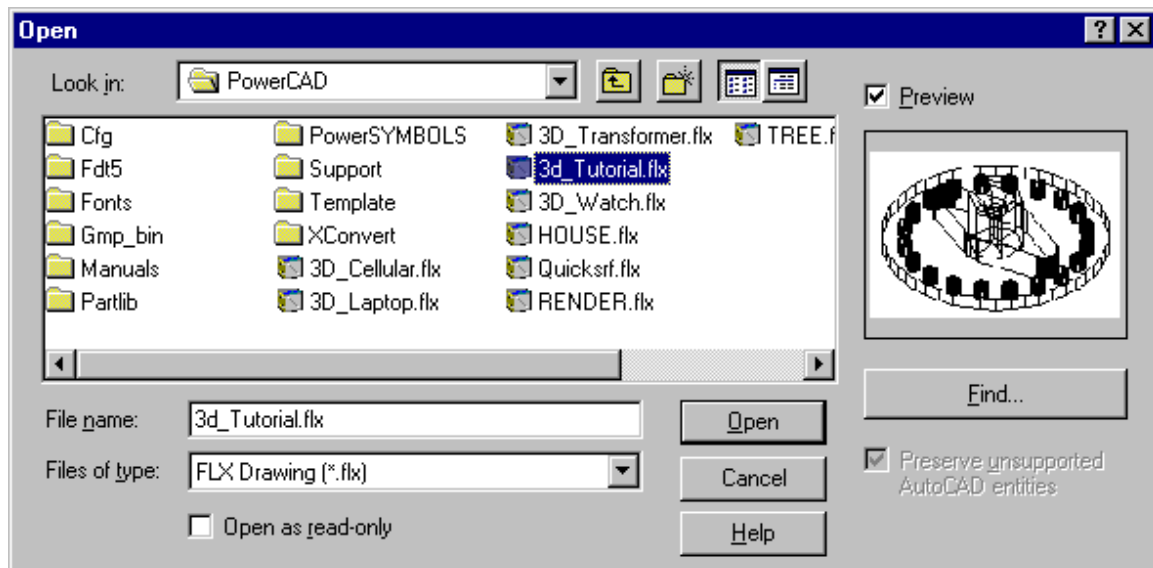
## AutoCAD File Exchange

## AutoCAD File Exchange



By default, AllenCad uses the open and published FCAD Flexible file format (extension *.FLX*). To allow for a shared AllenCad/AutoCAD experience, AllenCad R5 adds direct support for AutoCAD drawing (*.dwg* and *.dxf*) files up to R2000 format. Users can directly create or open files in AutoCAD DWG® and DXF® format from Release 2.5 through Release 2000, with "round-trip" retention of crucial R2000 entities. Files can be saved directly to R12 DWG/DXF through R2000 DWG format.

AllenCad conforms completely to AutoCAD standards for fonts (including SHX with bigfont support), layers, object properties, linetypes (supports direct loading of *.LIN* files), and hatch patterns (supports direct loading of *.PAT* files). Drawing support extends to externally referenced files, blocks (called Parts in AllenCad) management, and insertion.



## "Roundtripped" AutoCAD Entities and Table Information

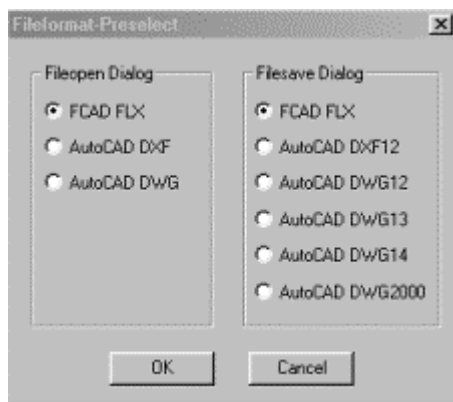
The following objects and database information, although they presently have no AllenCad equivalents and are invisible in the AllenCad drawing Editor, are nonetheless retained in the drawing database for later use.

ARCALIGNEDTEXT  
BODY  
IMAGE  
LEADER  
MLINE  
OLE2FRAME  
PROXY  
RAY  
REGION  
RTEXT  
SOLID3D  
TOLERANCE  
WIPEOUT  
XLINE  
XRECORD-OBJECTS

## "Roundtripped" AutoCAD Table Information:

DICTIONARYWDFLTOBJTYPE  
DICTVAR  
IMAGEDEF  
MLINESTYLE  
OBJECTPOINTER  
PLACEHOLDEROBJECT  
VBAPROJECTOBJECT  
WIPEOUTVAROBJECT

## Changing the Default File Format



The file format for opening and saving files can be easily changed from the default FLX format to DWG

or DXF format using the **config** command, **common** tab, then selecting the **Fileformat** button. File The Open default can be set to FLX, DWG, or DXF. The File Save default can be set to FLX, R12 DXF, or R12 to R2000 DWG format.  
format.

## File Exchange Troubleshooting

If you experience any difficulties working with drawing files that were developed in AutoCAD (or other compatible programs), we at ADSI certainly want to hear about them. In your email to [support@adsi-usa.com](mailto:support@adsi-usa.com) you should include a complete description of the problem so that it can be replicated and evaluated by our technicians, as well as attaching copies of the drawing files needed for this purpose.

Before contacting Technical Support, however, you should be sure to run AutoCAD's Drawing Recovery routine on any drawing (.dwg) files, as errors in these files are most often to blame for translation difficulties. Often these errors are simply unnamed drawing entities generated by third party AutoCAD applications. As AutoCAD tolerates many of these minor drawing file errors, users may only become aware that there is a problem with the file, when drawing conversion or insertion is attempted.

Users are also advised to use the latest recovery utility at hand, as earlier versions may not locate and properly identify all drawing file errors that may be present in the file.

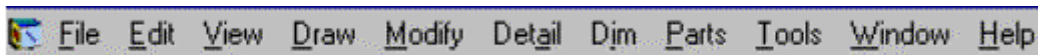
## The User Interface

### The User Interface

AllenCad includes its own default menu, commands, command aliases, accelerator keys, and floating toolbars (also referred to as palettes). Each has been carefully designed to deliver a productive and intuitive environment in which to work.

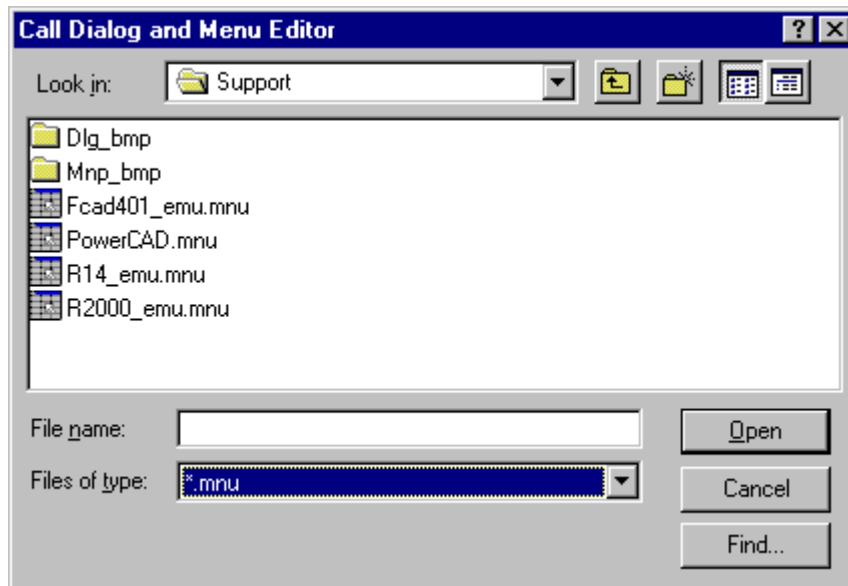
For the convenience of AutoCAD familiar operators, optional AutoCAD-emulated user interface components can be substituted for the default interface components as detailed in this section.

### Using Optional Emulation Menus

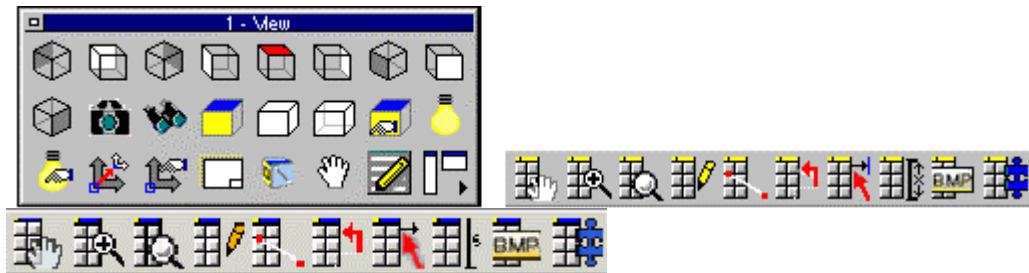


The standard AllenCad menu can be exchanged for another, using the **MENU** command as in AutoCAD. AllenCad comes supplied with additional menus (R14\_emu.mnu and R2000\_EMU.mnu) that emulate the AutoCAD 14 and 2000 pull-down menus. Menu configurations are stored in .mnu files, which can be easily modified using the Desktop and Menu editor.

Configurations are stored in .mnu files, which can be easily modified using the Desktop and Menu editor.



## Using AutoCAD Emulated Toolbars (Palettes)



For the convenience of AutoCAD-familiar operators, ten AutoCAD emulated palettes are included which can be loaded using the *palman* command. Once the palette manager is loaded, scroll to the bottom of the screen to view and/or load AutoCAD emulated palettes (named FACAD).

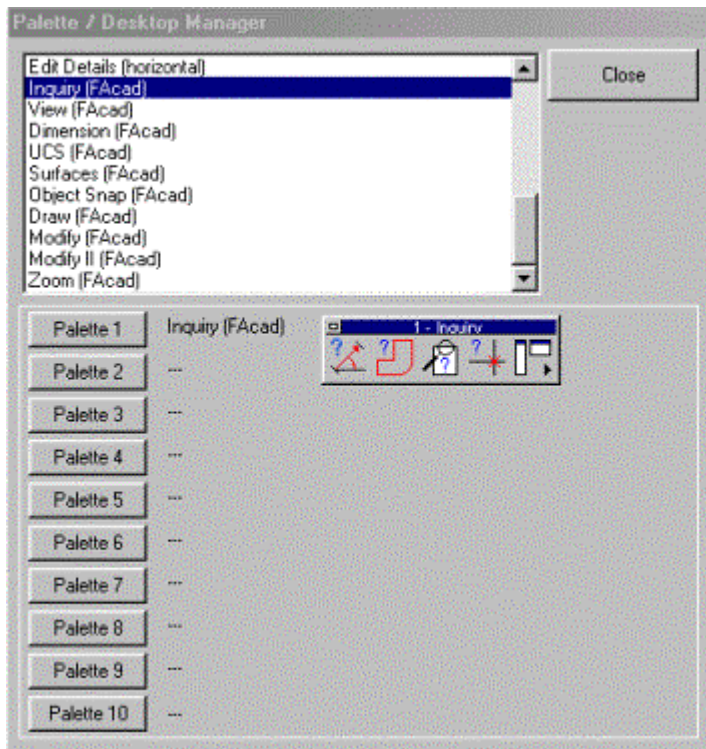
AllenCad uses its own set of floating toolbars, or palettes to speed program operation. Most palettes can be called onto the edit window by selecting the appropriate icon on the horizontal toolbar. These can be displayed horizontally or vertically onscreen by using a toggle icon located on the lower right corner of the palette.

Note that AllenCad palettes are "floating", which means that they can be placed strategically on the desktop by clicking on the blue title bar and dragging the palette to a desired position. The positions selected will be retained for future editing sessions.

## The Horizontal Toolbar



The Horizontal toolbar can be easily repopulated with different command buttons dragged and dropped from other toolbars or palettes, as described in Online Help. Command button assignments are registered in the desktop.cfg file.



## Desktop Layout

### Desktop Layout

AllenCad Pro ships with both an AutoCAD emulated desktop, as well as a AllenCad desktop. These desktops include all of the toolbar icons as well as toolbar visibility, that are default to each option. Each of these desktops can be further customized, if you wish, through the DESKCONF command, which controls bar visibility and AllenCad Pro further allows you to move and replace the command icons, through both drag and drop and dynamic button reassignment. You may also use these features to emulate other CAD desktops, or you create your own unique desktop layout, should you prefer.

Note: Both AutoCAD 15 and AllenCAD emulation menus are also available in addition to the default AllenCad menu, through the MENU or PULLDOWN commands.

If you wish to change the desktop layout between AllenCad and AutoCAD emulation after installation, you will need to note the following:

The current desktop is defined by the desktop.ini file that is located in the program's cfg directory. The two standard desktop configurations are stored in files ACAD\_Desktop.ini and AllenCad\_Desktop.ini in the same directory. To set a new desktop, copy one of the default files to a new desktop.ini file and start the program. The second step in the process is to select the appropriate menu file using the MENU command from within the program. See the Help documentation on the command for further details on the use of this command.

## Desktop Component Layout



In the *DESKCONF* dialog (pictured above), you control the display of the following desktop elements



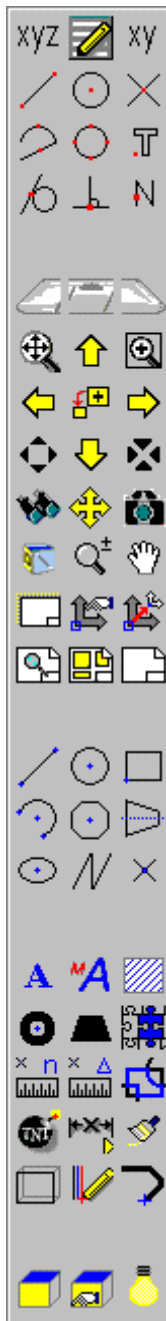
The **command line** can be turned on/off, or moved to the top or bottom of the desktop. Default location for both AllenCad and AutoCAD is bottom.



**Command Options** can be turned on/off, displayed in a pop-up option menu, or permanently displayed using the Option bar as shown here when executing the *line* command. Default for command Option display is Pop-up. To emulate the AutoCAD desktop, you would turn the Options off, however we would advise you to use either the Option menu or bar until you become familiar with all command options.



**The Properties bar**, containing continuous display of Ortho, Snap, and Grid buttons, as well as single click access to layer and property management, can be turned on or off as desired. This toolbar is unavailable as such in AutoCAD, so you might want to switch it off to provide a close emulation



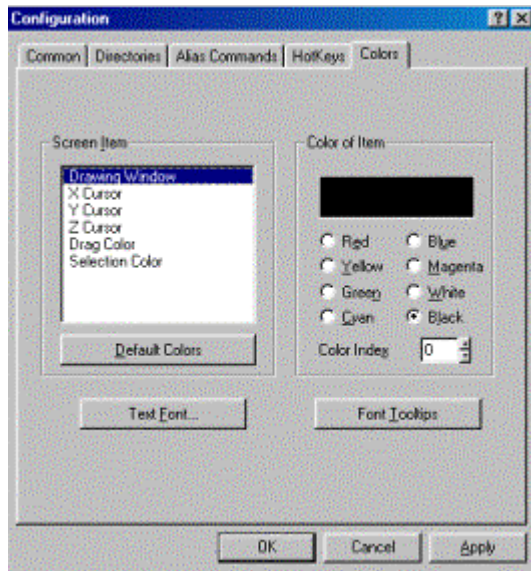
**The left Toolbar**, containing commonly used commands and transparent zoom/pan functions, can be turned on or off as desired. It can also be reconfigured or re-populated with command icons of your choice, through drag and drop methods as described in the Help file. Command button assignments are stored in the desktop.cfg file. This toolbar is also unavailable as such in AutoCAD, so you might want to switch it off to provide a closer emulation



The Status bar (bottom of screen) can be turned on or off as desired. The default setting for both AllenCad and AutoCAD is to have the status bar switched on.



## Desktop Color and Font Settings



Default desktop colors, text fonts, and font tooltips can be modified using the ***config*** command, under the ***Colors*** tab. This is the command you would use, for instance, to change the edit window background from black to white, or some other preferred color.

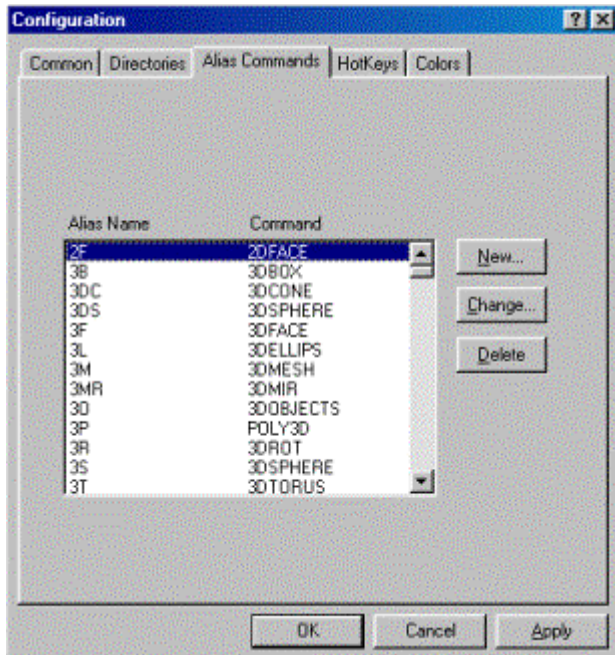
## Command Execution

### Command Aliasing and the ACAD.PGP

As with AutoCAD, all AllenCad commands can be executed by typing the command (eg. LINE), selecting the command from a pull-down menu, or by picking a desktop or toolbar icon.

With most system functions, AllenCad relays to you what is happening at the command line, or through a dialog box. Commands can be cancelled at any time by pressing the ESC key. Commands can be repeated or concluded by pressing the ENTER key, or by clicking the right mouse button.

Individual AllenCad commands are described in the included online help and included user manuals. This section describes the command aliasing system used by AllenCad.



AllenCad references a *.key* file when executing commands. A (*.key*) file is simply a text file containing a list of commands, shortcuts, and aliases used by AllenCad. The default AllenCad.key file, situated in the /Cfg folder after installation, can be viewed or edited using a standard text editor such as Wordpad®, or interactively edited using the *config* command, and accessing the *Alias Commands* tab.

The default key file (AllenCad.key) includes alias commands for all commonly used commands. In addition, all compatible AutoCAD commands are mapped directly to their AllenCad command equivalents. ***This means you can type in an AutoCAD command at the command prompt and AllenCad will automatically execute an equivalent supported AllenCad function.***

As with the AutoCAD .PGP command alias system, .key files can be edited as desired, to shorten or modify command execution. Using the example 3B for 3DBOX, the system will execute the 3DBOX function if either 3B or 3DBOX is typed at the command line. This method of command aliasing can greatly enhance your CAD productivity.

If you would normally make use of an AutoCAD .PGP file, you may wish to modify the existing AllenCad.key file, or create a new .key file containing your .PGP commands, shortcuts, and system modifications.

## Q Commands

Depending on the AutoCAD release you are familiar with, AllenCad may execute certain commands differently than you are used to. When compared with older AutoCAD releases (eg. R12 or R14), AllenCad may present a dialog box instead of a command line variant of a common command. AllenCad has been purposefully designed to provide dialog input (wherever practical), to better guide inexperienced users to the command options that are available. Normally, users prefer the simplified AllenCad dialog entry approach. Note also that users can bypass dialog entry simply by typing the option's short form designator at the command line, as you may be used to in AutoCAD.

AllenCad also includes a number of "Q" commands that when run, will cause certain commands to process similarly to a command line variant in AutoCAD. These commands bypass dialog display and

rely upon command option input instructions and are primarily provided for the use of Macro, Autolisp and experienced users.

As an example, typing the command *text* will call up the standard AllenCad text edit dialog. Typing *Qtext* on the other hand will run a command line variant of the text command, which some AutoCAD- familiar operators may prefer.

Note that Q commands can be "mapped" using the Command Alaising system described above. Refer to "Q" commands and command reference charts for further details.

## Accelerator Keys

By default, AllenCad provides AutoCAD's standard accelerator key assignments (eg. F1 for help, F2 to display an extended command line dialog, etc.). Accelerator key assignments can be very simply reassigned, using the *config* command, under the *Hot Keys* tab.

## LISP

AllenCad Pro provides full LISP support and uses a version of LISP called FLISP (Flexible Lisp) that is very similar to Autodesk's proprietary AutoLISP. There are some command variations between the two, that are documented in the Customization and LISP Programmer's Manual that is available in Adobe Acrobat (.pdf ) format in the program's \manuals directory. These function differences are fairly minor and many AutoLISP routines will run in AllenCad without editing.

## Scripts (Macros)

AllenCad Pro also provides full Script (also called Macro) support. Macros are created and used in AllenCad exactly as you are used to in AutoCAD. Macros are called, using the *MACRO* command and the files have a (.mac) filename extension, as opposed to AutoCAD's (.scr) extension.

## Other Features and Capabilities

### AllenCad Pro Extended Features List

**Drawing Navigator**™ an explorer style window, which displays all the layers, line types, blocks, text styles, views, and even coordinate systems contained in a drawing.

**Visual Block Manager** lets you quickly create graphic block libraries (supports Flexible Drawing Exchange (FLX), DWG, and DXF blocks). With the click of a mouse or digitizer button, blocks can be graphically inserted at any scale or rotation you need.

AllenCad Pro's **Image Library Tool** lets you store multiple slides and bitmaps in a single library. Supports AutoCAD® slide (.SLD) and Windows bitmap (.BMP) formats.

AllenCad Pro's **Hybrid Raster/Vector Image Overlay** capabilities allows you to integrate raster bitmaps directly into vector drawings.

AllenCad Pro's **Batch Processor** lets you perform unattended batch plotting or file conversions.

**Additional Program Highlights:**

1. Short learning curve
2. Context sensitive online help with print-ready documentation
3. Online Learning Center with instructional movies
4. Free access to Web help
5. Metric or Imperial Units
6. Fast pan and zoom based on an new 3D display list
7. Options Menus and dialog box support in many commands.
8. "Repeat previous command" function built-in.
9. Cut and Paste between drawings and the clipboard
10. Simultaneous opening of up to four drawings with up to four viewports of each
11. A full range of object drawing and editing commands
12. Context sensitive object editing (just pick an element and edit options are presented)
13. Zoom back up to 20 steps
14. Unlimited undo function
15. Icon menu support
16. Grouped functions on palettes, that can be called onto the drawing surface for quick access to commands
17. User-definable Alias commands
18. Drag and Drop user interface
19. Easy to organize part (block) libraries.
20. Preview of drawings, blocks and symbols
21. Easy definition of complex objects (parts, blocks, symbols)
22. Visual Part and Symbol Library Manager (DWG® and DXF® files supported)
23. Support for Extended Entity Data (EED)
24. Externally referenced drawings (XRef's)
25. Sophisticated object grouping
26. Associated dimensioning and hatching
27. User-definable dimensioning styles
28. Layout mode (Paper space / Model space)
29. Enhanced high speed printing/plotting with user selectable pen assignments and full plot previews
30. Batch printing/plotting and file conversion
31. Detailed plot menu with multiple configuration options

32. User-definable coordinate systems (UCS)
33. Auto-rendering based on Microsoft OpenGL® graphics
34. Fast hidden line removal
35. Interface for hybrid raster/vector applications
36. Digitizer support/Tablet menu support
37. Thickness command for common elements (3D entity extrusion)
38. Raster file import, manipulation and editing
39. Direct Windows Metafile (WMF) and Bitmap (BMP) Export

## **Customization and Programming for AutoCAD Developers**

### **Customization and Programming for AutoCAD Developers**

AllenCad Pro (and its developer-specific cousin PowerENGINE), allow complete program customization and extension using the built-in Desktop and Menu Editor (DME™), and through use of Lisp, C++, and/or Delphi programming.

Developers and users will appreciate the level of customization available with AllenCad. A variety of tools and translation utilities are included, allowing automated "porting" of customization components and code created for use with AutoCAD. In particular, AllenCad provides:

The Desktop and Menu Editor

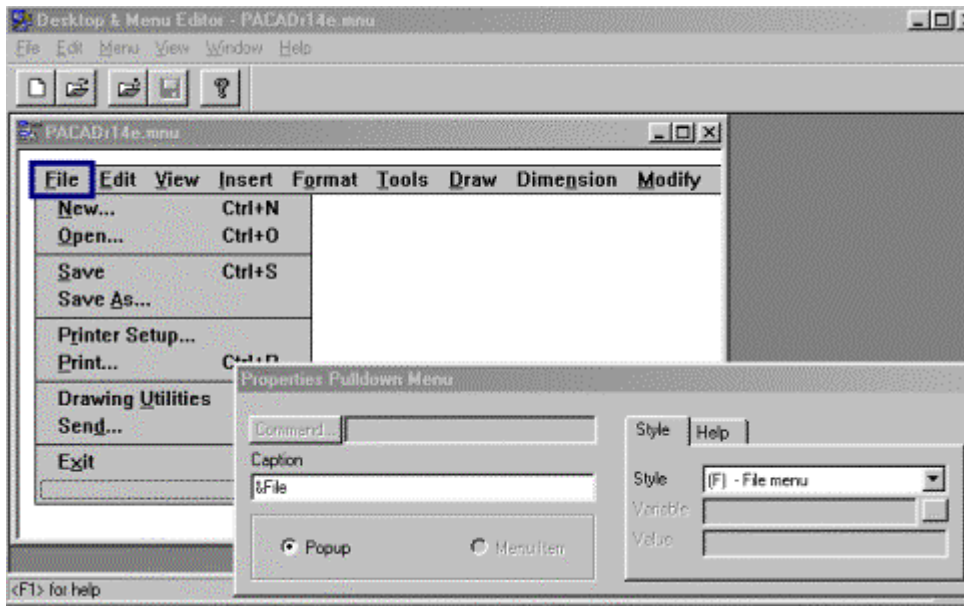
Drag and Drop Toolbar Reconfiguration

Full LISP support

ADS Compatibility

### **Desktop Reconfiguration Utilities and Capabilities**

#### **Desktop and Menu Editor**



### The Menu Editor Display

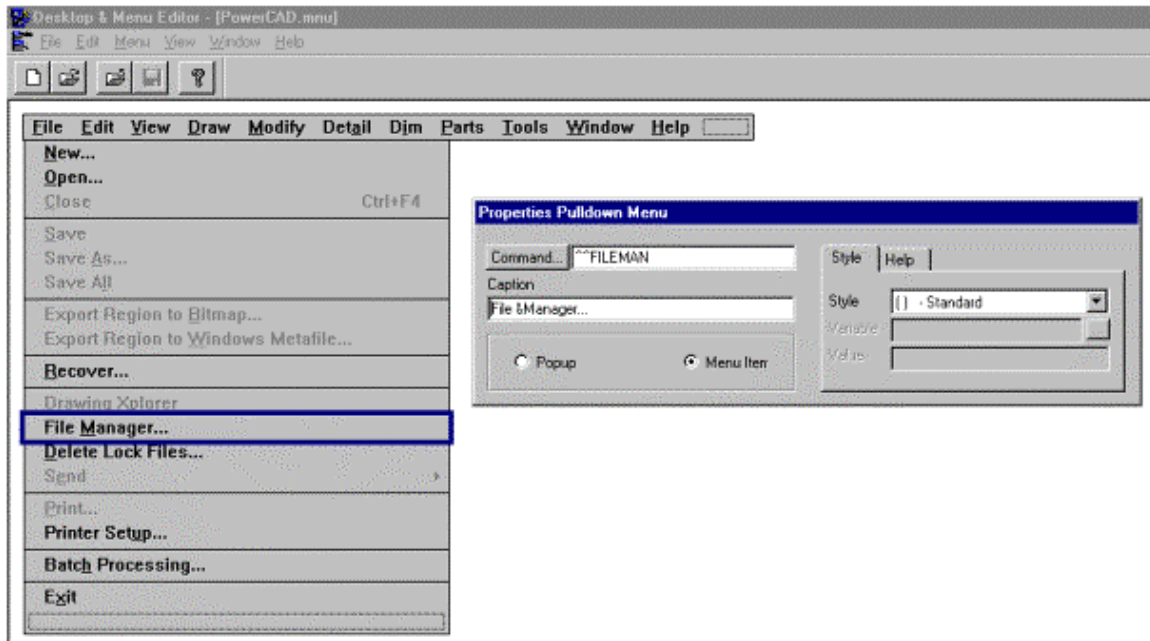
The Desktop and Menu Editor is actually a collection of three utility programs that function in a similar manner.

1. The Menu Editor is used to develop new pull-down menus (.mnu files) or to edit existing ones.
2. The Dialog Editor is used to develop and edit program dialogs and
3. The Palette Editor is used to develop and edit program toolbars (palettes)

Each of these utilities is very intuitive and easy to use. No programming experience is necessary for DME operation. DME can be run from directly inside AllenCad using the *dlgedit* command from within AllenCad, or as a stand-alone workbench using DME32.EXE.

Existing menus, dialogs, and toolbars (palettes) created for AutoCAD use can be imported or opened using DME and saved in a AllenCad-compatible format. Please refer to the Desktop and Menu Editor sections in AllenCad's online Help, Desktop and Menu Editor Help, as well as the Customization and Lisp Programming Guide for more detailed information.

### The Menu Editor



Using the Menu Editor, you can quickly modify existing menu (.mnu) files or create new ones. AllenCad .mnu files are located in AllenCad's /Support subdirectory. The display consists of two elements, the Menu Display and the Properties Pulldown Menu.

You use the menu display to arrange the layout for your menu file, using insert and delete as well as drag and drop capabilities, to create and place your menu selections.

The Properties Pulldown Menu is used to configure the selection that is currently highlighted in the Menu display. Here, you specify the command to be called, as well as the selection display parameters. Here, we have a standard menu item selected for "File Manager". If we wanted the selection to open another window containing sub-selections, we would simply select the Popup option and then edit the properties of the sub-selections we would be entering in the popup window.

Note: In the Caption Window, we place the ampersand (&) symbol immediately before the letter we wish to designate as the Alt key shortcut to the command.

## The Palette Editor

In the Palette Editor, we create and modify palette (.mnp) files. AllenCad .mnp files are located in AllenCad's /Support subdirectory.





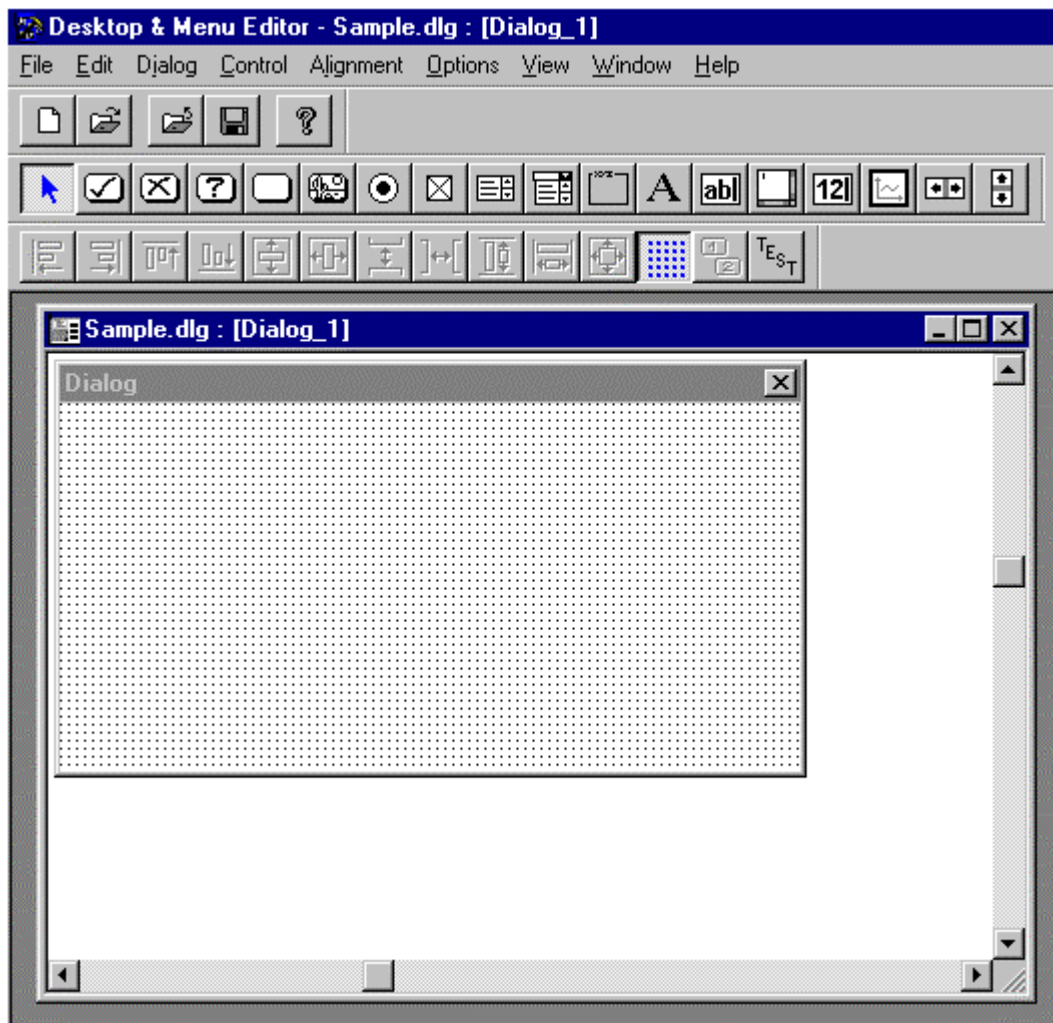
On the Palette Editor, we have the graphic representation of the palette being created, as well as a Palette Button Properties dialog, in which button parameters can be entered or edited for the currently selected button on the graphic display.

As with the Menu editor, the graphic display supports a drag and drop capability that allows you to very simply arrange your palette command buttons.

## The Dialog Editor

The Dialog Editor is used to create and modify new program dialogs. In the screen pictured below, we are opening a new dialog, represented by the central Dialog window.



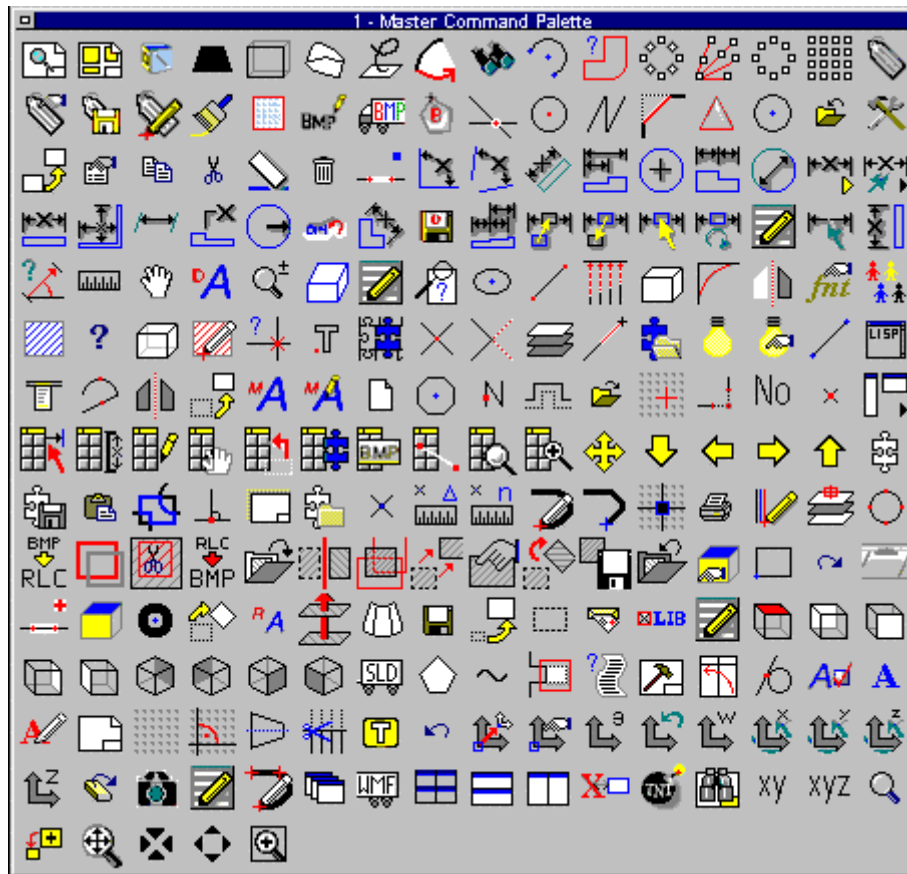


As we construct the dialog, we will click on the various dialog element buttons (located on the toolbars), that we need, and drag them into position on the dialog window. Here, they can be dragged and dropped to new positions, or directly re-sized, as required. The buttons on the lower bar can be used for more precise relative positioning and testing.

## Quick Palettes

Besides being able to quickly reconfigure the AllenCad toolbars with different command buttons, the master palette (pictured below) can be used to create new palettes within minutes. To do this, you:

1. Copy the master.mnp file to the intended new filename (.mnp)
2. Open new filename.mnp in the Desktop and Menu Editor's Palette
3. In the Editor's graphic window display of the palette, drag and drop the required command buttons into their planned locations, using the top left corner as origin.
4. Edit the Palette – Properties dialog from the pull-down menu and set the number of palette rows and columns, to remove all but the required buttons.
5. Save the file for use – Done!



## Dynamic Toolbar Reconfiguration

By default, AllenCad's Horizontal toolbar and Left Sidebar come provided with commonly used command buttons that are appropriate for a general purpose CAD application. Users and Developers alike, might prefer to replace these buttons with other command buttons more appropriate to their needs and specific application of the program.

To simplify this, AllenCad provides a simple drag and drop capability that allows you to copy or move command button to these two bars from other bar locations, or to copy buttons from the palettes. The master palette is available for this purpose, as it contains the most commonly used AllenCad commands. By default, the desktop icons are protected and cannot be moved. Simply right click on the button to edit all button properties. To delete a button, you remove all references in the button property fields.

## AutoLISP Compatibility

### AutoLISP Compatibility

AutoLISP was the first programming interface that Autodesk provided for AutoCAD. The addition of the LISP interpreter made the program more popular, since it allowed users and third-party developers to write specialized add-on programs.

AllenCad includes a full-featured LISP interpreter called FLISP (Flexible Lisp Format). It has nearly all functions available in AutoLISP. In addition, FLISP provides a number of useful enhanced functions.

Some LISP routines written for AutoCAD (and other CAD programs) can be used unmodified within AllenCad. Just as in AutoCAD, you load LISP routines into AllenCad with the load function. Applications can utilize the autoload mechanisms as described later.

In some cases, however, you may have to modify your LISP code to work correctly with AllenCad. This section provides a guide to adapt existing AutoLISP code for AllenCad implementation. Using dialog boxes with LISP is also covered later in this section.

#### **Note to Applications Developers**

This section can be used as a guide and checklist for porting existing AutoLISP applications to AllenCad. There are only a few AutoLISP functions not recognized by FLISP. Use the atoms-family function in both systems to create a pair of function lists to compare names. The (flx\_af) function lists the LISP functions in the kernel of AllenCad.

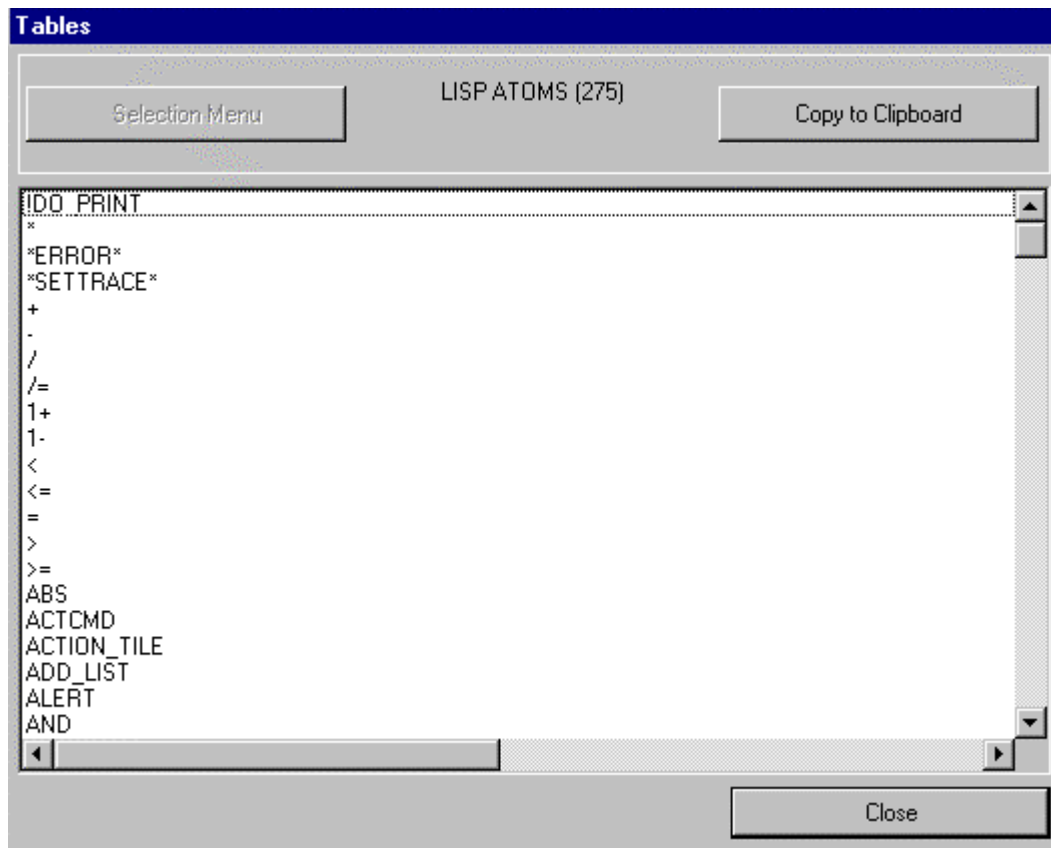


Figure 4.1: The dialog box displayed by the (flx\_af) function.

### **Command and System Variable Names**

Several LISP functions require that you pass symbolic names as arguments to the functions. In most cases, this does not create a problem for AutoLISP routines running in AllenCad. Symbol table names are the same in FLISP as in AutoLISP (for example "LAYER" and "UCS"). This is also true for object snap modes and object selection keywords.

There are, however, three functions where you need to check the symbolic names in your program code: command, setvar, and getvar. These are described in the following sections.

### **The Command Function**

AutoCAD commands are passed to AutoLISP via the command function. Many (but not all) commands in AllenCad are identical to AutoCAD, and those AutoCAD command names that have AllenCad equivalents, have been mapped to the equivalent command, meaning that most AutoCAD command names can be used in AllenCad without conversion. Equivalent commands are listed above in the Command Compatibility Chart. A similar situation exists for command options and the sequence of command requests.

## Notes to Application Developers

1. When porting your AutoLISP routines to FLISP you should first browse (or grep) your source code for command names and parameters: search for the command function. You may need to modify the code in some of these places.
2. If you want to maintain one source code for both CAD systems, in many cases you can apply functions like entmake or entmod as an alternative to the command function.

## Compounded Command Sequences

One difference between AutoLISP and FLISP is that FLISP can "leave open" a command expression. This means that the user will be prompted for all requests of the command that are not determined by the arguments of your application program. When the user terminates the command sequence, control is passed back to the LISP interpreter (your application program, in other words).

This allows certain command sequences to be compounded into one command, as shown by the following example. This FLISP routine defines a Merge command. It inserts a block into the drawing with default settings for scale factor (1:1) and rotation angle (0). The user need only determine the insertion point and (optionally) edit the block's attributes.

```
(defun C:MERGE( / title fn)
  (setfunhelp "C:MERGE" "QINSERT")
  (setq title "Insert External Part File")
  (FLX_FUNC_INIT)
  (if (setq fn (getfiled title "" "flx" 0))
    (progn
      (setvar "CMDECHO" 0)
      (command ".QINSERT" fn "_F" "1" "1" "_R" "CMDECHO-ON" "0")
    )
  )
  (FLX_FUNC_EXIT)
  (princ)
)
```

This advanced capability, unknown in AutoLISP, is based on the method of handling communications between the FLISP interpreter and the AllenCad command interpreter. Take caution, however, in applying this command functionality in code that might be used by AutoLISP

## Splitting command Expressions

In AutoLISP, you can apply the command function in a form that does not necessarily terminate in one expression. In subsequent expressions, you utilize other AutoLISP functions before continuing with

another command expression to continue and terminate the command that was started earlier. This mechanism is not provided in AllenCad. FLISP requires that you always pass a valid command name to the command function as the first argument. If this proves to be a problem, modifying the source code will always resolve the issue.

## PAUSE Symbol in Command Expressions

In AutoLISP, you can use the PAUSE symbol in command expressions to halt for user input. The PAUSE symbol causes the command interpreter to wait for a meaningful input from the user. Further processing of AutoLISP expressions will continue only after the user has provided valid data input, such as a point specification or a valid numerical value. This, however, does not work this way in FLISP.

In FLISP, you can solve the problem in several ways.

1. One method is to use the getxxx functions to get input from the user before processing the command with the command function.
2. Another method is to "open" the command sequence, as described above.
3. Another way is to use the command expression inside an FLISP function that waits for meaningful user input. In simple cases, it is a getxxx function that corresponds to the data type requested. The disadvantage is that no validity check of user input is implied. In this case; you can work around this by using application-defined functions in these locations that substitute for the AutoLISP PAUSE.

## FLISP Functions with Alternate Names

Programmers should note that some FLISP functions may have a different name from the AutoLISP function, but that they operate in the same way.

## Proprietary AutoLISP Functions Names

Some FLISP functions have different names because the AutoLISP function names contain the word "acad," which is short for AutoCAD, or "ads," which is short for AutoCAD Development System. Both are proprietary to Autodesk. The following table lists the alternate FLISP function names for these AutoLISP functions. Notice that the syntax is identical:

### FLISP Functions with Alternate Names

AutoLISP Function	FLISP Function	Comments
Acad_strlsort	Stringsort	Sorts a list of string-items ascending alphabetically. In AutoCAD, AcadApp defines an external ARX application. In AllenCad, stringsort is the native FLISP function.
Syntax: (acad _strlsort list)	Syntax: (string list)	
Acad_colordlg	Getcolorindex	Displays the Colors dialog box, allowing the user to select a color. In AutoCAD, AcadApp defines an external ARX application. In AllenCad, getcolorindex is the native FLISP function.
Syntax: (acad_colordlg color_no [flag])	Syntax: (getcolorindex [color_no] [flag])	
Acad_helpdlg	Help	Displays a help topic. In AutoCAD, acadrl4.lsp defines an external AutoLISP application.
Syntax:	Syntax:	

<code>(acad_helpdlg helpfile topic)</code>	<code>(help [topic][helpf ile] [command])</code>	In AllenCad, help is the native FLISP function.
Ads	Fdt	Returns a list of strings containing the loaded externally defined applications.

## Differences in Error Tracing Functions

FLISP uses different names for AutoLISP's tracing (debugging) functions, as shown by the table below. AllenCad provides the `(*settrace* number)` function for error tracing LISP routines and functions.

### Different Error Tracing Function Names

AutoLISP Function	FLISP Function	Comments
Trace	settrace	The trace function is not supported. Use the <code>(*settrace* 16)</code> function. Note the difference in trace controlling: <code>(lambda(fcn) (*settrace* 16) fcn)</code>
Untrace	settrace	The untrace function is not supported. Use the <code>(*settrace* 0)</code> function. Note the differences in trace controlling: <code>(lambda(fcn) (*settrace* 0) fcn)</code>

## Alternate Functions for Applications Loading

Since AllenCad does not support ObjectARX applications, you need to replace these AutoLISP functions in your code. In general, use the *xload* group of functions to load FDT applications, which are the equivalent to ADS in AllenCad. The alternate functions are listed in the following table

AutoLISP Function	Comments
Arxload	Replace with the FLISP xload function.
Arxunload	Replace with the FLISP xunload function.
Autoarxload	Replace with the FLISP autoxload function.

## Unsupported AutoLISP Functions

The following table lists AutoLISP functions that are not recognized when run inside of AllenCad:

### Unsupported AutoLISP Functions

AutoLISP Function	Comments
Menucmd	<p>The menucmd function was used in DOS-based AutoCAD versions to display a context-sensitive side menu.</p> <p>This function is not supported in FLISP.</p> <p>As an alternative, use the initget function with flag 256 to display AllenCad's option menu.</p>
Menugroup	<p>This function is not supported in FLISP because menu groups are unknown in AllenCad.</p> <p>For information about AllenCad's concept of menus and other customization items, see Chapter 3 "Customization Compatibility."</p> <p>As an alternative, use palettes (toolbars) with the Palette command.</p>
Ssname	<p>This function is not supported in FLISP.</p> <p>As an alternative, if you only need to get a list of entities inside of the selection set, use a while loop with the sname function.</p>
Ssgetfirst	Grips are not yet supported.
Sssetfirst	Grips are not yet supported.
Tablet	In AutoCAD, the tablet function retrieves and establishes digitizer calibration.

## FLISP Functions With Different Arguments

Some FLISP functions have different arguments from the AutoLISP function of the same name. The following table lists those functions. Notice that the function names are identical:

### FLISP Functions with Different Arguments

AutoLISP Function	FLISP Function	Comments
Gread	gread	<p>Check the arguments.</p> <p>Note that this function works differently in AutoLISP and FLISP.</p>
Grtext	grtext	<p>Check the arguments.</p> <p>This function is only for output at the status line.</p>
Nentselp	nentselp	<p>Take care with the transformation matrix, which is different.</p> <p>See also the MatrixMode system variable.</p>
Ver	ver	<p>Returns the version number of the LISP interpreter of the current program release.</p> <p>For example, "5.0" in AllenCad.</p>

### The getenv Function

In AutoCAD, the getenv function retrieves the value of a specified DOS environment variable. In FLISP, it does not take an argument and retrieves the path settings from the *AllenCad5.ini* file (found in the \Windows folder) as a list of strings.

## Memory Management Functions Not Supported

Some functions are rarely used in AutoLISP. They are not supported by FLISP because the AllenCad implementation of the LISP interpreter and its memory management does not need them. The following table lists such memory management functions not found in FLISP

### Rarely-used AutoLISP Functions Unsupported by FLISP

AutoLISP Function	Comments
Alloc	Remove this function or replace it with a dummy: (lambda (x) x)
Expand	Remove this function or replace it with a dummy: (lambda (x) x)
Xdroom	The xdata size of EEDs in AllenCad is theoretically unlimited (limited only by disk space). Remove this function or replace it with a dummy: (lambda (ent) 65712)
Xdsize	The xdata size of EEDs in ALLENCAD theoretically unlimited (limited only by disk space). Remove this function or replace it with a dummy.

## Dictionary Functions Not Supported

The dictxxx dictionary functions in AutoLISP are strongly comparable to the tblxxx table functions found in FLISP. In many cases, you can apply the tblxxx functions similarly or in the same way as those listed in the table below.

The corresponding functions are not identical, however, and the management of dictionaries in the drawing database is different from table handling in the AllenCad drawing database. For this reason, there is a loss of non-graphical application-specific data when exchanging drawings between the two systems.

### AutoLISP Dictionary Functions Unsupported by FLISP

AutoLISP Function	Comments
Dictadd	Substitute with the tblmake function in table "APPDATA".
Dictdel	Substitute with they tbldel in table "APPDATA".
Dictnext	Substitute with the tblnext in "APPDATA".
Dictremove	Substitute with the tbldel in table "APPDATA".
Dictrename	Substitute with the tblmode and tblrename in table "APPDATA".
Dictsearch	Substitute with the tblsearch in table "APPDATA".
Namedobjdict	Substitute with the tblxxx functions in table "APPDATA".
Entmakex	Use entmake in place of this function.



For non-graphical application data, use the tblxxx function and table "APPDATA".

## Notes to Applications Developers

1. The AllenCad drawing database contains a table called "APPDATA" that stores non-graphical, application-specific data.

The "APPDATA" table can be read and manipulated, just like the other symbol tables (such as Layer, Style, Ltype, Dimstyle, and View) with the FLISP functions that handle named objects in tables (tblnext, tblsearch, tblobjname, snvalid, etc.)

### Enhanced FLISP (FLEXIBLE LISP FORMAT) Functions

AllenCad has enhanced several FLISP functions, as described in the following sections.

### The defun Function

You use the defun function in FLISP as you do in AutoLISP. The enhancement in AllenCad is that FLISP allows you pass a variable number of arguments to the defun function.

### Additional FLISP Functions

The functions listed below do not have a direct AutoLISP equivalent:

#### Additional Functions in FLISP

FLISP Functions	Comments
Actcmd	Reactivates a built-in command of PowerENGINE ( AllenCad's Graphic Developer's Engine), when it had been previously deactivated by function delcmd.
Delcmd	Deactivates (deletes) a built-in PowerENGINE command. This disables the command for the user. The argument specifies the name of the command.
Entcheck	Verifies that the entity specified by ename is (still) valid in the current drawing database. Returns the entity name if ename is valid or is contained in a block definition of the current drawing; otherwise it returns nil.
Entpos	Sets the database pointer in the entity section to an absolute or relative position. Returns the name of the entity found at that position.
Symbtos	Returns any LISP expressions in a string.
Tan	Returns the tangent of an angle.
Tbldel	Deletes a record (specified by the name of the item) in a specified table a record, if the table entry is not referenced in the drawing.
Tblmake	Generates a new table-entry, as described by an association list.
Tblmod	Modifies an existing table-entry, as described by an association list.
Tblpurge	Purges an entire table of the specified type and deletes all not-referenced entries.
Tblrename	Allows the renaming of a table entry. The entry old_name is replaced by the entry new_name.
Tblset	Sets a table entry as current entry. In the table specified by table_type the entry item_name

is set as current (default or active). An associated system variable is updated.

## **Autoload Mechanism**

AllenCad provides two ways to automatically load application specific routines:

autoload and autoloading: These functions are provided and works virtually identically to AutoCAD (in a LISP routine itself).

S::STARTUP mechanism: You can apply startup function S::STARTUP in FLISP, just as in AutoLISP.

Menuload mechanism: LISP code, included with an .mnl file, is loaded automatically when a menu is called with the same name as the .mnl file. This mechanism corresponds to the method used in AutoCAD.

## **Protecting Lisp Routines**

The Protect program that comes with AllenCad can be used to encrypt your LISP routines so that they cannot be read with a text editor.

Please note that you cannot use protected FLISP routines in AutoCAD. Similarly, you cannot run AutoLISP files that have been protected by AutoCAD.

## **The SETVAR and GETVAR Functions**

The first argument that must be passed to the setvar and getvar functions is the name of a system variable.

Please note in this context:

Not all AutoCAD system variables are supported in AllenCad.

A few system variables have a different name in AllenCad but have the same purpose in AutoCAD.

AllenCad has some system variables that are unknown to AutoCAD. You can probably use these for special purposes in your application.

### **Notes to Application Developers**

When porting your program code written for AutoLISP/ADS, first browse (or "grep") your code for locations where you may have used the setvar and getvar functions.

In FLISP, the command function serves the same purpose as in AutoLISP. There are, however, some differences, which are detailed in the following sections.

## **ADS Compatibility**

### **Understanding ADS Compatibility**

AutoCAD provides an API (application programming interface), which allows third party developers to write applications in C/C++. In AutoCAD, it is called ADS, short for "AutoCAD Development System."

The AllenCad equivalent is called FDT, short for "Flexible Developer's Toolkit." FDT is an API with outstanding compatibility to the ADS interface. (AllenCad does not support the equivalent of AutoCAD's ObjectARX.)

Numerous third-party developers provide applications built for AllenCad based on FDT — either as AllenCad add-on applications, or as stand-alone OEM applications.

The majority of these developers use C and C++ (Microsoft Visual C/C++ Version 6 or higher is required) to build their applications.

The primary difference between FDT and ADS functions is that FDT uses the `fdt_` prefix, whereas ADS uses the `ads_` prefix. Please note that:

ADS has some function names that use other prefixes, such as `acad_` and `acrx_`.

FDT's dialog-related functions use the `dlg_` prefix.

The differences between FDT and ADS functions are listed in the tables found in "Dialog Functions in FDT". Dialog box functions are listed in "Dialog Box Functions in FDT".

#### **The `ads2fdt.h` Include File**

AllenCad provides the `ads2fdt.h` include file, which redefines `ads_` function names to their `fdt_` equivalents.

The `ads2fdt.h` header file and libraries are provided with AllenCad Pro and the PowerENGINE Software Development System.

### **Porting ADS Applications to FDT**

To execute existing applications in FDT, simply recompile the source code using the `ads2fdt.h` header file and link the FDT libraries.

For more information, read the online documentation of the FDT function library provided as Acrobat® Acrobat file: `fdt40.pdf`. Also, have a look at the `ads2fdt.h` file.

### **Initialization and Definition Differences**

Please note the differences in the application initialization between ADS and FDT, as shown in the following table.

#### **Initialization and Definition Differences**

##### **ADS Initialization**

```
AcrxEntryPoint(AcRx::AppMsgCode msg,  
void*);
```

```
Void ads_init (int argc, char  
*argv[]);  
Int ads_link (int cbc);
```

##### **FDT Initialization**

```
FDT_INIT  
int fdt_app_init(void);  
int fdt_app_exit(void)
```

```
Int ads_defun _((const char *sname,
short funcno));
```

```
Int ads_regfunc (int (*fhdl) (void),
int fcode);
```

```
int fdt_register (const char
*CommandName, const char
*FunctionName);
```

See also: `fdt_register_param()`

The following code fragment provides a template for FDT-based applications and demonstrates the mechanism of initialization and registering functions:

```
#include <windows.h>
#include "fdt.h"
/* Initialization of the Windows DLL (defined in fdt.h) */
FDT_INIT
/* DLL's LibMain (defined in fdt.h) */
/* Initialization of the application's DLL */
/* Required to be contained in any application! */
int EXPORT fdt_app_init()
{
int iCode;
/* Registering the internal function "func_1" (function name f1) */
iCode = fdt_register("func_1", "f1");
/* Registering the external function "c:func_2" (function name f2)*/
iCode = fdt_register("c:func_2", "f2");
/* Further FDT statements may follow */
.
.
.
/* DLL is loaded in "preload-mode"! = return 1! */
return RTPRELOAD;
}
/* De-Initialization of the application's DLL (exit function) */
/* Required to be contained in any application! */
int EXPORT fdt_app_exit()
{
/* E.g. release memory */
.
.
.
return 0;
}
int EXPORT f1(void)
{
fdt_printf("Internal Function func_1 (Procedure Name f1)");
.
.
.
return 0;
}
int EXPORT f2(void)
```

```

{
    fdt_printf("External Function c:func_2 (Procedure Name f2)");
    .
    .
    .
    return 0;
}

```

## Functions with Minor Differences

Not all FDT functions are completely equivalent to ADS functions. Some are different; however, these functions fulfill the same purpose. Usually, modification of your source code can be done without much effort. Conditional compile is recommended in these cases.

### Note to Applications Developers

FDT has its own memory management. This means that memory dynamically allocated with the functions `fdt_malloc()`, `fdt_strmalloc()`, `fdt_getvar()`, and `fdt_fgetenv()` can only be released by `fdt_free()` and `fdt_strfree()`, respectively.

The following table compares ADS and FDT functions that are partially compatible:

ADS Function	FDT Function	Comments
<code>void *acad_malloc (size_t sz)</code>	<code>void *fdt_malloc (unsigned int sz)</code>	Both functions allocate a region of memory for an array.
<code>int ads_getenv (const char *VarName, char *Result)</code>	<code>int fdt_getenv (char *VarName, char *Ini_section, char *Ini_file, int length, char *Result)</code>	The ADS function retrieves a value from the current environment. The FDT function gets a value from the current environment or from a named INI file. See also <code>fdt_fgetenv()</code> .
<code>int ads_getcfg (const char *sym, char *var, int len)</code>	<code>int fdt_getcfg (const char *sym, char *var, int len);</code>	The ADS function retrieves application data from the AppData section of the <code>acad.cfg</code> file. The FDT function retrieves application data from the AppData section of the <code>appdata.ini</code> file.
<code>int ads_getcname (const char *Cmd, char **Result)</code>	<code>int fdt_getcname (const char *Cmd, char **Result);</code>	The ADS function retrieves the localized name of a command from the language independent name of a command and vice versa. The FDT function retrieves the primary command name from a localized name.
<code>int ads_getfiled (const char *Title, const char *Default, const char *ext, int flags, struct resbuf *Result)</code>	<code>Int fdt_getfiled (const char *Title, const char *Default, const char *ext, int flags, struct resbuf *Result);</code>	The ADS function allocates memory for the path name string in the Result argument. The FDT application must allocate sufficient memory for the result buffer where the filename is returned (including the path). To do so, use <code>fdt_strmalloc()</code> .
<code>int ads_grtext (int Box, const char *Text, int hl)</code>	<code>Int fdt_grtext (char *Text)</code>	The ADS function displays the specified text in the menu area or in the status area. The FDT function always displays the

```
int ads_initget (
    int val,
    const char *kwl);
```

```
int ads_setcfg (
    const char *sym,
    const char *val);
```

```
int ads_setfunhelp (
    const char *FunctionName,
    const char *HelpFile,
    const char *Topic,
    int iCmd);
```

```
int ads_setview (
    const struct resbuf *view,
    int vport);
```

```
struct resbuf *ads_tblsearch (
    const char *TableName,
    const char *Sym,
    int setnext)
```

```
Int fdt_initget (
    int val,
    const char *kwl);
```

```
Int fdt_setcfg (
    const char *sym,
    const char *val);
```

```
int fdt_setfunhelp (
    char *FunctionName,
    char *HelpFile,
    char *Topic);
```

```
int fdt_setview (
    const struct resbuf *view,
    int vport);
```

```
Struct resbuf * fdt_tblsearch (
    char *TableName,
    char *Sym)
```

text in the status line.

The FDT function uses a bit-coded sum of val with the control bit 256 to display keyword(s) in the options menu.

The ADS function writes application data to the AppData section of the acad.cfg file.

The FDT function writes application data to the AppData section(s) of the appdata.ini file. The .ini file is created automatically in the configuration directory, if it does not exist.

In ADS, the parameter iCmd must be in accordance to the ads\_defun() parameter, which identifies the function in subsequent kInvkSubrMsg message from AutoLISP.

This parameter is not required in FDT.

In ADS, if the argument vport is set to 0, the current viewport receives the new view. AutoCAD counts viewport numbers beginning with 2, in ascending order.

In FDT, if vport is -1, the current viewport receives the new view. AllenCad counts the viewport numbers beginning with 0 in ascending order

In ADS, the setnext argument specifies whether the ads\_tblsearch() call affects the next call to ads\_tblnext() or not.

FDT does not provide a default sequence for fdt\_tblsearch().

## ADS Functions Without Direct Equivalents in FDT

Some ADS functions are not supported in FDT. These include the functions as listed in the following table. Where applicable, you find information on "work-arounds"

### ADS Functions Not Supported by FDT

ADS Function	Comments
acad__msize	Use standard C commands.
acad__strdup	Use standard C commands.
acad_calloc	Use standard C commands.
acad_realloc	Use standard C commands.
ads_abort, acrx_abort	Not supported.
ads_asetenv	Use standard C-commands.
ads_dictadd	Dictionary function.
ads_dictnext	Dictionary function.

ads_dictrename	Dictionary function.
ads_dictsearch	Dictionary function.
ads_entmakex	Dictionary function.
ads_exit	Not supported.
ads_getappname	Not supported.
ads_getfuncode	ADS interface function; not required in FDT.
ads_getstringb	Of minor importance (ads_getstring with buffersize); use fdt_getstring().
ads_link	ADS interface function; not required in FDT.
ads_menucmd	Functionality is implemented by fdt_initget().
ads_namedobjdict	Dictionary function.
ads_regappx	Dictionary function.
ads_ssgetfirst	Grips related function; not supported.
ads_ssnamex	Selection method information; not supported.
ads_sssetfirst	Grips related function; not supported.
ads_tablet	Not supported.
ads_xdroom	In FDT, extended entity data size is unlimited; not required in FDT.
ads_xdsize	In FDT, extended entity data size is unlimited; not required in FDT.
ads_xstrcase	Not supported.
ads_xstrsave	Not supported.
adsw_acadDocWnd	Not supported.
adsw_acadMainW	Use fdt_gethandle() to get main window handle.

## Functions Unique to FDT

The following table lists functions that are unique to FDT:

### FDT Functions Not Found in ADS

FDT Function	Comments
fdt_actcmd()	Reactivates a built-in command which had been previously deactivated by fdt_delecmd().
fdt_addimage2lib()	Adds an image file (BMP or WMF) to the specified image library. Image libraries .ilb files store multiple image files within one library file.
fdt_appinters()	Returns the implied (apparent) intersection point of two entities found in the specified snap points pt1 and pt2.
fdt_boundingbox()	Returns the lower left and the upper right corners of the smallest box that encloses the entire drawing, a selection set, or an entity.
fdt_bspline2poly()	Converts a B-spline curve back to a regular 2D polyline.
fdt_call_notify_app()	Notifies an application of critical operations, including New, Open, Save, Close, and Exit.
fdt_catrom2poly_2d()	Converts a 2D Catmull-Rom curve back to a 2D-polyline.
fdt_compfont()	Compiles a font source file (.shp or .fnt) to a system binary font file (.fsh).
fdt_copyresbuf()	Copies a linked result buffer list. Copies the result buffer source_rb and returns in target_rb the start address of the new allocated result buffer.
fdt_deldbmodhook()	Deletes (uninstalls) a registered database modification hook function (removes it from the hook registration list).
fdt_delimagefromlib()	Removes a specified image from the specified image library .ilb file.
fdt_delmousehook()	Deletes the current active hook function, which had been installed to retrieve

	mouse click event information.
<code>fdt_deselecthook()</code>	Deletes (uninstalls) a registered hook function for entity selection control.
<code>fdt_delcmd()</code>	Deactivates (deletes) a built-in CAD-Engine command. This disables the command for the user.
<code>fdt_entcheck()</code>	Verifies if the drawing entity specified by <code>ename</code> is (still) valid in the current drawing database.
<code>fdt_entget_dbhook()</code>	Returns the defining data of a drawing entity specified by <code>ename</code> contained in the current temporary (virtual) Update Database.
<code>fdt_entmod_dbhook()</code>	Allows modification of the defining data of an entity that is a member of the current Update Database based upon the information supplied as the linked result buffer list.
<code>fdt_entmod_type()</code>	Enables requesting the type of modification. This function can only be used within a database modification hook function!
<code>fdt_entnext_dbhook()</code>	Retrieves an entity name <code>next_ename</code> (which follows the entity specified by <code>ename</code> ) from the temporary (virtual) Update Database, which is established temporarily as soon as a user command or program operation is going to modify the drawing database to allow to control database modification by accordingly hook functions.
<code>fdt_entpos()</code>	Sets the database pointer in the entity section to an absolute or relative position and returns the name of the entity found at that position.
<code>fdt_exit_dragmode()</code>	Terminates the usage of a drag mode previously installed by <code>fdt_init_dragmode()</code> .
<code>fdt_flxclose()</code>	Closes the current drawing (with or without saving).
<code>fdt_flxcloseviewport()</code>	Close the current viewport.
<code>Fdt_flxnames()</code>	Returns a list containing the database numbers and the filenames of the drawings currently opened in a linked result buffer list.
<code>fdt_flxnew()</code>	Opens a new drawing specified by filename.
<code>fdt_flxopen()</code>	Opens an existing drawing specified by filename in read-write or read-only mode.
<code>fdt_flxsave()</code>	Performs a quick save of the current drawing in .flx drawing format.
<code>fdt_flxsaveas()</code>	Saves the current drawing as specified by filename.
<code>fdt_getactviewport()</code>	Returns the ID of the current drawing and its current viewport.
<code>fdt_getcolorindex()</code>	Displays the Colors dialog box to retrieve a color number specified by the user.
<code>fdt_getcursor()</code>	Retrieve the current cursor style setting: crosshair, rubberband, snap box, select box, or window. To set a new cursor, see the function <code>fdt_setcursor()</code> .
<code>fdt_getflxinfo()</code>	Reads drawing annotations stored in the FLX drawing file.
<code>fdt_getimagetype()</code>	Retrieve the type of an image stored within the specified Image Library (.ilb file).
<code>fdt_hatch()</code>	Hatches the entities of a selection set specified by <code>sname</code> .
<code>fdt_init_dragmode()</code>	Allows an application to utilize several drag or rubberband modes
<code>fdt_input()</code>	A very powerful general user input function: reacts directly on user input. It has some characteristics in common with <code>fdt_gread()</code> , but allows the user to input object snap options and transparent commands, and apply them by the application.
<code>fdt_norm_dxfcode()</code>	Normalizes a given DXF Group Code to a standard result type (RTINT, RTSTR ...) as defined within the include file <code>fdtcodes.h</code> .
<code>fdt_pcontour()</code>	Create a polyline boundary contour.
<code>fdt_poly2bspline()</code>	Converts a 2D polyline to a B-spline curve.
<code>fdt_poly2catrom_2d()</code>	Converts a 2D polyline to a 2D Catmull-Rom curve.
<code>fdt_purgedbmodhooklist()</code>	Deletes all registered hook functions for drawing database modification control.
<code>fdt_purgeselecthooklist()</code>	Deletes (uninstall) all registered hook functions for entity selection control.
<code>fdt_set3dview()</code>	In a single operation, set a 3D view direction and zooms to a specified region.



<code>fdt_setactvport()</code>	Sets another drawing as current drawing, or sets another viewport as active by determining the ID of a currently opened drawing and the ID of the drawing's viewport.
<code>fdt_setcursor()</code>	Sets a new cursor style for successive operations.
<code>fdt_setdbmodhook()</code>	Registers and installs a hook function to control entity modification of the drawing database.
<code>fdt_setflxinfo()</code>	Writes new drawing information to an FLX drawing file.
<code>fdt_setmousehook()</code>	Registers a hook function, which is called with any mouse click to a drawing window (mouse click event in the drawing).
<code>fdt_setselecthook()</code>	Registers and installs a hook function <code>fnSelHook</code> to supervise entity selection.
<code>fdt_ssdbnr()</code>	Returns the database ID to verify for which drawing the specified selection set name <code>sname</code> is valid.
<code>fdt_strfree()</code>	Releases the memory of a string allocated by the function <code>fdt_strmalloc()</code> .
<code>fdt_strmalloc()</code>	Allocates memory of a string in certain applications, The function must be used to modify variables of type <code>rstring</code> in a result buffer list of type <code>struct resbuf</code> .
<code>fdt_tbldel()</code>	Deletes from a specified table a record specified by the name of the item, but only if the table entry is not referenced in the drawing.
<code>fdt_tblmake()</code>	Creates a new table entry. In the current drawing, a new table item is created based on the data contained in the result buffer list <code>tbl_record</code> .
<code>fdt_tblmod()</code>	Modifies a table record.
<code>fdt_tblpurge()</code>	Purges an entire table of the specified type and deletes all not-referenced entries.
<code>fdt_tblren()</code>	Renames a table entry (table item).
<code>fdt_tblset()</code>	Sets a table entry as current entry. In the table specified by <code>tbl_type</code> the entry <code>item_name</code> is set as current (default or active). An associated system variable is updated automatically.
<code>fdt_wmfout()</code>	Writes the specified area of a drawing to a Windows metafile (Aldus Metafile).

# Chapter 6 AllenCad and PowerENGINE for FelixCAD Users

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## Introduction

AllenCad Pro™ is a re-branded and dramatically enhanced version of FCAD, or FelixCAD® by Felix Computer Aided Technologies. Based on core technology refined over 7 years and several major releases, AllenCad Pro has evolved into the world's best value in flexible, powerful, and industry compatible computer-aided design processing.

AllenCad Pro is also available to as the affordable and completely scalable PowerENGINE™ Software Development System, whereby all or selected components of PowerENGINE can be embedded into vertical market solutions ranging from traditional AEC, GIS, and manufacturing applications to non-traditional business applications such as automated sales and wireless stock trading systems.

FCAD 4.01 users and developers will be instantly familiar with AllenCad Pro operation and customization techniques, and will appreciate the significant improvements available to them. By default, AllenCad utilizes the open and published FCAD flexible file format (.FLX), with direct opening and saving of AutoCAD DWG® and DXF® files up to R2000 format.

### Significant Enhancements in AllenCad R5 over FCAD 4.01 include:

AllenCad features direct support for AutoCAD drawing files up to R2000 format. Users can now directly open DWG and DXF files from Release 2.5 through Release 2000, with "round-trip" retention of crucial R2000 entities. Files can be saved directly to R12 DWG/DXF through R2000 DWG format for shared AllenCad/AutoCAD work places.

AllenCad features an improved and simplified default interface, with optional FCAD 4.01 and AutoCAD R14 emulated menus and toolbars.

AllenCad features direct support for all common FelixCAD and AutoCAD commands, with virtually no re-learning required by the user.

AllenCad supports Multiline Text (MTEXT), with a built-in MTEXT editor. AllenCad also adds support for Dynamic single line Text (DTEXT).

AllenCad offers True Type Font support, and SHX vector fonts (AutoCAD compatible) with Bigfonts.

AllenCad features enhanced OFFSET and HATCH functionality.

AllenCad supports Light Weight polylines.

AllenCad includes a new Dynamic View (DVIEW) function, with perspective viewing and bitmap creation using rendered or hidden line imaging.

AllenCad features rigorous drawing recovery and "clean up" through use of enhanced purging mechanisms.

AllenCad includes the new DynaSnap™ system to enhance productivity through the use of dynamic object snaps with accompanying tooltips.

AllenCad is faster and more efficient in virtually all areas of operation, including graphics display and regeneration.

AllenCad includes over 1,000 new pages of enhanced user documentation in a new Winhelp 2000 online help system, accessible through the Help menu, by pressing F1, or by clicking the question mark (?) in most system dialogs. The new Learning Center is designed to assist both new and experienced users in operation of the program.

Many more enhancements are contained, as outlined below and in the new online help system included with AllenCad.

**Note:** The default Desktop has changed considerably to give greater functionality, with new commands and features added to improve productivity. Even greater configurability and legacy support exists over what you have come to expect in FelixCAD. All drawing files and FLISP (Flexible Lisp) routines are directly portable from FelixCAD to AllenCad without any conversion.

For those users who may be more familiar and comfortable with the FCAD 4.01 menu system, an FCAD emulated menu file has been provided for use in AllenCad. It can be loaded, using the MENU command and will remain until changed. Similarly, FCAD palettes have been supplied with the program. To configure them for use, you will need to edit the command icon properties on the toolbar and modify the menu selection properties, using the Desktop and Menu Editor. Full details on these procedures are available in the Program Help file.

The following topics describe significant new features and enhancements that you will want to be familiar with, both from the user and program development/customization standpoint.

## User Enhancements

### New Entities

#### Lightweight Polyline

AllenCad supports the optimized LWPOLYLINE-Entity. The creation of lightweight or "old-style" polylines can be controlled with the global system variable PLINETYPE.

This variable specifies whether AllenCad uses optimized 2D polylines. PLINETYPE controls both the creation of new polylines with the PLINE command and the conversion of existing polylines in drawings from previous releases.

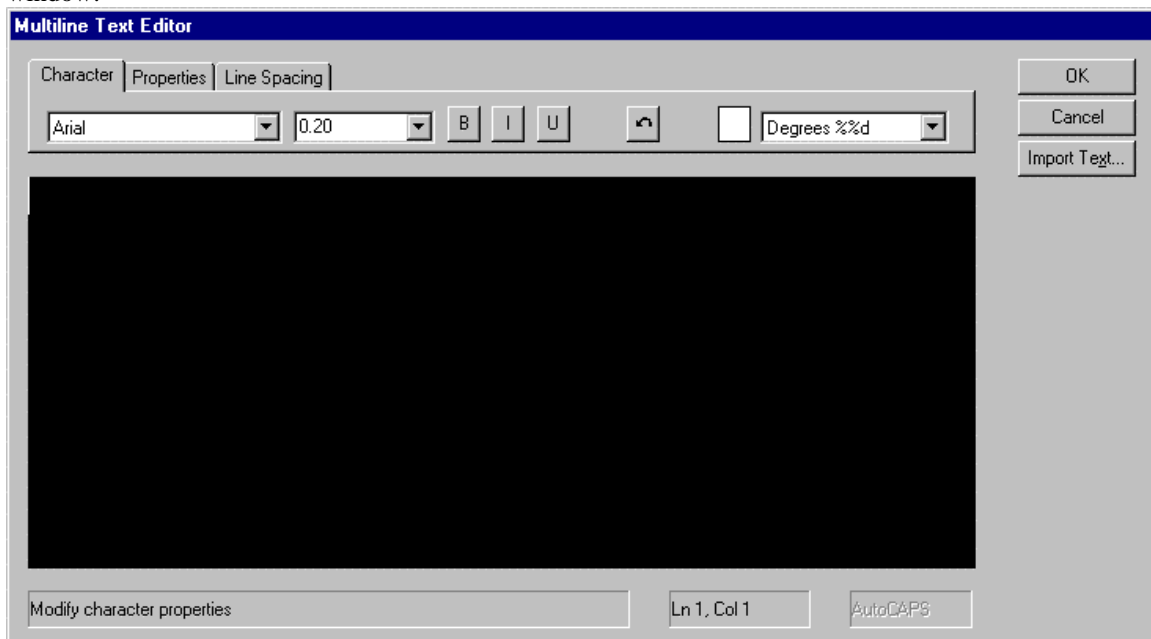
- |   |   |
|---|---|
| 0 | Polylines in older drawings are not converted when opened; PLINE creates old format polylines |
| 1 | Polylines in older drawings are not converted when opened; PLINE creates optimized polylines  |
| 2 | Polylines in older drawings are converted when opened; PLINE creates optimized polylines      |

### New Commands

#### Command MTEXT

AllenCad fully supports MTEXT. After issuing the command and specifying the location of the new text, you are presented with a dialog in which you enter your text. Three tabs allow you to exercise control

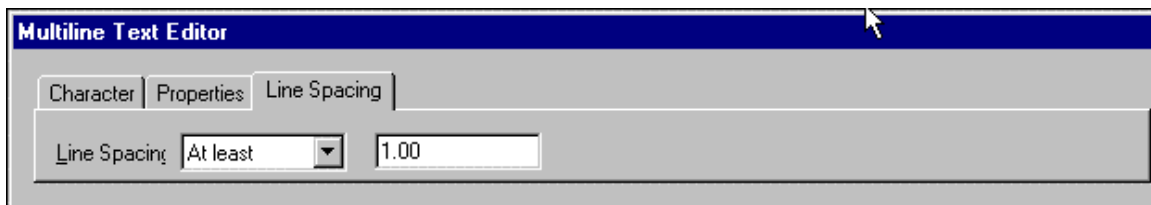
over all text parameters and an Import function enables you to import an external text file into the window.



MTEXT Dialog showing Character options



MTEXT Dialog showing Property options



MTEXT Dialog showing Line Spacing options

## Pre-Selection of Objects

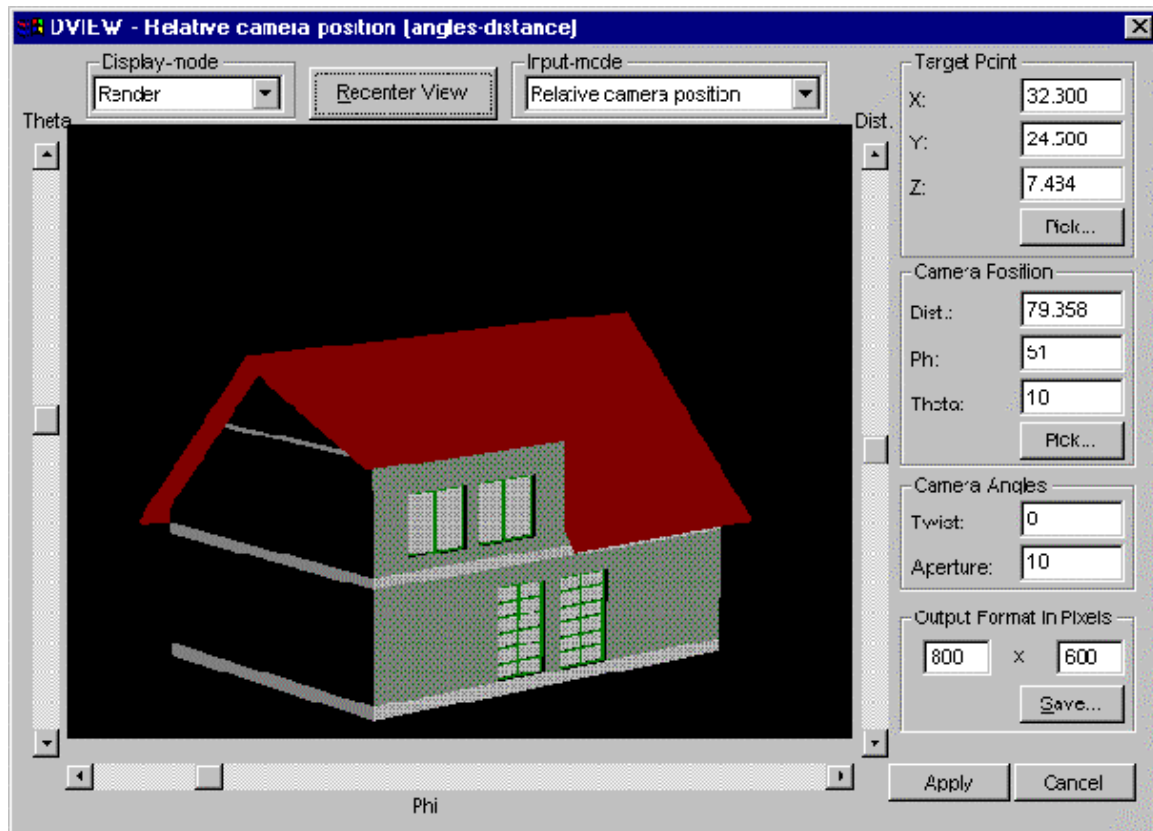
The global PICKFIRST system variable controls whether you select objects before (noun-verb selection) or after you issue a command.

- 0 Turns off PICKFIRST; you select objects after you issue a command
- 1 Turns on PICKFIRST; you select objects before you issue a command

## Command FLXRECOVER

This new feature enables the user to recover damaged flx-drawing files.

## Command DVIEW



The DVIEW command creates a perspective view in different display modes. Supported display modes are rendered, hidden line and wire frame.

Different input modes make it easy to adjust the viewpoint of the drawing. The target point and the camera point can be selected in the drawing. The result can be saved in a bitmap file with a predefined resolution.

The center view button brings the drawing into a base position at the center of the DVIEW output window. If the dialog is closed by the apply button, the current view direction of DVIEW will be set in the drawing.

## Command SAVEAS401 and Command SAVEAS402

Saves drawings in prior FCAD file formats (Versions 4.01 or 4.02).

## Command DRAWORDER

DRAWORDER changes the drawing and plotting order of selected objects in the AllenCad drawing database. In addition to moving objects to the "front" or "back" of the sort order, you can order objects relative to another selected object. DRAWORDER provides the following options:

Above Under Front Back

**Above** - Moves the selected object above a specified reference object.

**Under** - Moves the selected object under a specified reference object.

**Front** - Moves the selected object to the top of the drawing order. The selected object becomes the last drawn entity in the drawing database.

**Back** - Moves the selected object to the bottom of the drawing order. The selected object becomes the first entity in the drawing database.

## Differences between AllenCad and AutoCAD

Different object sort methods are not supported in AllenCad. The object order is changed in the database directly, instead, using DRAWORDER. A SORTENTS system variable for controlling the object sort mode is therefore not required.

## Recording of User Input

Primarily for support purposes, file logging ( a new user-input-recording function) has been incorporated into AllenCad. AllenCad writes all text window contents into a specified file. The command can be activated with the command LOGFILEON. The LOGFILEOFF command stops the prompt echoing to a file. An output file name and path can be set via the global system variable LOGFILENAME. The global system variable LOGFILEMODE controls whether or not a logfile can be written. (Please see also the system variable chapter in the developer section)

## Command and Function Enhancements

### Object Snap

Snap points of hatch pattern lines will be ignored during the point selection.

### Purge Command

The performance of the PURGE command has been enhanced. An internal reference counter makes the purging of unreferenced AllenCad table entries faster.

## Transparent Alias Commands

Alias commands can be used while a command is active (transparently). See the example of the transparent alias command 'Z below:

Definition in the ALLENCAD5.key file:

Z=ZOOM

> LINE

From point:

To point:

To point: 'z

Zoom scale factor: Window

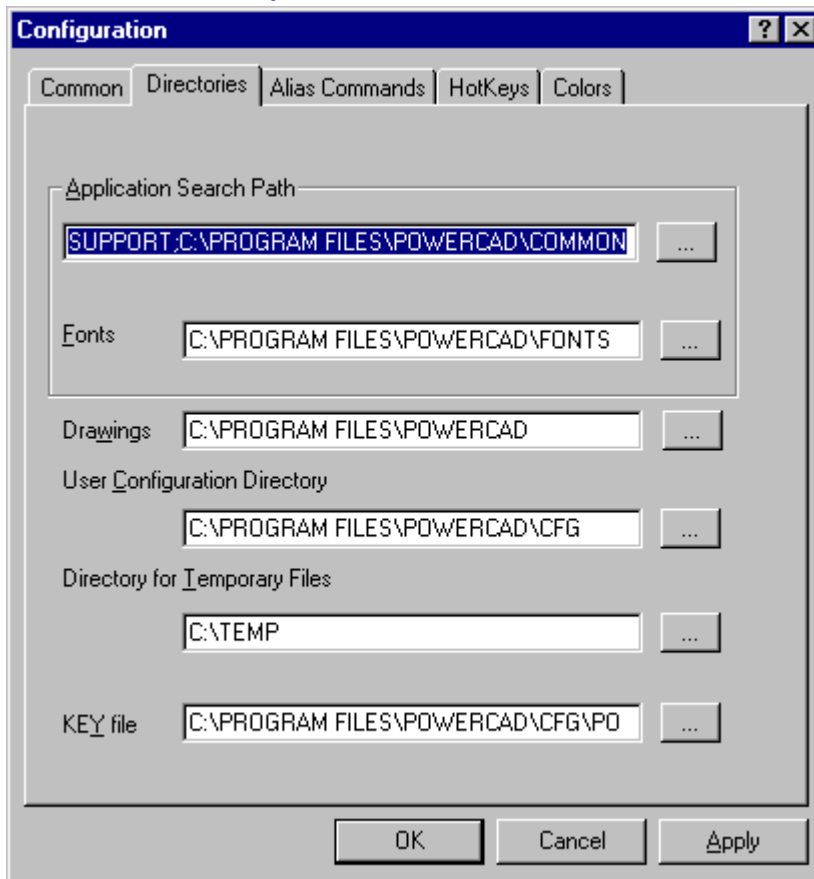
First corner:

Second corner:

To point:

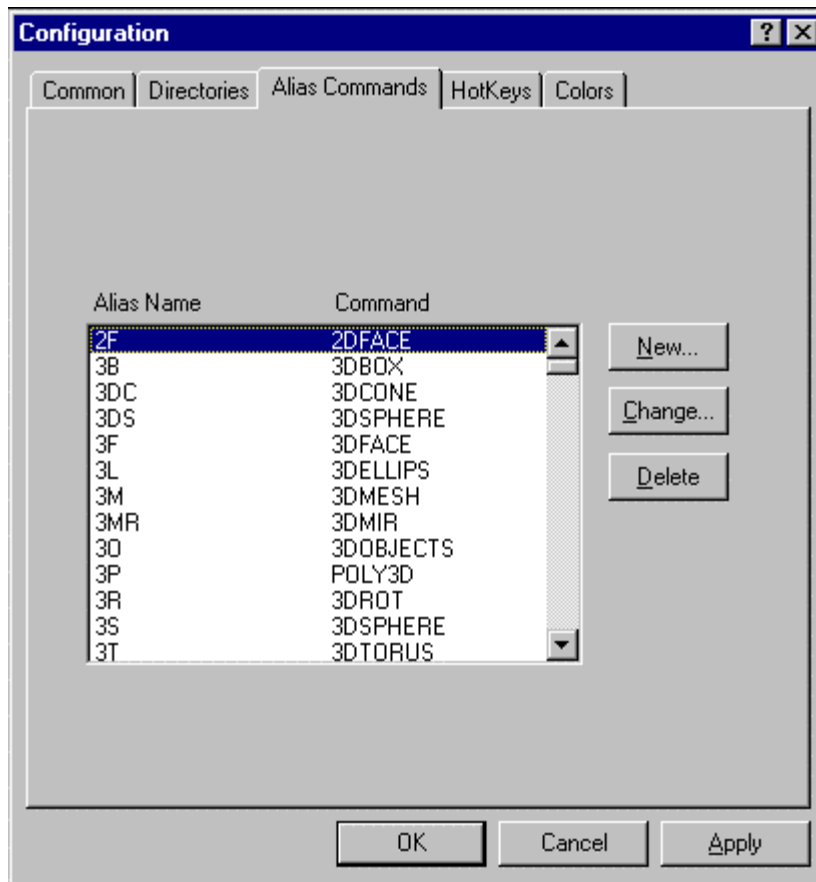
## CONFIG Command

## Default Directory Browse Buttons



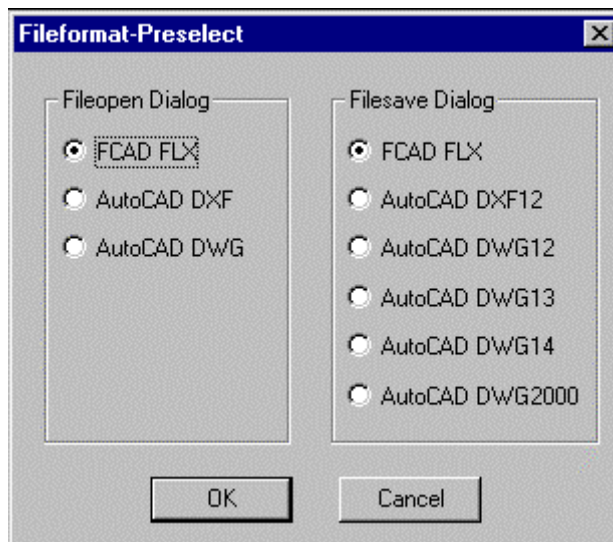
Under the **CONFIG>Directories** tab, browse-buttons extend the ability to adjust AllenCad path settings. The font path supports multiple path selection, which can be useful if you have multiple text font directories. If two font directories contain the same font definition, the first-found font will be used.

## New Alias (KEY) File



A new **KEY** file containing command equivalents for virtually all common FCAD and AutoCAD commands is now utilized to simplify program operation for FCAD or AutoCAD familiar operators. The **KEY** file can be edited under the **CONFIG>Alias Commands** tab, or by using a regular text editor (**KEY** files are located in the \Cfg folder upon installation).

## Set Default File Format





The default file format for opening and saving files can be changed from the default FLX format to DWG or DXF format under the **CONFIG>Common** tab, *by selecting the option Fileformat*. File Open default can be set to FLX, DWG, or DXF. File Save default can be set to FLX, R12 DXF, or R12 – R2000 DWG format.

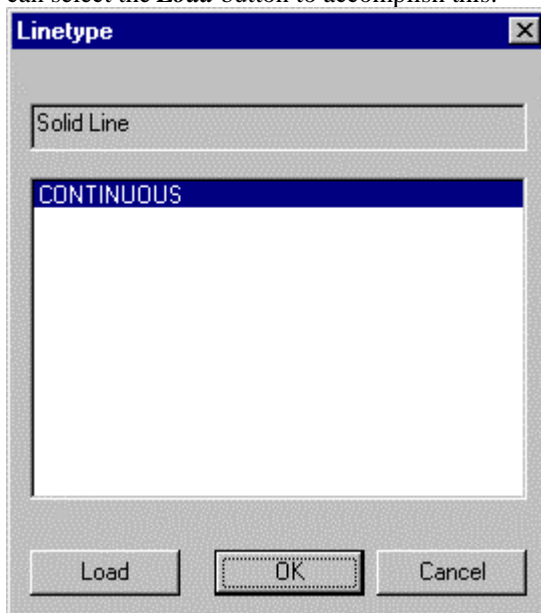
## Autosave

Autosave in AllenCad essentially operates in the same manner as in AutoCAD, with parameters set in the CONFIG>Common command tab.

## Command Extensions:

### Layer Command:

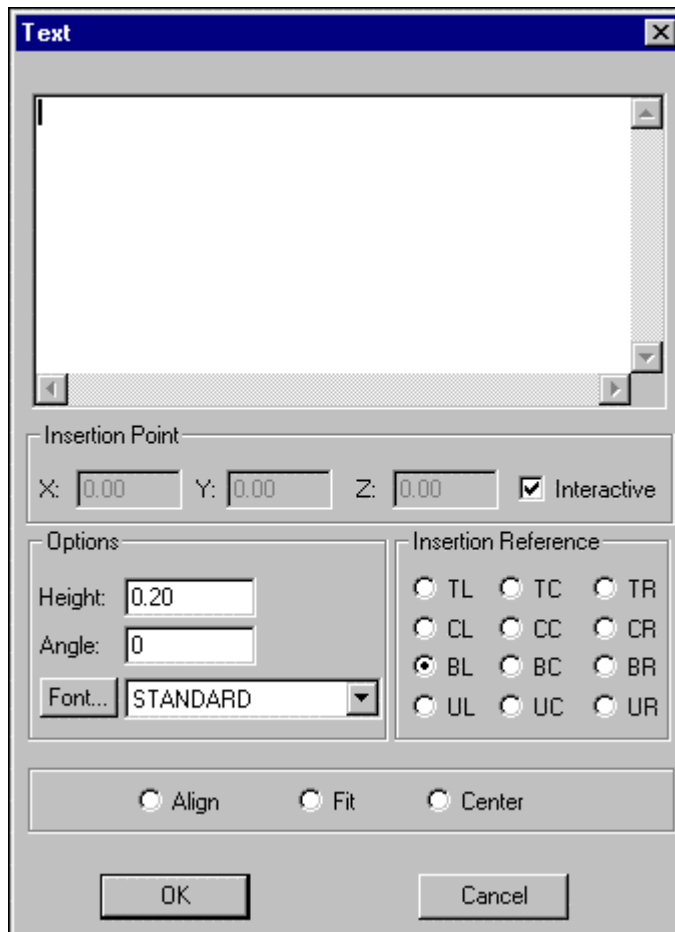
New line type definitions can be loaded without leaving the layer command. In the sub-dialog box you can select the **Load** button to accomplish this:



In the layer manager the default button has been changed to the **New** button. If you type in a layer name and press the return key, a new layer will be created while you remain in the layer manager.

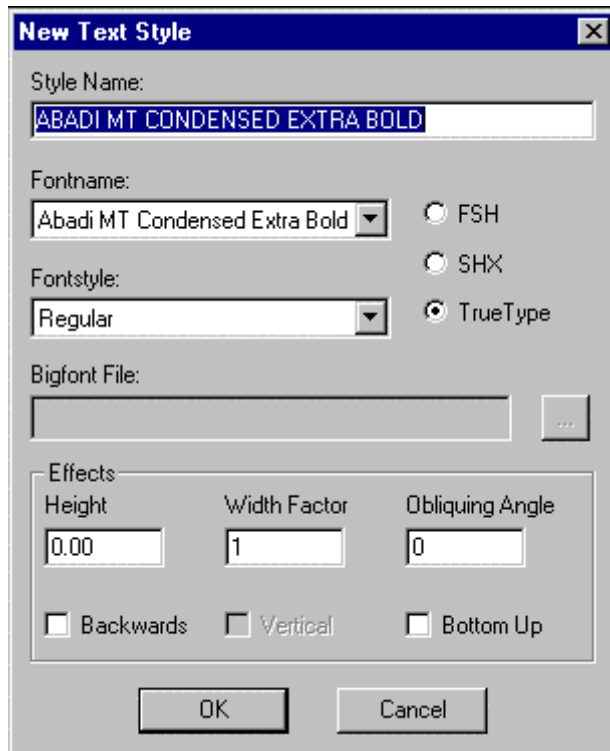
### Text and ATTDEF Command:

New text styles can be directly created while executing the text and ATTDEF command. The FONT command can also be started, by clicking on the font button beneath the current font list box.



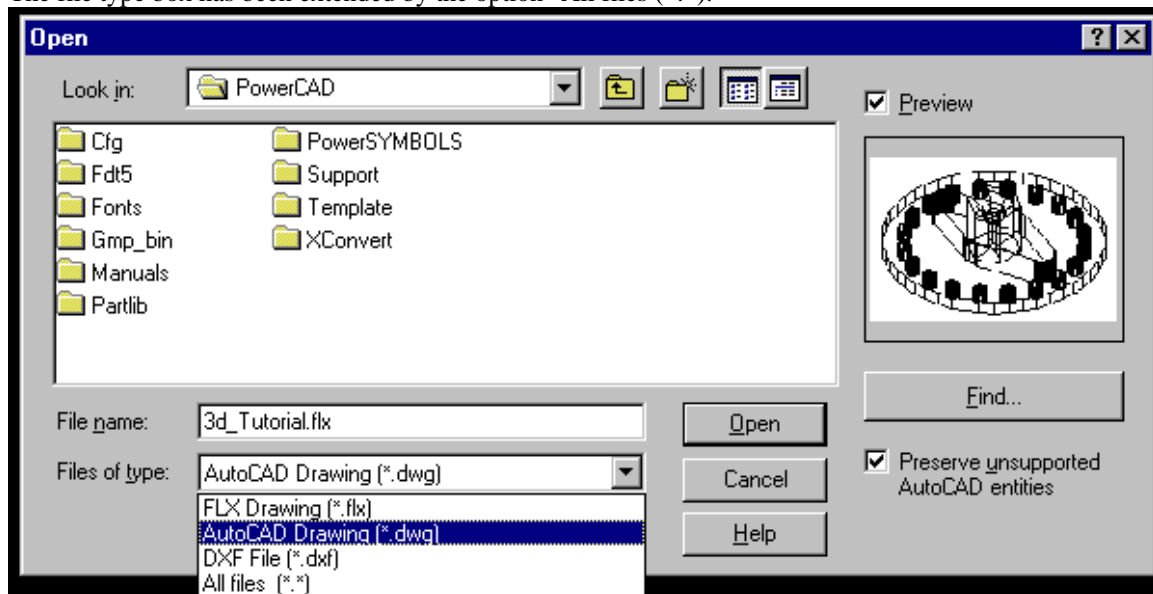
### Command Font:

During font creation, a default text style name is displayed that is dependent upon the selected font name. If you make changes to the default name in the text style edit box, a new default name will no longer be suggested and your changes will remain in effect.



## Command Open

The file type box has been extended by the option "All files (\*.\*)".



## Open Command And The Import of MTEXT-Entities

On opening an AutoCAD format DWG or DXF file, "MTEXT" entities are converted directly to text entities.

## Command Save

The SAVE command creates a preview bitmap for AutoCAD R14 and 2000 DWG files. This bitmap is previewed while OPENing a new file and assists in verifying that you have selected the correct drawing file to open.

## Command Hatch

A new solid hatch pattern is integrated into AllenCad and is applied through the use of the Hatch or BHATCH commands. This allows you to fill in areas with solid colors, as well as patterned hatches.

## Commands 3DVIEW and SETVIEWDIR (VIEW TWIST)

The commands 3DVIEW and SETVIEWDIR support the rotation of the view in the XY-Plane. This is especially useful for quickly re-orienting the drawing within the drawing surround prior to plotting.

## Roundtrip DWG/DXF 2000

If you open an AutoCAD drawing, you have the possibility to "Roundtrip" all unsupported AutoCAD 2000 entities. That means, that while being edited in AllenCad, unsupported AutoCAD 2000 entities remain unchanged and invisible the background. If the drawing is saved in AutoCAD 2000 drawing format again, all entities including these unsupported AutoCAD 2000 entities will be written back to the file.

### "Roundtripped" AutoCAD Entities:

ARCALIGNEDTEXT  
BODY  
IMAGE  
LEADER  
MLINE  
OLE2FRAME  
PROXY  
RAY  
REGION  
RTEXT  
SOLID3D  
TOLERANCE  
WIPEOUT  
XLINE  
XRECORD-OBJECTS

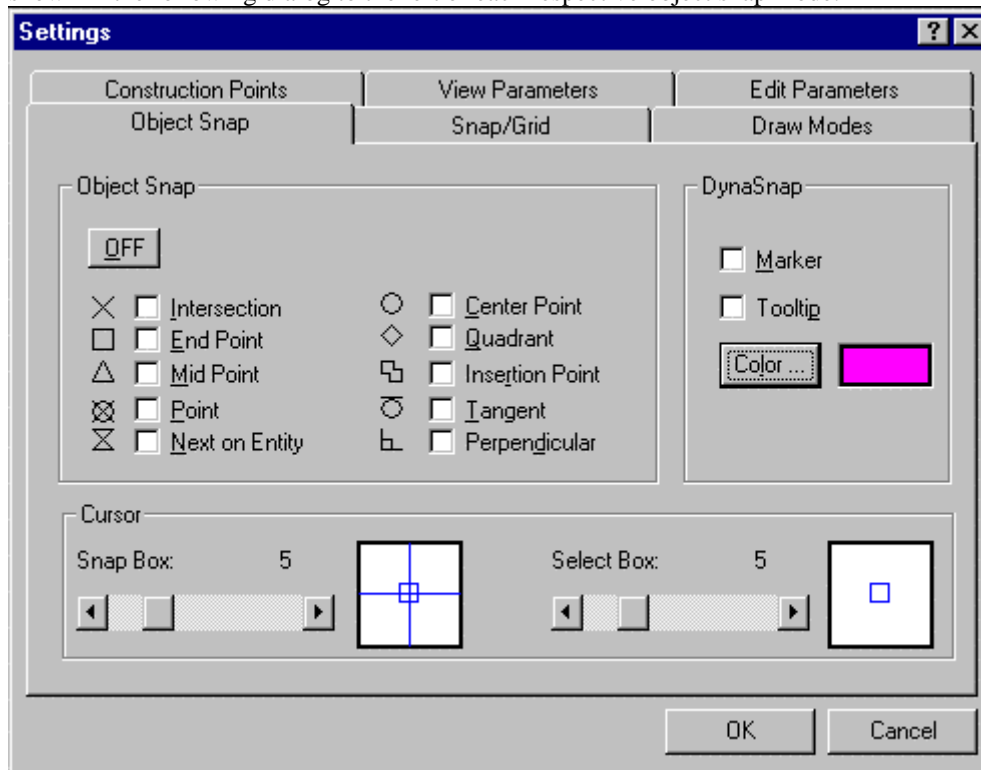
### "Roundtripped" AutoCAD Table Information:

DICTIONARYWDFLT OBJTYPE  
DICTVAR  
IMAGEDEF  
MLINESTYLE  
OBJECTPOINTER  
PLACEHOLDEROBJECT  
VBAPROJECTOBJECT  
WIPEOUTVAROBJECT

## DynaSnap

DynaSnap stands for dynamic snap functionality. AllenCad has the possibility to identify the currently

active snap point dynamically, by using distinctive symbols (markers). You can choose the desired snap point or switch between different modes by the tab-key. Dynasnap mode can be turned on in the register **"Object Snap"** of the SETTINGS dialog or by switching the marker and/or the tool-tip button on. Different color of the marker can be also selected. The displayed markers or symbols are the same as shown in the following dialog to the left of each respective object snap mode.



## Object Snap Modes:

Turns on running object snap:

Marker	Mode	Supported Entities
X	Intersection	Snap to the intersection of an arc, line or, polyline segment.
□	End Point	Snap to the closest endpoint of an arc, line or polyline segment, solid, or 3dface.
△	Mid Point	Snap to the midpoint of an arc, line or polyline segment, solid, or 3dface.
⊗	Point	Snap to a point object.
⊗	Next on Entity	Snap to the nearest point on an arc, circle, line, point, or polyline segment.
○	Center Point	Snap to the center of an arc, circle, or polyline arc segment.
◇	Quadrant	Snap to the quadrant point of an arc, circle, or polyline arc segment.
⌵	Insertion Point	Snap to the insertion point of an attribute definitions, part/block, or text.
○	Tangent	Snap to the tangent of an arc, circle, or polyline arc segment.



Perpendicular

Snaps to a point perpendicular to an arc, circle, polyline segment, line, solid, or 3dface.

## New Temporary Object Snap Mode NONE

In AllenCad, all current selected object snap modes can be switched off for a single selection by selecting the **NOne** option

## Menu Echo Toggle ^P

With the option ^P the user can toggle the menu echo on and off in one pull down menu command line.

For example:

```
MENUITEM, (&|ACTDB|<0), "&Line", {Draw line(s)|LINE}, [FCMD]^P^LINE
```

# Developer/Programming Enhancements

## New AllenCad Ini-File Entries and Changes

### [Adjust] Section

The default file format of the SAVEAS command has been extended by the 2000 DWG file format, so a new index number was necessary. The index 5 is used for 2000 DWG default format and DXF-index has moved to index 6.

**DefFileSaveExtIndex=1**

Index	Saveas default file format
1	Flx
2	dwg12
3	dwg13
4	dwg14
5	dwg2000
6	Dxf

### [Path]-Section

The bitmap-path of the desktop buttons is located in the AllenCad ini-file. If no BMPPATH entry is found in the "Path" section of the AllenCad5.ini-file, AllenCad uses the entry of the desktop.cfg file as in version 4.01.

**BMPPATH=c:\ALLENCAD5\SUPPORT\MNP\_BMP**

## Flexible Developer Toolbox (FDT) - Function Changes and Enhancements

## Mousehook

A new flag controls the pressed mouse button.

### **fdt\_get\_curr\_mousepos()**

Retrieves the current mouse position in coordinates.

## New and Changed Local System Variables in AllenCad

### List of New Local Variables:

Name	Value	read only	saved
OPENFORMATVERSION	0	X	

## New and Changed Global System Variables in AllenCad

### List of New Global Variables:

Name	Value	read only	saved
PARAMETRIC	1	X	
PARAMMODELACTIVE	0	X	
PLINETYPE	2		X
LOGFILEMODE	0		X
LOGFILENAME	""		X
PICKFIRST	1		X
FONTALT	"Simplex.shx"		X
AROUNDTRIP	0		X
DYNASNAP	3		X
DYNASNAPCOLOR	6		X

### Changed Global Variables:

Name	Value	read only	saved
FCVERSION	"5.00"	X	

## New Local System Variables

### **OPENFORMATVERSION**

The variable can be used to get the currently opened file format version. All possible values are listed in the following table.

## Values

Value	Opened File Format
0	FLX – AllenCad/FelixCAD
2	DXF - AutoCAD R 2.5

102	DWG - AutoCAD R 2.5
8	DXF - AutoCAD R 2.6
108	DWG - AutoCAD R 2.6
9	DXF - AutoCAD R 9.0
109	DWG - AutoCAD R 9.0
10	DXF - AutoCAD R 10
100	DWG - AutoCAD R 10
11	DXF - AutoCAD R 11
111	DWG - AutoCAD R 11
12	DXF - AutoCAD R 12
112	DWG - AutoCAD R 12
13	DXF - AutoCAD R 13
113	DWG - AutoCAD R 13
14	DXF - AutoCAD R 14
114	DWG - AutoCAD R 14
15	DXF - AutoCAD R 2000
115	DWG - AutoCAD R 2000

## Properties

<i>Type</i>	Integer
<i>Range</i>	0 - 115
<i>Default</i>	0
<i>Status</i>	Read-Only
<i>Commands</i>	The commands OPEN, QOPEN, DXFIN and NEW change the status of this local system variable

## New Global System Variables

### LOGFILEMODE

Specifies whether or not the contents of the text window are written to a log file.

## Values

0	No log file active. (off)
1	File logging (LOGFILEON) is active. (on)

## Properties

<i>Type</i>	Integer
<i>Range</i>	0   1
<i>Default</i>	0
<i>Status</i>	Read-Only
<i>Commands</i>	The commands LOGFILEON and LOGFILEOFF change the status of this global system variable
<i>See also</i>	Global system variable LOGFILENAME

### LOGFILENAME




Specifies the name of the current log file, the extension and the path.

## Properties

<i>Type</i>	String
<i>Default</i>	""
<i>Status</i>	Read/Write (saved in the AllenCad CFG File).
<i>Example</i>	"c:\temp\logfile.log"
<i>Commands</i>	LOGFILEON, LOGFILEOFF
<i>See also</i>	Global system variable LOGFILEMODE

## Chapter 7 EXTRA Topics

You find this option in the Configuration Dialog called by the command CONFIG.

The CONFIG command can be activated by choosing the symbol  in the toolbar.

### Tablet Menu

The program supports the use of a digitizer tablet as input medium and cursor control.

Coordinates for drawing points can be entered as well as commands and functions by the proper symbols.

To control the tablet it is necessary to load a Tablet Menu file.

```
> LINE
From point: P1
To point: P2
To point: P3
To point: <ENTER>
<ENTER>
> LINE
From point: P4
To point: P5
To point: P6
To point: Close
```

### Renaming a Layer

Choose the layer to be renamed from the layer list of the dialog box.

The name appears now in the entry window. Write the new name over the old one.

Finish renaming by clicking on RENAME.

The renamed layer will be listed under its new name in the dialog box.

### Define Linetype

A newly created layer will automatically have the linetype CONTINUOUS assigned to it. A new linetype can be defined with the button LINETYPE.

Click on Linetype;

The available linetypes are shown in the dialog box that appears. Select the linetype to be used; and Verify your choice by clicking on OK.

This will make the selected linetype the standard linetype of this layer. All drawing elements will be drawn with this linetype, that now becomes the linetype BYLAYER

When the linetype is changed in this manner all elements of this layer that were drawn with the former standard linetype will now be drawn with the newly selected linetype.

All elements of the layer that were drawn with a linetype different than the BYLAYER linetype will not be affected by the change.

### Sample Macro

These lines define a macro that draws a circle at the coordinates 0,0 with a radius of 10 drawing units:

```
CIRCLE
```

```
0,0
```

```
10
```

An equivalent is (note spaces!):

```
CIRCLE 0,0 10
```

Save the file in plain text format and use .mcr as file extension.

## Parts and Attributes



### Parts

A part is a complex object that consists of several drawing entities or objects. After an entity or group of entities is selected and defined as a Part, it is saved under a Part name and treated as one single entity for future operations.

Defining Internal Parts

External Parts

Inserting Parts

Exploding Parts

Part/Symbol Library



### Attributes

Attributes are, in general, text objects that are related to or joined to Parts. They are treated together with the Part as a whole object. The text objects may be constant, or variable. They may be visible, or hidden. There may be a single Attribute with a Part, or several.

Using Attributes, the parts can be further defined by textual information.

Defining Attributes

Inserting Parts with Attributes

Editing Attributes

Attribute Modification

## ORTHOGONAL MODE



Orthogonal Mode aids in designing by allowing:

Lines, chains and polylines only to be designed along lines parallel to the axes of the coordinate system);

Changes of direction when drawing lines, chains or polylines are always perpendicular to the current axes;

Construction points for geometrical objects such as radius, diameter, arcs, ellipses, ending points for polylines, chains and so forth will always be parallel to the axes of the coordinate system.

When the user is modifying objects, ORTHOGONAL MODE means:

- Points for scrolling, scaling, rotating and so forth will be exactly parallel to or perpendicular to the axes of the defined base point.
- Objects or copies will be scrolled, stretched and so forth either parallel or perpendicular to the starting point.

The ORTHOGONAL MODE is effective to use in drawing and modifying rectangular aligned geometry.

Example Snap Grid:

XY-Origin: 1.0,1.25

Width 1

Height:1

Entering these parameters, assuming the horizontal and the vertical distance between the points of the snapping grid is one drawing unit will enable the user to pick points:

1. Whose X-coordinates have the value 1 or a multiple of it; and
2. Whose Y-coordinates exceed the whole-numbered coordinate values by 0.25 drawing units.

Example: Activating Object Snap function 'END'

> LINE

From point: P1

To point: end

of object: P2

To point:

...

#### Standard Views

- Top (Plan View)
- Front
- Left
- Right
- Back
- Bottom

#### Standard Layer - Layer "0"

The standard layer is always called Layer 0. It is generated automatically by the program when a new drawing is created. The layer color 7 (black) as well as the linetype continuous is automatically established for this layer. This layer cannot be erased or renamed.

#### Lock Files (.flk)

Lock files are backup files that the program creates as soon as the drawing file is opened. They have the same file name as the original file, but are allocated with the .flk default file extension. Lock files are automatically removed as soon as the original drawing file is correctly closed.

#### Application Search Path

This directory contains the support files required for pull down menus, shading and patterns, line definitions and layouts as well as the toolbar configuration files.

#### CFG Directory

The configuration directory contains the current drawing editor configuration details.

#### Driver Directory

Directory for device driver interface files (currently only tablet driver interface DLL; normally same as kernel directory)

#### Temporary Directory

Directory for temporary user-specific files (UNDO-List and Display-List files)

#### Drawing Directory

Default directory for drawings

#### Key file

The key file contains the Hot Key Definitions and Alias Commands defined by the user.

#### Palettes

Palettes are toolboxes that contain a number of buttons linked to various commands and instructions. These windows can be placed anywhere on the screen and act as an alternative method of calling up commands, in addition to those already offered by Pull-down menus or direct input.

#### Selection Set...

A collection of selected entities is called a selection set.

#### Text Window...

The Command Line, or the Text Window, serves as communication between the program and the user. In its default condition, the program uses a Command Line situated directly underneath the Menu Bar.

The button T on the right of the Text Window will allow the user to switch between this mode of display and a bigger Text Window, placed freely on the drawing field.

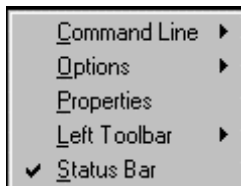
#### Select Box...

Indicates the pixel size of the cursor window when selecting objects.

#### Snap Box...

This value indicates the pixel size of the cursor window when using the object snap option.

#### Desktop Manager



The Desktop Manager controls the appearance of the program desktop. With it, you can toggle the display on/off of the Command Line, Left Toolbar, Status bar, Toolbar, Menu, Command Option display, etc. In this way, the operator can choose to display only those elements that suit his or her personal drawing style, leaving a maximum drawing edit area.

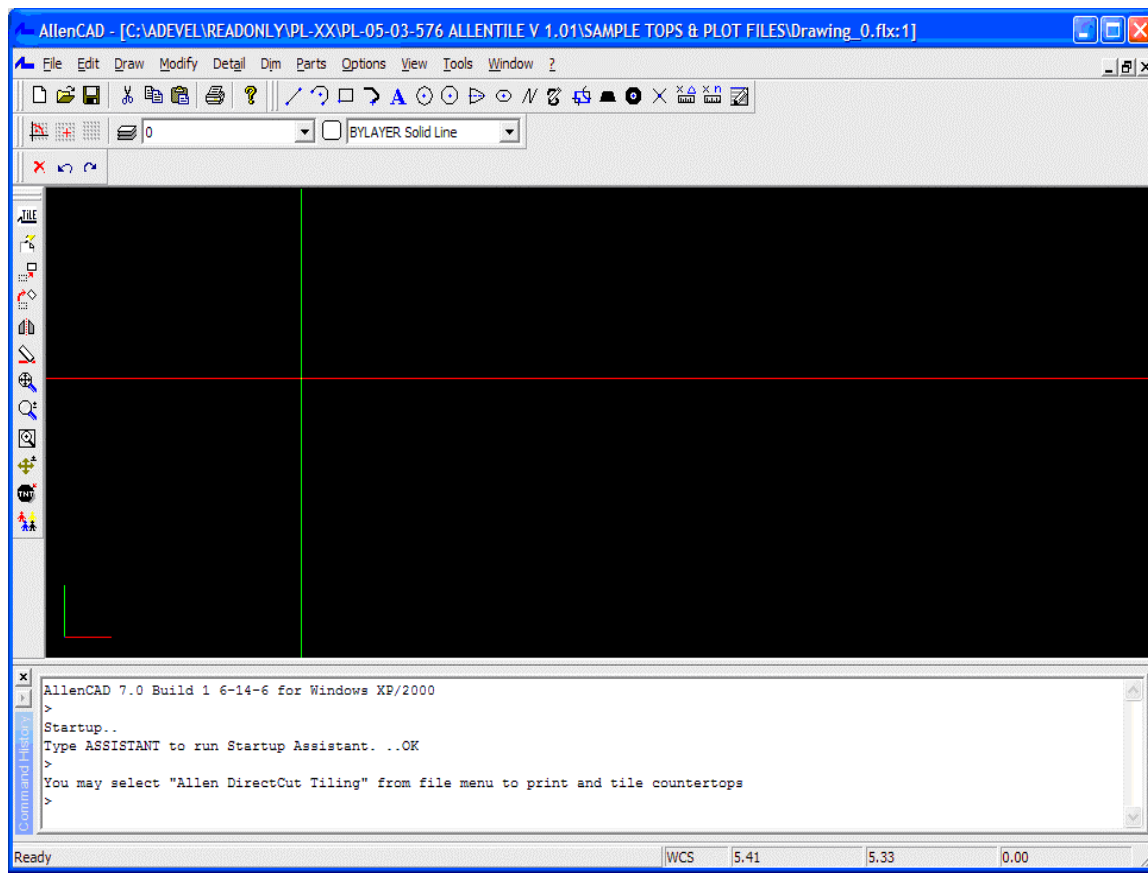
#### AutoCAD Compatibility

AllenCad is compatible with AutoCAD in these crucial areas:

1. Drawing file format (DWG and DXF®).
2. Commands and input modes.
3. Customization.
4. LISP programming.
5. C/C++ programming interface.
6. Application-defined dialog boxes.

This section reviews the similarities and differences between AllenCad and AutoCAD. It is addressed to CAD users and programmers who are already familiar with AutoCAD and want to start using AllenCad productively in their work - right away!

AllenCad ships with its own user interface, however, it allows you to change the interface to look more like AutoCAD Release 13 to 2000. Changing the user interface involves simply executing a couple of commands and loading a few files. When you follow the steps listed below, the AllenCad desktop will appear as below:



## AutoCAD Migration Tools

For those areas where AllenCad is not 100% compatible with AutoCAD, our AutoCAD migration tools help ease the conversion process. These include:

### Dialog Box Converter:

Dcl2Dlg.Exe converts AutoCAD *.dcl* (dialog control language) files to AllenCad *.dlg* files. The tool is located in the `\xconvert\dcl2dlg` directory.

### Desktop and Menu Editor:

DME is a visual workbench for creating and modifying dialog boxes, palettes and pull-down menus. You immediately see the result; no DCL programming skills are necessary to create dialog boxes. DME can also be used for application prototyping. You access DME within AllenCad by selecting Tools > Resource Manager > Run Desktop and Menu Editor from the menu bar. The DME can also be run externally by clicking on the DME icon in the AllenCad folder in the Windows' the Start - Programs menu.

### Include File:

AllenCad provides the *ads2fdt.h* include file, which redefines *ads\_* function names to their *fdt\_* equivalents. You must recompile the ADS® source code using the *ads2fdt.h* header file and link the FDT libraries. For more information, read the documentation in file *fdt40.pdf*. Also, have a look at the *ads2fdt.h* header file and libraries that are provided with AllenCad.

**Tools Folder:**

The \tools subdirectory on the CD-ROM contains sample files that may be useful to OEM developers. Tools\dltutor contains examples for creating AllenCad dialog boxes. Tools\dwgcnvrt contains an FLISP (FLEXIBLE LISP FORMAT) program example to convert drawing files. Tools\protlisp contains the utility to protect FLISP files. Tools\shapes contains font source files (SHP).

**Program Features**

**Drawing Navigator TM** an explorer style window, which displays all the layers, line types, blocks, text styles, views, and even coordinate systems contained in a drawing.

**Visual Block Manager** lets you quickly create graphic block libraries (supports FLEXIBLE DRAWING EXCHANGE (FLX), DWG, and DXF blocks). With the click of a mouse or digitizer button, blocks can be graphically inserted at any scale or rotation you need.

**AllenCad Pro's Image Library Tool** lets you store multiple slides and bitmaps in a single library. Supports AutoCAD® slide (.SLD) and Windows bitmap (.BMP) formats.

**AllenCad Pro's Hybrid Raster/Vector Image Overlay capabilities** allows you to integrate raster bitmaps directly into vector drawings.

**AllenCad Pro's Batch Processor** lets you perform unattended batch plotting or file conversions.

**Additional Program Highlights:**

Short learning curve

Context sensitive online help with print-ready documentation

Online Learning Center with instructional movies

Free access to Web help

Metric or Imperial Units

Fast pan and zoom based on an new 3D display list

Options Menus and dialog box support in many commands.

"Repeat previous command" function built-in.

Cut and Paste between drawings and the clipboard

Simultaneous opening of up to four drawings with up to four viewports of each

A full range of object drawing and editing commands

Context sensitive object editing (just pick an element and edit options are presented)

Zoom back up to 20 steps

Unlimited undo function

Icon menu support

Grouped functions on palettes, that can be called onto the drawing surface for quick access to commands

User-definable Alias commands

Drag and Drop Modifying\_the\_Desktop\_On-The-Fly user interface

Easy to organize *part* (block) libraries.

Preview of drawings, blocks and symbols  
Easy definition of complex objects (parts, blocks, symbols)  
Visual Part and Symbol Library Manager (DWG® and DXF® files supported)  
Support for Extended Entity Data (EED)  
Externally referenced drawings (XRef's)  
Sophisticated object grouping  
Associated dimensioning and hatching  
User-definable dimensioning styles  
Layout mode (Paper space / Model space)  
Enhanced high speed printing/plotting with user selectable pen assignments and full plot previews  
Batch printing/plotting and file conversion  
Detailed plot menu with multiple configuration options  
User-definable coordinate systems (UCS)  
Autorendering based on Microsoft OpenGL® graphics  
Fast hidden line removal  
Interface for hybrid raster/vector applications  
Digitizer support/Tablet menu support  
Thickness command for common elements (3D entity extrusion)  
Raster file import, manipulation and editing  
Direct Windows Metafile (WMF) and Bitmap (BMP) Export

**AutoCAD Compatibility Highlights:**

Exceptional AutoCAD® compatibility  
Direct reading and writing of AutoCAD DWG® and DXF® files up to Release 2000  
Transparent execution of all compatible AutoCAD® functions  
Industry standard layers, line types, blocks, fonts, and line types  
AutoCAD DCL® Dialog Converter  
Options to load AutoCAD® emulated menus (R14 and R2000), toolbars, command aliases and "hot key" assignments  
Direct insertion of DWG or DXF drawing parts (blocks)  
Direct reading of AutoCAD 1,0 and 1.1 SHX fonts, linetypes and Hatch Patterns (SHX/SHP, LIN and PAT files) – Including Bigfont support.  
WCS/UCS coordinate systems

**Programming and Customization Highlights:**

True 32-bit CAD  
High-Speed Performance  
Optimized for Windows 95/98/NT/ME and 2000



Network-ready

Customizable toolbars

Open software architecture

Visual Menu, Dialog, and Tool box editor

Lisp Interpreter with code protection option

C/C++ Application Programming Interface (API) similar to AutoCAD

Delphi™ Advanced Programming Interface

Dynamic Link Library support with DDE

Callback and Notification functions

SQL2/ODBC Support

True Type Font support

AutoPort™ utility library automates the conversion of AutoLisp®/ADS® programs, menus, and dialogs written for use with AutoCAD®

OpenDWG Alliance



Our firm, ADSI Inc., is a member of the OpenDWG® Alliance (<http://www.opendwg.org>) and we support its efforts. AllenCad, employs the DWG read-write technology provided by the OpenDWG Alliance.

Prior to the founding of the OpenDWG Alliance, AllenCad used the AutoDirect DWG/DXF read-write technology from MarComp. The AutoDirect technology has since become the basis of the OpenDWG Alliance's OpenDWG Toolkit.

Drawing Objects

AllenCad supports most imported AutoCAD objects. It converts some AutoCAD objects to another format for modification purposes, and retains but does not display a small number of objects that are not supported in AllenCad. The following sections describe how AllenCad deals with all objects found in AutoCAD Release 2000.

Entity Properties

AutoCAD objects have the following properties:

Layer

Color

## Linetype

Space assignment (Model or Paper Space)

Thickness (entity extrusion).

All of these properties are fully supported by AllenCad and they have the same characteristics as in AutoCAD.

For example, AllenCad applies the same color palette to objects, as does AutoCAD. There are 255 RGB colors and two logical colors, BYLAYER and BYBLOCK.

### **Note for Applications Developers:**

When an entity extrusion (thickness) is applied to *TEXT* and *POINT* entities, the thickness is not presently displayed by AllenCad.

## Blocks

**Note:** Another commonly used term for "Blocks" is "Parts".

As with AutoCAD (especially Release 12 and earlier), AllenCad organizes groups of different types of entities as blocks in the BLOCK section of the drawing database. These include:  
User-defined blocks.

Dimensioning blocks, stored as anonymous blocks.

Hatch blocks, stored as anonymous blocks.

Application-defined anonymous blocks.

External references (Xref).

These blocks and their organization in the AllenCad database correspond to the AutoCAD database.

## Symbol Tables

Symbol tables store non-graphical objects in the drawing database. AllenCad manages and organizes symbol tables as AutoCAD does. These symbol tables are maintained in the TABLES section of the drawing database.

### **Symbol Tables Supported by AllenCad**

Symbol Name	Comments
LAYER	Layer names and settings.
LTYPE	Linetype names.
STYLE	Text styles names and settings.
UCS	User coordinate systems.
DIMSTYLE	Dimensioning styles.
VIEW	Named views.
VPORTS	Drawing viewports.

APPID                      Registered application identifiers for extended entity data.

## System Variables

Most of AutoCAD's system variables are supported in AllenCad. This means that you find the most recent settings restored when you exchange drawings between the two systems.

See the System Variable Compatibility Chart for a complete listing.

## Entering Commands

All AutoCAD 2000, commands that have direct equivalent commands in AllenCad can be entered at the command line, much as you would do in AutoCAD.

When you want to draw a line, for instance, you can simply type in the command LINE at the prompt:

```
>LINE
From point: [pick]
To point: [pick]
To point: [pick]
To point: c (as in 'Close' the shape, just like AutoCAD)
```

To relocate drawing objects, you apply the Move command in AllenCad, as in AutoCAD:

```
> MOVE
Select objects: 1
1 selected.
Select objects: [Enter]
*** 1 selected. ***
Basepoint: [pick]
Second point of displacement: [pick]
```

These examples, like most of AllenCad's drawing and editing commands, are equivalent in their names in both systems, and are essentially similar in the way they operate. On the whole, commands behave similarly in both systems.

## Command Prefixes and Modifiers

AllenCad is compatible with many of AutoCAD's command prefixes, such as transparent commands and point filters. The following sections detail the similarities and differences between the two CAD programs.

## Command Set Comparison

As described previously, you can use AutoCAD command names and command syntax in most cases, but AllenCad has its own set of additional commands. AllenCad has numerous commands not found in AutoCAD that give AllenCad unique capabilities. There are, however, many commands that have a direct equivalent functionality in AutoCAD.

## Support Files and Customization Tools

From the beginning, ADSI Inc. has made a special effort to provide customization tools for the user, that do not require knowledge of support file syntax or even programming skills. AllenCad provides intuitive methods and visual tools that allow you to adjust the program's desktop and the method of communication with the program — tailored to your individual needs in an easy-to-use way.

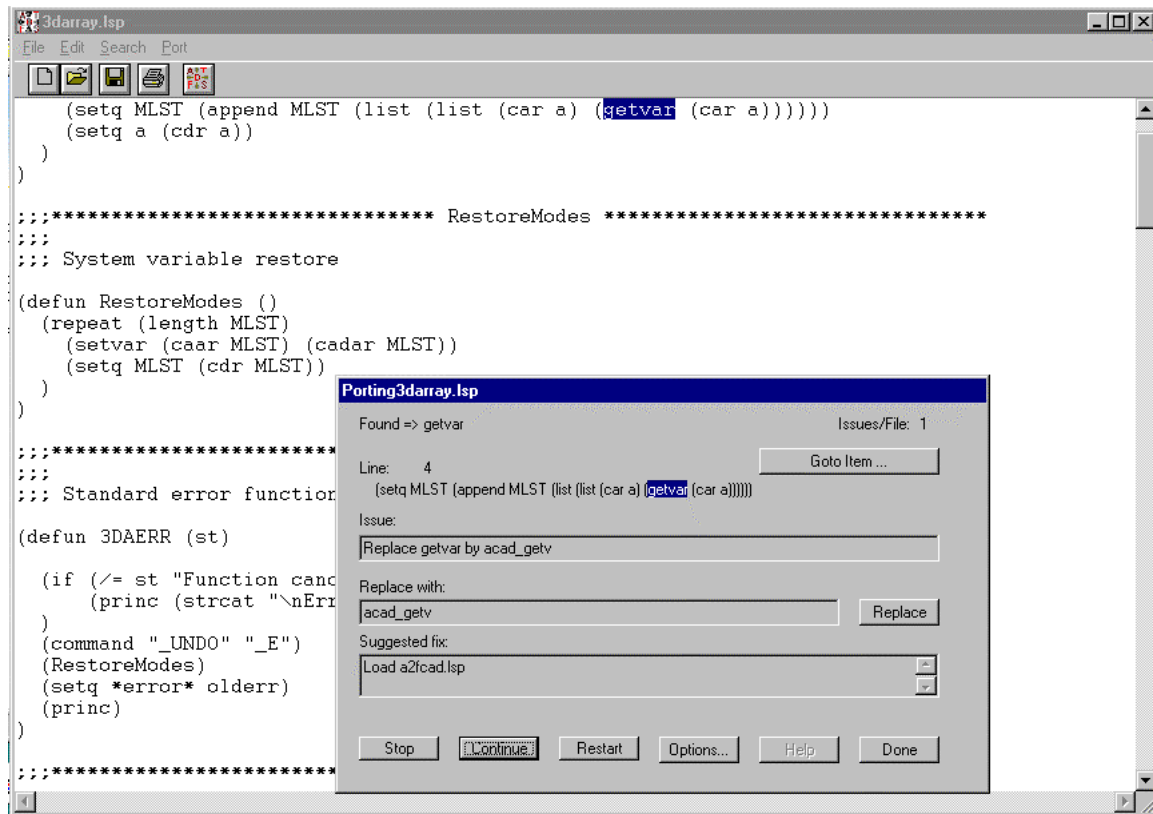
Please refer to the Programming and Customization Manual, as well as the online Help file in the Desktop and Menu Editor program for details on customizing AllenCad to your specific needs.

### The A2FLISP Converter

AllenCad Pro and PowerENGINE incorporate a Flexible LISP (FLISP) interpreter, which supports advanced programming using Lisp. In limited cases, AutoLisp code written for AutoCAD will run directly. In most cases, existing AutoLisp program code will require conversion. As the conversion process with the A2FLISP converter can require user interaction, it is recommended that the user involved in converting LISP code be a LISP programmer.

It handles direct translation on its own and directs you to the location of possible problems, with a description of the difficulty. It edits the source code at the highlight location and provides for simple text replacement, if needed.

The A2FLISP converter is located in the \Xconvert\a2flisp folder of an installed AllenCad. For more information, please read the a2flisp.doc documentation file in the same folder.



Illustrated: The A2FLisp program interactively checking AutoLISP code for incompatibilities.

**Note:** This utility uses normal Windows Text Control (as does Windows Notepad), so the size of files that can be processed is dependent upon the Operating System being used. For Windows '98 there is a 64k limitation, whereas in WinNT or 2000 there is no practical limitation.

## Menu Syntax

The syntax in files that define pull-down menus, icon menus, palettes, and toolboxes is different from the one used by AutoCAD. However, this syntax is easy to understand. If you use AllenCad's Desktop and Menu Editor, you don't even have to know the syntax.

## AutoCAD Compatible Menu, Palettes and Commands

AllenCad delivers two AutoCAD-compatible menu files and a complete alias command file. The menu files are named *R14\_emu.mnu* and *R2000\_emu.mnu* and are found in the AllenCad\support subdirectory. You load the menu with the Menu command. The alias command file is named *PCAD5.key* and it is loaded by default when the program opens.

It can also be helpful to load the AutoCAD-compatible palettes (toolbars) via the palette manager. Use the PalMan command. The ten AutoCAD-compatible palettes are located at the end of the selection list and are identified by the (Pacad) suffix. See figure 3.1.

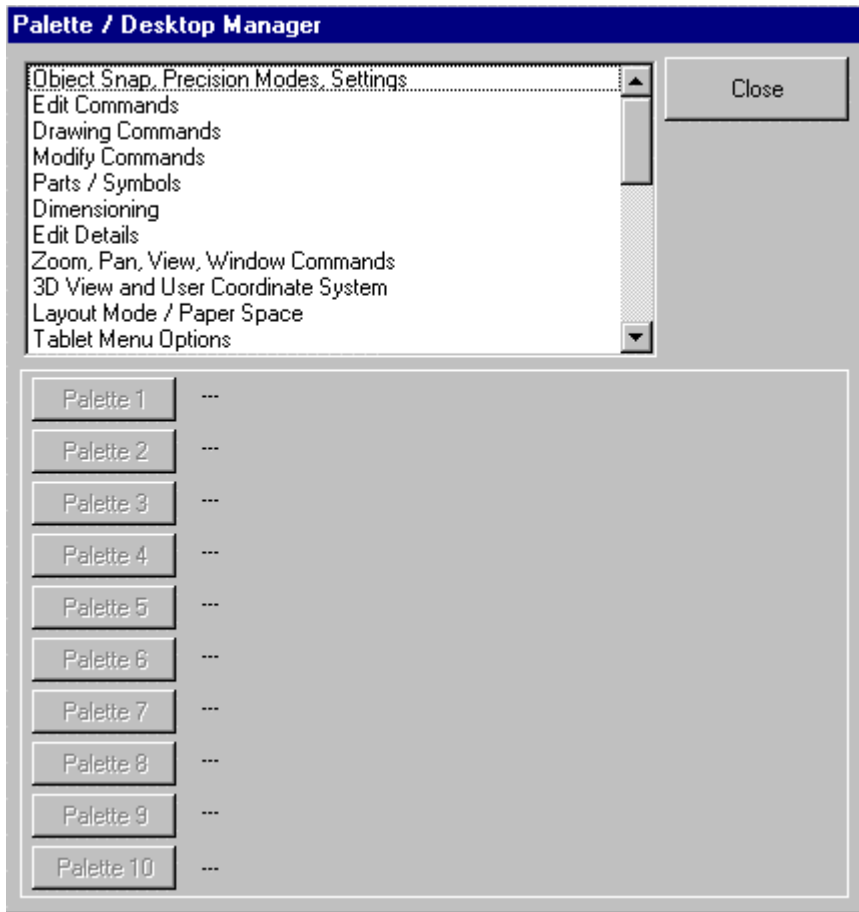


Figure 3.3: The Palette Manager opens and closes toolbars. The (PACad) suffix indicates a palette compatible with AutoCAD's toolbars.

## DCL Converter

AllenCad provides a utility, called Dcl2Dlg.Exe, that converts *.dcl* files to *.dlg* files. The tool is located in the \Xconvert\dcl2dlg directory of your installed AllenCad.

To start the program, double-click Dcl2Dlg.Exe in the Windows Explorer or a shortcut to this program. You specify the DCL file to be converted, and the name of the destination DLG file.

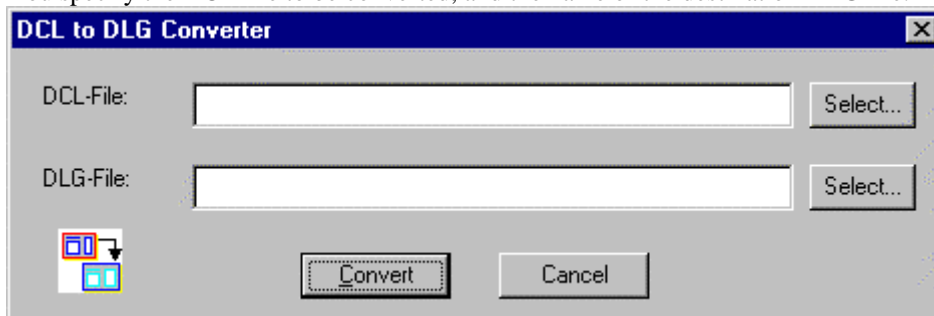


Figure 6.2: The Dcl2Dlg utility converts AutoCAD DCL files to AllenCad format.

Before you start converting your DCL files, make sure that the *base.dcl* (and any other DCL files referenced by your DCL source files) is located in the same directory as the DCL files to be converted.

The *base.dcl* file is not converted because the file is used only by AutoCAD to include global defined dialog entries.

After conversion, the position and size of the dialog controls are not totally optimized in the resulting files. You may have to edit the resulting *.dlg* files with the Desktop and Menu Editor. In some very rare cases, one control might be hidden by another.

In general, we recommend:

In DME, activate the tab setting mode and toggle from one dialog control to another with the Tab key to evaluate whether there are hidden controls in the dialog box.

Make use of the alignment tools in DME to rearrange the controls.

Please check the dialog control properties via the Property dialog.

To run a batch routine to convert multiple DCL files in one process, create a batch file. *cvt\_samp.bat* is a sample file that shows how to this. The syntax is:

DCL2DLG Input\_DCL\_file [Output\_DLG\_file]

Directory paths names can be used in input and output file names.

For more information on the DME click here: DME Help 

## Initiating a Dialog Box

There are some differences in how AutoCAD and AllenCad handle dialog boxes. The most important difference is how they initiate and interact with dialog controls.

A typical AutoLISP code fragment loads the DCL file, sets the controls, takes actions on the controls, and then starts.

In contrast, FLISP loads the dialog box, and then calls the `start_dialog` (or `Dlg_DialogStart`) function with a function\_name as an argument that contains the user dialog interaction function.

The following sample LISP code shows the differences:

### AutoCAD AutoLISP

```
(if (not (new_dialog "setlayer" dcl_id)) (exit))

;; initialize list box
(start_list "list_lay")
(mapcar 'add_list longlist)
(end_list)
;; Display current layer, show initial layer name
in
;; edit box, and highlight list box
(setq old-idx lay-idx)
(if (/= lay-idx nil) (laylist_act (itoa lay-idx)))
(set_tile "cur_layer" (getvar "clayer"))
(action_tile "list_lay" "(laylist_act $value)")
(action_tile "edit_lay" "(lay_edit_act $value)")
(action_tile "accept" "(test-ok)")
```

### AllenCad FLISP

```
(defun InitDlg ()
;; initialize list box
(start_list "list_lay")
(mapcar 'add_list longlist)
(end_list)
;; Display current layer,
;; show initial layer name in
;; edit box, and highlight list box.
(setq old-idx lay-idx)
(if (/= lay-idx nil)
(laylist_act (itoa lay-idx)))
(set_tile "cur_layer" (getvar "clayer"))
(action_tile "list_lay" "(laylist_act $value)")
(action_tile "edit_lay" "(lay_edit_act $value)")
(action_tile "accept" "(test-ok)")
```

```
(action_tile "cancel" "(reset-lay)")
```

```
(if (= (start_dialog) 1)
(progn
...

```

```
(action_tile "cancel" "(reset-lay)")
```

```
(if (not (new_dialog "setlayer" dcl_id))(exit))
(if (= (start_dialog "InitDlg") 1)
(progn
...

```

## Dialog Box Equivalents

The table below lists the FLISP dialog box functions and their equivalents in AutoLISP. Most of the functions are identical. There are, however, some differences.

The difference between AutoCAD and AllenCad in initializing dialog box controls with the start\_dialog function is discussed above in Initiating a Dialog Box.

AllenCad has special functions to control sliders in dialog boxes, whereas AutoCAD uses the set\_tile and get\_tile functions.

Another difference is that there is a separate function (dlg\_ImageLib) to display an image from a library file in an image box of a dialog box; in AutoLISP you use the slide\_image function. Note that you can import entire slide libraries from AutoCAD into AllenCad image libraries. In this way, you can use the same pictures in image controls of your dialog boxes.

The following table shows that AllenCad has some extra functions that might be useful for your application. For example, you can display bitmaps and WMF files or drawing preview bitmaps as images in your dialog boxes.

### Comparison of AutoLISP and FLISP Dialog Box Functions

AutoLISP Function	FLISP Function	Comments
...	Dlg_ImageWmf	Displays WMF file in an image box.
...	Dlg_ImageBmp	Displays BMP file in an image box.
...	Dlg_ImagePrev	Displays a preview bitmap stored in the specified FLX file in an image box.
...	Dlg_ListSetColumnWidth	Sets width of columns in a list box.
...	Dlg_ListTabStops	Sets tab stops in a list box.
...	Dlg_TileSetFont	Sets a font for a control element.
action_tile	action_tile Dlg_TileAction	Assigns an action to a tile.
add_list	add_list Dlg_ListAdd	Adds or modifies a list box item.
client_data_tile	client_data_tile Dlg_TileClientData	Assigns data to a control.
dim_tile	dim_tile Dlg_TileDim	Returns the size (height and width) of a control.
done_dialog	done_dialog Dlg_DialogDone	Closes a dialog.
done_positioned_dialog	done_positioned_dialog Dlg_DialogDonePositioned	Closes a dialog and returns the last screen position of that dialog.

AutoLISP Function	FLISP Function	Comments
End_image	end_image Dlg_ImageEnd	Terminates the operation of an image box.



End_list	end_list Dlg_ListEnd	Terminates operation in a list box.
fill_image	fill_image Dlg_ImageFill	Draws a filled rectangle in an image box.
Get_tile	get_tile Dlg_TileGet	Reads the current value of a control.
get_tile	Dlg_SliderGet	Reads the values and properties of a slider control.
load_dialog	load_dialog Dlg_DialogLoad	Loads a dialog file.
mode_tile	mode_tile Dlg_TileMode	Determines the display mode of a control.
new_dialog	new_dialog Dlg_DialogNew	Provides a new dialog (loaded to memory).
new_positioned_dialog	new_positioned_dialog Dlg_DialogNewPositioned	Provides a new dialog at a specified screen position (loaded to memory).
set_tile	set_tile Dlg_TileSet	Sets a control to a value.
set_tile	Dlg_SliderSet	Sets the values and properties of a slider.
slide_image	slide_image Dlg_ImageSlide	Displays a slide file (.sld) in an image box.
slide_image	Dlg_ImageLib	Displays an image from an image library (.ilb file). Within image libraries graphics of type Bitmap, WMF, or Slide can be stored.
start_dialog	start_dialog Dlg_DialogStart	Initializes and displays a dialog.
start_image	start_image Dlg_ImageStart	Start function to display a bitmaps or vector graphic in an image box.
start_list	start_list Dlg_ListStart	Start function for list operations in a list box.
term_dialog	term_dialog Dlg_DialogTerm	Terminates all open dialogs.
unload_dialog	unload_dialog Dlg_DialogUnload	Unloads a dialog file.
vector_image	vector_image Dlg_ImageVector	Draws a vector in an image box.

## Command Compatibility

Most native AllenCad command names and options are very similar (if not identical) to those available in AutoCAD. In addition, AutoCAD commands that have AllenCad command equivalents have been mapped to allow you to enter AutoCAD commands at the AllenCad command line and execute the equivalent AllenCad command.

See the Command Compatibility Chart for a complete listing.

## Customization Compatibility Issues

One of the reasons for AutoCAD's success has been the ability users have had to customize the CAD software. AllenCad also supports a highly customizable user environment that is similar, but not identical

to AutoCAD's. We invite you to check out the additional customization possibilities implemented in AllenCad.

Please refer to the **AutoCAD Conversion** section below for step-by-step instructions on configuring AllenCad to emulate the AutoCAD user interface.

## Database Compatibility Issues

AllenCad provides full support for reading and writing AutoCad drawing files:  
Read DWG files from Release 2.5 up to Release 2000, as well as all versions of AutoCAD LT.

Write DWG files in formats that are read by AutoCAD Release 12 up to R2000, as well as by all versions of AutoCAD LT.

In addition, AllenCad imports and exports files in DXF format.

The proprietary drawing file format of AllenCad is *Flexible Drawing Exchange (FLX)*. From the beginning, the *.flx* file format was designed to be very similar to the *.dwg* format. Beginning with AllenCad, "round-trip" compatibility ensures that all AutoCAD 2000 entities (even those not supported in AllenCad) are retained in the drawing database. There is therefore no loss of data when exchanging drawings back and forth between the two CAD systems.

In terms of drawing database compatibility, AllenCad provides these features:

Blocks and WBlocks, including those with attributes, are handled just as in AutoCAD. You can insert *.flx*, *.dwg*, and *.dxf* files into drawings.

You can attach externally referenced drawings (xrefs). Again, you have the choice of inserting *.flx*, *.dwg*, and *.dxf* files into drawings.

Supports user coordinate systems (UCS).

Supports model space and paper space.

Supports grouping of drawing objects.

You can use the same font files as in AutoCAD (*.shx*), including BigFonts.

You can use the same linetype definition files as in AutoCAD (*.lin*).

You can use the same hatch pattern definition files as in AutoCAD (*.pat*).

AllenCad provides a high degree of DWG and DXF compatibility. This means you have a secure investment when you want to set up a new computer-aided design workstation, or want to add CAD workstations in a mixed configuration.

## Menu Bar

## Cursor

The cursor corresponds to your mouse or digitizer pointer position, while in the drawing editor widow. In

this window, it is used to select onscreen objects, command options in options windows and dialogs and to define two-dimensional coordinate locations. Beyond the edit window, the cursor symbol changes to the standard mouse pointer.

## UCS Icon

The UCS icon is your reference to your view of the drawing. As you change your viewpoint 3-dimensionally, the icon changes to reflect the current viewpoint.

## Command Line Prompt Area

## Status Line

## Emulating the AutoCAD Desktop

From its inception, AllenCad has been designed to offer unparalleled compatibility with AutoCAD® by Autodesk, Inc., the world's leading CAD system with over 4 million registered users.

To aid AutoCAD-familiar operators, a number of optional desktop and interface components are included allowing the system to be quickly reconfigured to work more like later releases of AutoCAD (R14 and R2000).

### **AutoCAD Compatible User Interface**

To change the overall user interface to look more like AutoCAD's, select Tools > Desktop Manager (command DESKCONF) from the menu bar. Please follow this link for more detailed information on this command's capabilities. In the DESKCONF dialog, you need to change the settings of the following options:

1. Command Line: Bottom.
2. Options: Off. However, unless you are familiar with the commands and command options, we recommend that you set Options to Toolbar or Menu.
3. Properties: On.
4. Left Toolbar: Off.
5. Status Bar: On.

To repeat a command, press the spacebar, right-mouse click or the Enter key, just as in AutoCAD.

## AutoCAD Compatible User Interface

To change the overall user interface to look more like AutoCAD's, select File > Desktop Manager from the menu bar. You need to change the settings of the following options:

1. Command Line: Bottom.
2. Options: Off. However, we recommend that you set Options to Toolbar or Menu.
3. Properties: On.
4. Left Toolbar: Off.

## 5. Status Bar: On.

### AutoCAD Compatible Commands and Aliases

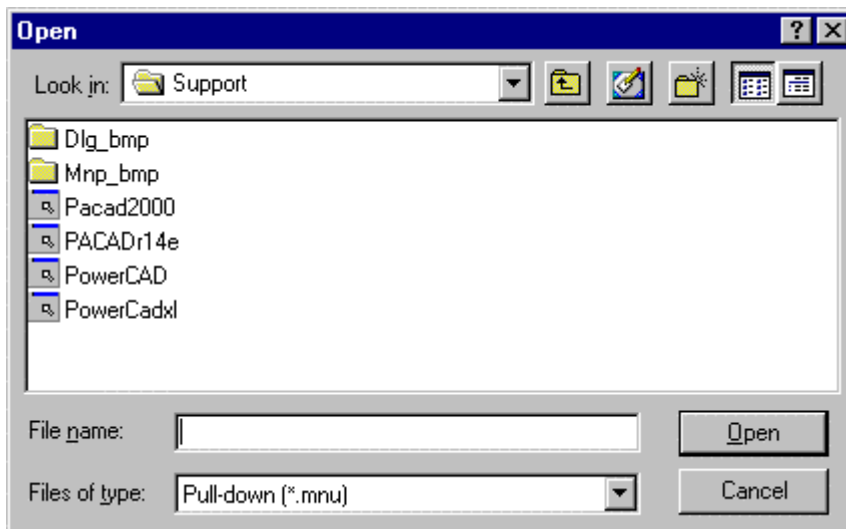
In the AllenCad \cfg directory you will find an AutoCAD-compatible alias command file, *AllenCad.key* (Note: AutoCAD's equivalent of this alias file is called ACAD.PGP). This file is automatically loaded when the program starts up and can be simply edited through the CONFIG command. This file contains both the "two finger" alias command mappings, as well as the mapping of ACAD command names to their AllenCad equivalent commands. This means that you can enter AutoCAD commands at the command line and AllenCad will execute its equivalent command.

See the Command Compatibility Chart for a complete listing of command equivalents.

To repeat a command, you can press the spacebar, right-mouse click or the Enter key, just as in AutoCAD.

### AutoCAD Compatible Menus

To make the pull down menus look like AutoCAD 14's, load the *R14\_emu.mnu* menu file (found in the AllenCad \support folder) with the Menu command. A similar menu file, *R2000\_emu.mnu* is available for those users more familiar with version 2000.



Simply select the menu file you want to load, from the MENU command dialog.

### AutoCAD Compatible Palettes

To make the palettes emulate AutoCAD's, use the PalMan command to display the Palette/Desktop

Manager dialog box.

This command can also be issued by selecting Desktop Manager from the Tools pull-down menu. Scroll down to the end of the list. The ten AutoCAD compatible *palettes* (toolbars) are at the end of the selection list and are identified by the (PACad) suffix. To open a toolbar, select a palette name, then click a Palette button. Repeat until all ten palettes are loaded.

For more information about the AllenCad user interface and the desktop layout, see the topics in *The Programs User Interface* section at beginning of this Help file.

## Multiple Document Interface

AllenCad can open as many as four drawings at a time. In addition, each can display up to four viewports. Each drawing appears in an individual drawing window.

The advantages to open multiple drawings on the program's desktop are that you can:

1. Copy objects from one open drawing to another.
2. Simultaneously view and edit blocks utilized in the current drawing.
3. Simultaneously view and edit externally referenced (xref) drawings.

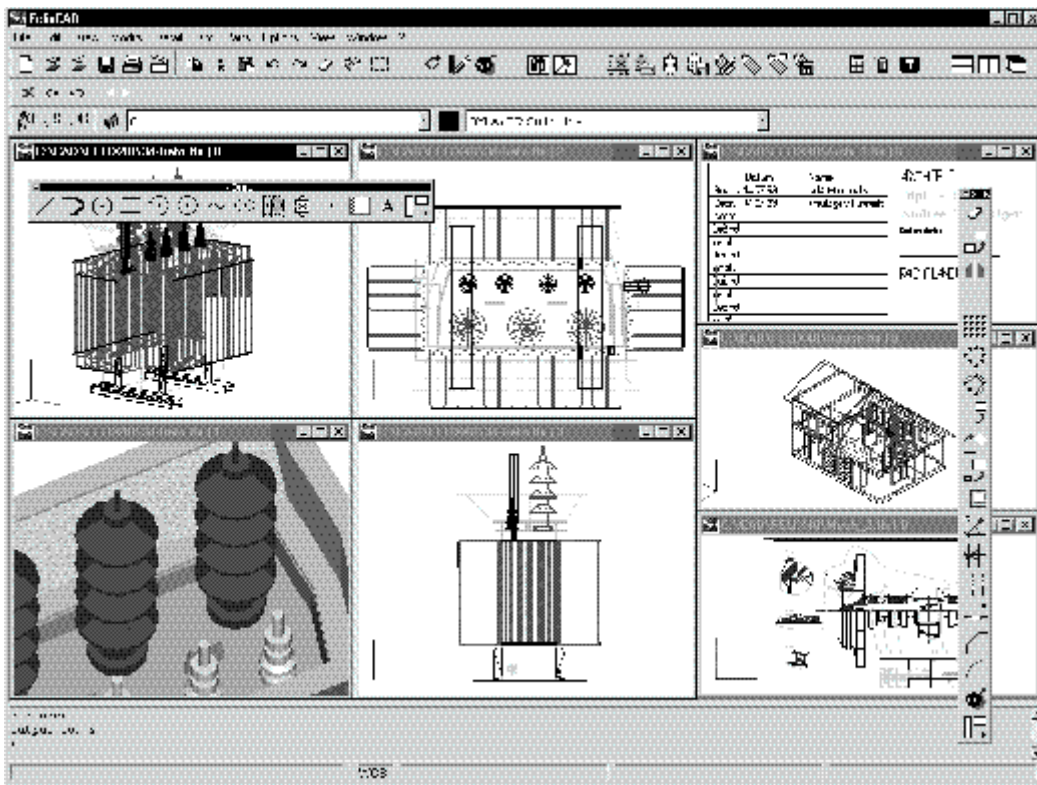


Figure i.3: AllenCad displays up to four drawings, each with four viewports, for a total of sixteen windows. In this figure, one drawing has four viewports open (left and center), with another three drawings to the right.

## Options Menus and Dialog Support

From the start, AllenCad has been designed for Windows integration. Wherever possible, AllenCad displays a dialog box to allow you to make your choices.

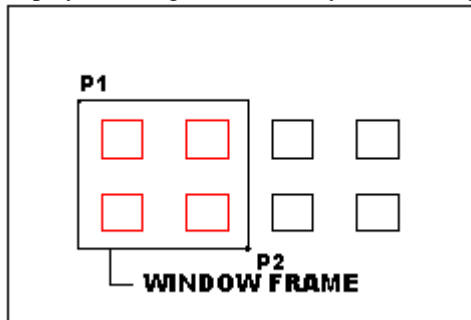


Figure i.4: Many AllenCad commands display a dialog box. This dialog box is displayed by the Settings command.

When a precise specification (like coordinate input) is required, AllenCad displays the command at the command prompt. In this case, in addition to the Command Prompt area information, AllenCad displays a context-sensitive option menu or option toolbar. You can choose which you prefer with the DeskConf command: select File > Desktop Manager and choose Menu or Toolbar from the dialog presented.

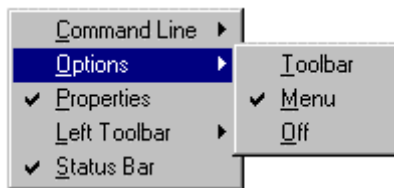


Figure i.5: Some AllenCad commands display an Option Menu (top) or Option Toolbar, depending which interface you prefer. These options are displayed by the Circle command.

## Drawing Navigator

The Drawing Navigator provides fast access to important non-graphical information of all the drawings that are open. You can view and check the names and status of layers, linetypes, dimension styles, or user coordinate systems defined in the drawings. You can use the Drawing Navigator to organize and manipulate these tables in your drawing.

You access the Drawing Navigator with the Xplorer command.

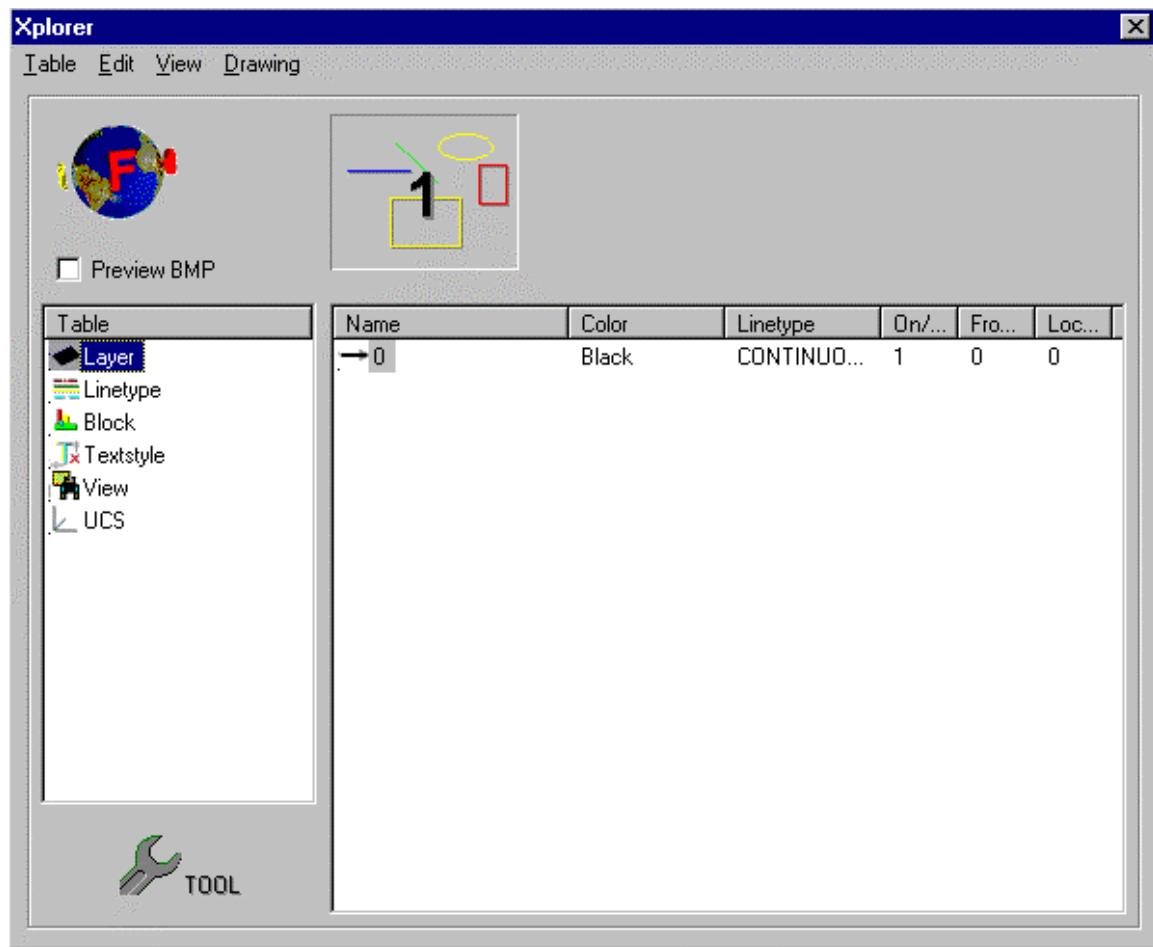


Figure i.6: The Drawing Navigator lists all data known about layers, linetypes, dimension styles, and user coordinate systems for all open drawings.

#### Additional User Interface Tools

AllenCad Pro is delivered with a number of tools and features that let you efficiently customize the user interface without wasting time in studying the syntax of support files:

The Desktop and Menu Editor (DME) allows you to lay out your menus, palettes, and dialog boxes in a visual environment; knowledge of programming is not required. You access the DME via the DlgEdit command. See Desktop and Menu Editor for more details.

The Image Library Tool organizes images (raster type: i.e. \*.BMP files, WMF files, and SLD slide files) in libraries. The images can be used in icon menus, dialog boxes, palettes, and toolbars. You access the Image Library Tool via the ImageLib command.





this section allows the organization of non-graphical objects in individual sub-sections called "Dictionaries." AutoCAD maintains two dictionaries: *Groups* (references of entities that belong to a group) and *Multiline Styles*. In addition, applications can store and manage non-graphical application data in separate Dictionaries.

You can similarly group entities in AllenCad with the Group command. AllenCad stores groups in a *table* named GROUPS. This is slightly different from AutoCAD, but has the advantage that you can apply the same LISP and C API functions to groups as you do to layers, linetypes, etc.

AllenCad lets third-party applications store and manage their non-graphical application data in a table named APPDATA. Again, this is slightly different from AutoCAD but has the advantage that you can apply the same LISP and C API functions to groups as you can with layers and linetypes.

## Notes to Application Developers

1. Since version 4.0, AllenCad accesses symbol table entries via entity names (enames). This means that applications can now use the entmake and entmod functions to create and modify table entries, respectively.
2. The new tblobjname function returns the entity name of an existing table record.
3. Attaching extended entity data (xdata) to entries of symbol tables is not yet supported. This has been allowed in AutoCAD since Release13.

### Command Line Area and Text Window

As with AutoCAD, AllenCad provides a command line area and a text window. Notice that the above examples show that the AllenCad command prompt is an angle bracket, >, rather than AutoCAD's Command: prompt.

Pressing F2 will display the Command History text window that views the command history. Opening the text window is also useful when evaluating and testing LISP expressions.



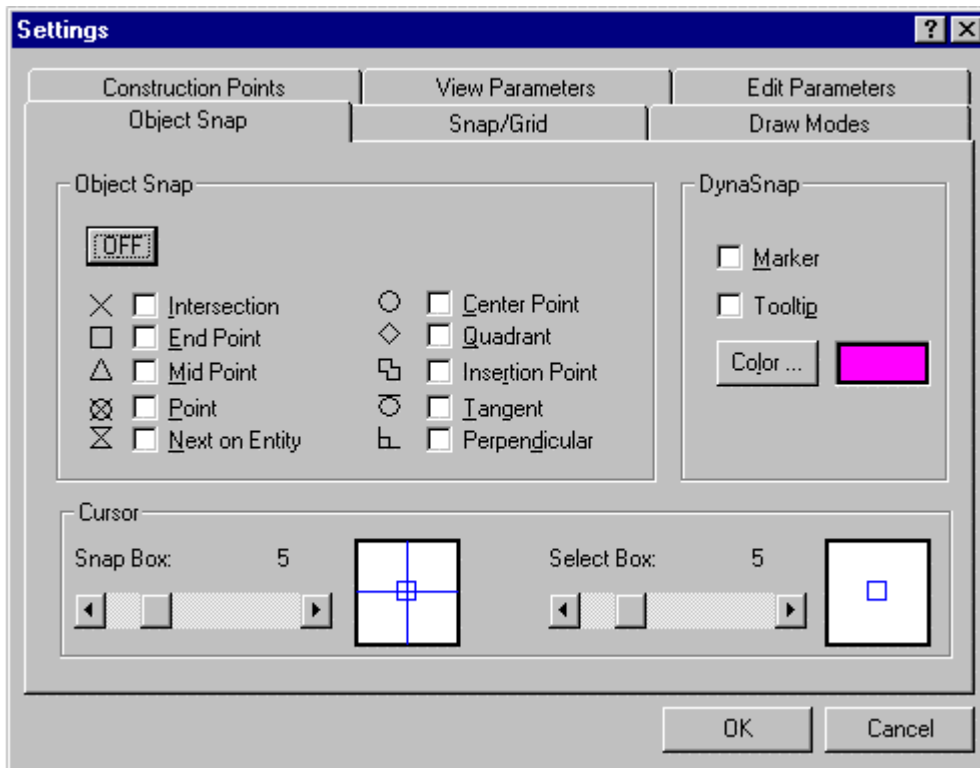
The AllenCad command line.

You can use the MS Windows standard Ctrl+C and Ctrl+V commands to copy and paste text from and to the command line and text window. As an alternative, the user can click on the up and down scroll arrows located at the right side of the AllenCad command line. This allows the user to review command history 'in place' a few lines at a time.

### Dialog Boxes vs Command-Line Input

From the beginning, AllenCad has used dialog boxes for any command for which they made sense. For example, AllenCad's Layer command has always displayed a dialog box, whereas AutoCAD didn't switch its Layer command to a dialog box until Release 14.

Whereas earlier releases of AutoCAD required multiple steps of command line entries to adjust its settings, AllenCad has always offered clearly arranged dialog boxes to let you select your preferences at one go. A good example is SETTINGS command.



On the other hand, whenever a command requires precise coordinate and geometric input, AllenCad presents the command's prompts at the command line, such as the Line and Move commands.

### Invoking Dialog Box Commands

Since Release 9, AutoCAD commands that invoke a dialog box usually (but not always) have the DD prefix (short for "dynamic dialog"), such as DdStyle and DdLModes (the 'Layer' command with dialog box).

With AutoCAD Release 14, several commands were changed to dialog boxes. In these cases, the dialog box version got the name of the command line variant, whereas the command line variant now requires a hyphen prefix, such as -Style and -Layer.

AllenCad instead uses "Q" commands for this purpose.

## "Q" Commands

In contrast to AutoCAD, program versions 3.1 and 4.0 added command-line variants to existing dialog box-based commands. To continue the tradition of the command naming convention, these commands are identified by the Q prefix in the command line version. For example, the Layer command displays a dialog box and the QLAYER command displays prompts at the command line. More examples are shown below. We call these "Q commands."

A consistent command naming convention between AllenCad and AutoCAD cannot be achieved for several commands, especially with introduction of the hyphen-commands in Release 14. The following table shows some of the differences:

**Table 2.11: Subtle Differences Between Commonly Used AutoCAD and FelixCAD Commands**

<b>AutoCAD Command</b>	<b>FelixCAD Command</b>	<b>Comments</b>
<b>QSave</b>	Qsave	In AutoCAD, if the drawing is named, the drawing is saved without requesting a filename. If the drawing is unnamed, the Save Drawing As dialog box comes up and allows to save the drawing under the file name the user specifies.
<b>Array</b>	Qarray	In FelixCAD, the Array command is processed differently from AutoCAD's Array command. However, the QArray command corresponds to the AutoCAD command in operation and options.
<b>Mirror</b>	Qmirror	In FelixCAD, the Mirror command is processed differently from AutoCAD's Mirror command. However, the QMirror command corresponds to the AutoCAD command in operation and options.
<b>Offset</b>	Offset Qoffset	In FelixCAD, the Offset command is processed different from AutoCAD's Offset command. The QOffset command corresponds to the AutoCAD command in operation and options.
<b>Text</b>	Qtext	In FelixCAD, the Text command uses a dialog box. FelixCAD's QText command corresponds to AutoCAD's Text command in operation and options.
<b>AttDef</b>	QattDef	The native FelixCAD AttDef command uses a dialog box. QAttDef corresponds to the AutoCAD AttDef command in operation and options.
<b>AttEdit</b>	QattEdit	The native FelixCAD command AttEdit uses a dialog box. QAttEdit corresponds to the AutoCAD AttEdit command in operation and options.
<b>-Layer</b>	Qlayer	The FelixCAD QLayer command corresponds to AutoCAD R14's -Layer command in operation and options.
<b>View</b>	Qview	The FelixCAD View command uses a dialog box. QView corresponds to AutoCAD's View command in operation and options.
<b>-Style</b>	Qfont	The FelixCAD QFont command corresponds to AutoCAD R14's command -Style in operation and options.
<b>CHPROP</b>	QPROPEDIT CHPROP	
<b>SELECT</b>	SELECT	Creates a selection set for use in subsequent editing commands.
<b>RENAME</b>	RENAME	
<b>PURGE</b>	PURGE	

### Unsupported Command Options

<b>CHPROP</b>	<b>Ltscale</b>
<b>EXTEND</b>	Edge Mode
<b>LENGTHEN</b>	
<b>TRIM</b>	Edgemode, Projectmode, Undo

### Transparent Commands

As in AutoCAD, many AllenCad commands can be applied transparently. This means that you can use such commands in the middle of another command using the ' (apostrophe) prefix. This is particularly useful in using commands like Move or Copy in conjunction with Zoom, for instance.

## Note to Application Developers

Since AllenCad version 4.0, applications defined by the defun function via *C:command* can be called transparently. To allow this, your function must not contain the function command itself; recursion is not permitted.

### Command Scripts

As in AutoCAD, you can create scripts with AllenCad. The MACRO command (known as SCRIPT in AutoCAD) executes command sequences in the same manner as in AutoCAD. The SCRIPT command is also recognized in AllenCad for those more familiar with this command name.

AllenCad's *Command Line Editor* is a built-in tool to paste portions of the text window into and then edit and test command sequences. You then can copy the script into the Windows clipboard to save your script file with a text editor like Notepad.

**Note:** Notepad can be called from within AllenCad by issuing the EDITOR command.

#### Note to Application Developers:

In AllenCad, the default extension for macros (script) files is .mcr, whereas AutoCAD uses the .scr extension. You can, however assign any extension you wish to a macro file.

## For Further Information

Detailed information on the topics discussed in this section is found in the *Customization and Programmer's Guide*.

### Unique AllenCad Commands

<b>AllenCad Pro Commands</b>	<b>Command Description</b>
3DELLIPS	Creates a 3d ellipsoid
3DSPHERE	Creates a 3D sphere
ARENDER	Autorendering
ATTDEDIT	Modify existing attribute definition
'ATTMOVE	Move attribute text
(Application)	
ATTROT (Application)	Rotate attribute text
ATTVALUE	Edit attribute text value
ATTXEDIT	Edit Part (block) attributes
BATPROC	Patch processing - Plots and conversion
(Application)	
BOUNDARY	Creates a boundary polyline (command line version of BPOLY)
CHAIN	Draws single entity continuous lines
CLOSE	Exit the drawing but not close the program
CONFIG	Configure program operating parameters
COPYPROP	Change object properties to that of selected entity
DELFLK	Deletes AllenCad lock files (*.flk)
DELPARTIAL	Delete part of an entity
DESKCONF	Configures the desktop options
DGNIN	Imports a Microstation .dgn drawing file
DGNOUT	Exports a Microstation .dgn drawing file
DIGICONFIG	Digitizer configuration
DIGIOFF	Switch digitizer mode off
DIGION	Switch digitizer mode on
DIMA4P	Creates an angular dimension out of four definition points
DIMHOR	Horizontal dimensioning
DIMOBL	Modifies the orientation of existing linear dimensioning objects.
DIMREST	Reinstate a stored style file so that it can be reinstated as the current style
DIMROT	Rotate existing dimensions
DIMSAVE	Allows the user to combine the individual parameters and store them in a named dimensioning style
DIMSER	Adds multiple additional dimension labels in the same orientation
DIMTHOME	Undoes the movement of dimension text
DIMTMOVE	Moves dimension text
DIMTNEW	Modify Dimension text value
DIMTROT	Rotate dimension text
DIMTYPE	Accesses dimensioning dialog
DIMUPD	Updates dimension
DIMVER	Labels dimensions along the Y (vertical dimension) axis
DINSERT	Insert a part without registering it with the drawing
DLGEDIT	Desktop and Menu editor
DOFFSET	Distance offsetting of entities
DRAWMODE	Specify drawing modes for subsequent entities
DZOOM	Dynamic zooming
EDITPAR	Dialog to change edit parameters
FCOMPIL	Compile a shape file to a binary font file
FLIP	Mirror and delete original entity
FONT	Loads and customizes font files
HPUPDATE	Updates changes in Hatch pattern assignments
ICONMENU	Presents graphics for dialog presentation and selection
LOADLTYPE	Loads linetype definitions from the command line
PALCLOSE	Removes a palette from the desktop
PALETTE	Places a palette on the desktop

PALMAN	Opens the palette manager
PALOPEN	Opens a new palette
'PANDOWN	Pans down
'PANLEFT	Pans left
'PANRIGHT	Pans right
'PANUP	Pans up
PARTLIB	Calls the part library manager
PCONTOUR	Creates a boundary polyline on top of other entities
PLANVIEW	Returns a 3d view to being planar to current UCS
PLBSETUP	Sets up or modifies existing part (block) libraries
POLY3D	Draws 3D polylines
PSETUP	Sets up print devices
QARRAY	Command line version of ARRAY command
QBMPOUT	Command line version of BMPOUT command
QFONT	Command line version of FONT command
QHATCH	Command line version of HATCH command
QHPEDIT	Command line version of HPEDIT command
QINSERT	Command line version of INSERT command
QLAYER	Command line version of LAYER command
QMTEXT	Command line version of MTEXT
QOPEN	Command line version of OPEN command
QPRINT	Command line version of PRINT command
QPROPEDIT	Command line version of PROPEDIT command
QSLDOUT	Command line version of SLDOUT command
QVIEW	Command line version of VIEW command
QVIEWPORT	Command line version of VIEWPORT command
QWMFOUT	Command line version of WMFOUT command
QWOPEN	Command line version of WOPEN command
QXLINK	Command line version of XLINK command
RTEXT	Writes new text with same properties as a selected text entity
SAVEALL	Saves all open drawings
SENDALL	Sends all drawings via Microsoft Exchange
SENDFLX	Sends the current drawing via Microsoft Exchange
SETCOLOR	Sets an explicit color for use in subsequently drawn objects
'SETLAYER	Change current layer to selection
SETLIB	Displays current part library dialog
SETUP or UNITSETUP	Sets up drawing units
TABCONFIG	Configure tablet menu areas
TABSECTION	Replaces a section of the tablet menu
TEXTEDIT	Modifies a line of text
'TGRID	Toggles grid display on/off
TRAPEZOID	Creates a 3D trapezoid
'TSCREENFULL	Toggles command line window on/off (full screen)
'TSNAP	Toggles snap modes on/off
VIEWPAR	Selects view parameters
WCASCADE	Arrange windows in cascade format
WCLOSE	Closes the current viewport
WICONARR	Arranges all drawing window icons
WOPEN	Opens new viewport
WTILE	Arranges Windows in a tiled format
WTILEHOR	Arranges Windows in a horizontally tiled format
WTILEVER	Arranges Windows in a vertically tiled format
XINSERT	Attaches an external reference (Xref)
XPLOER	Accesses and allows alteration of the current drawings properties

## Controlling Toolbars and Windows

The table below lists the commands that allow flexible handling of AllenCad's *palettes* (known as toolbars in AutoCAD). In addition,



A AllenCad palette (toolbar).

Click the small box in the upper-left corner to dismiss the palette. Click the arrow in the lower-right corner to toggle between horizontal and vertical orientation.

### Commands for Controlling Palettes in AllenCad

AllenCad Command	Comments
DeskConf	Determines the layout of the program's desktop.
PalClose	Closes the specified palette.
Palette	Opens a palette specified by filename.
PalMan	Palette Manager.
PalNext	Open a specified palette at the next available position (can be applied in command macros, menus, and in the programming interfaces).

AllenCad can handle up to four drawings in one session, each with up to four viewports – for a total of 16 windows. AllenCad provides the following commands to organize and control multiple windows:

### Commands for Controlling Windows in AllenCad

AllenCad Command	Comments
SaveAll	AllenCad can handle up to four drawings in one session. This command saves all open drawings.
Wcascade	Cascade all drawing viewport windows.
Wclose	Close the current viewport window.
WiconArr	Arrange drawing viewport icons.
Wopen	Opens a new viewport window of the current drawing (maximum. four viewports.) Opens a new viewport window of the current drawing (command line variant)
QWOpen	
Wtile	Tile all drawing windows.
WtileHor	Tile all windows horizontally.
WtileVer	Tile all windows vertically.



## Commands With Enhanced Options

AllenCad has enhanced several commands by providing them with additional options. The table below lists examples of commands with enhanced options.

### Commands with Enhanced Options in AllenCad

Command	Option	Comments
2Dface	Append	Continues 2D face creation at the edge of an existing face.
Arc	Append	Appends an arc to arc or line in tangent direction.
Copy	Rel.Point	Copies objects relative to a specified point.
Line	Segments	Draws subsequent line in individual segments.
	Append	Appends a line to another line or arc in tangent direction.
	TT	Creates tangent to arcs or circles.
Move	Rel.Point	Moves objects relative to a specified point.
Offset	BothSides	Copies an entity parallel on both sides.
Polyline	Append	Appends a polyline to another line or arc in tangent direction.

## Preference and Configuration Commands

A number of AutoCAD commands serve solely to set preferences for subsequent commands. These commands, in fact, modify system variables.

In AllenCad, you can use the SETVAR command to set system variables as in AutoCAD. The setting of preferences is also done in dialog boxes, primarily with the Settings command. The Settings dialog box has tabs for grouping related preferences: Object Snap, Snap/Grid, Draw Modes, Construction Points, View Parameters, and Edit Parameters.

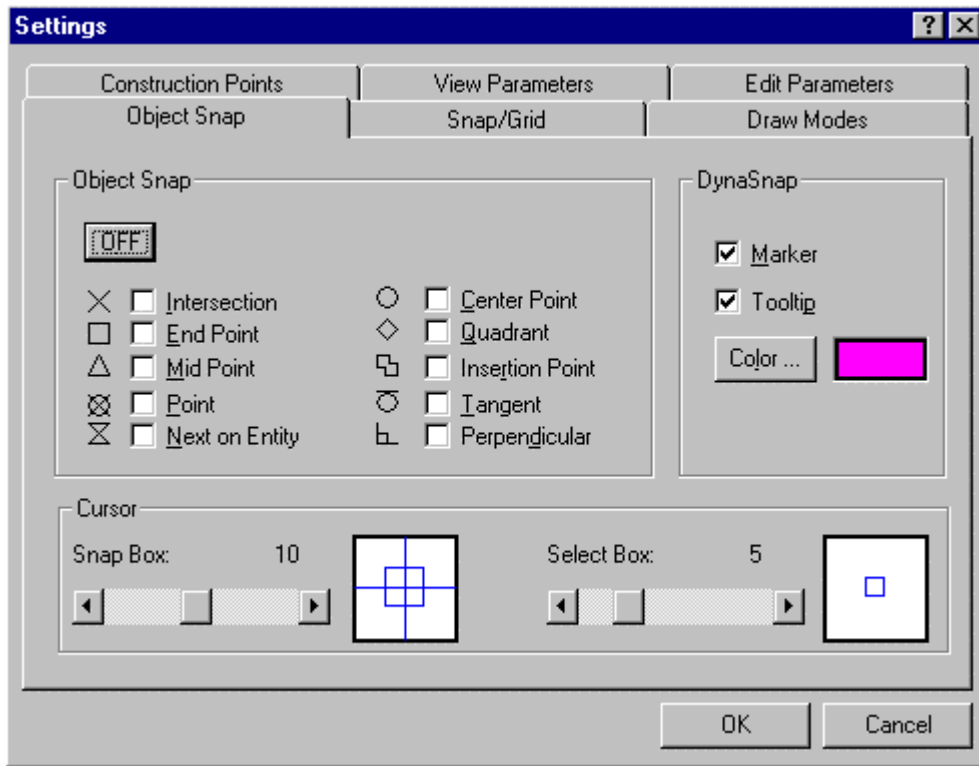


Figure 2.5: The Settings dialog box controls many system variables.

## Notes to Applications Programmers

The following table shows AutoCAD commands and their AllenCad alternatives specific to setting up drawing preferences. The table also lists the system variable related to these commands.

### Commands and System Variables for Setting Drawing Preferences

AutoCAD Command	AllenCad Command	Related System Variables
Aperture	'PrecPar or 'Settings	(setvar "SNAPBOX" number)
AttDisp	'Drawmode or 'Settings > Drawmode	(setvar "ATTMODE" mode)
Base	'InsBase	(setvar "INSBASE" point)
BlipMode	'Settings > View Parameters	(setvar "BLIPMODE" 0   1) 0 = Off 1 = On
DragMode	'Settings > View Parameters	(setvar "DRAGMODE" 0   1   2) 0 = Off 1 = On 2 = Auto
Elev	'Elevation and 'Thickness	(setvar "ELEVATION" number) (setvar "THICKNESS" number)
Fill	'DrawMode or 'Settings > DrawMode	(setvar "FILLMODE" 0   1) 0 = Off 1 = On
Grid	'PrecPar or 'Settings > Snap/Grid or	GRIDMODE and GRIDUNIT

	'Tgrid	
IsoPlane	'PrecPar or 'Settings > Snap/Grid	(setvar "SNAPISPAIR" 0   1   2 ) 0 = Left 1 = Right 2 = Top
Limits	C:Limits	(setvar "LIMMIN" number) and (setvar "LIMMAX" number)
LtScale	'DrawMode or 'Settings > DrawMode	(setvar "LTSCALE" number)
Snap	'PrecPar or 'Settings > Snap/Grid or 'Tsnap	SnapMode, SnapStyl, SnapAng, SnapUnit, and SnapBase
Units	UnitSetup	LUnits, LUPrec, AUnits, AUPrec, AngDir, DefLUnits, DefLUPrec, DefAUnits, DefAuPrec, and DefAngDir
ViewRes	'CircleRes	(setvar "CIRCLERES" number)

## Other Command Differences

In addition to differences in commands for setting drawing preferences, AllenCad handles a number of other AutoCAD commands differently. The following table lists these differences.

### Differences Between AutoCAD and AllenCad Commands

AutoCAD Command	AllenCad Command	Comments
MVSetup (mvsetup.lsp)	UnitSetup and FCTemplate	MVSetup assists with setting up a new drawing in AutoCAD. In AllenCad, third-party applications are available that provide a similar Setup Wizard to set the preferences for new drawings. TIP: You can set the defaults for new drawings with the UnitSetup command, and by setting a default prototype drawing via the system variable FCTemplate.
Plot (CmdDia = 0)	Print	The name and the design of the dialog boxes are different in AutoCAD and AllenCad, but the commands do a similar job.
Plot (CmdDia = 1)	QPrint	In AutoCAD, the Plot command displays prompts on the command line when the CMDDIA system variable is set to 0 (off). In AllenCad, CmdDia is unknown. Instead, use the Qprint command.
Preview	Print	In AutoCAD, the Preview command displays a full-page preview of the drawing. The preview is based on the current plot configuration, as defined

		by the Plot command. In AllenCad, the plot preview feature is integrated in the Print command. Select the full-page button to display a full-page plot preview of the current drawing.
VPLayer	ViewPort Layer or PLayer QViewPort Layer or PLayer	In AutoCAD, the VPLayer command makes a layer visible in one or more paperspace viewports and invisible in other viewports. In AllenCad, use the command ViewPort or QViewPort with option Layer or PLayer to control layer visibility.
Vpoint DDVPoint	SetViewDir 3DView	These commands set the viewing direction for 3D viewpoints.

## Dialog Box Functions in FDT

FDT provides equivalent dialog box functions to FLISP. For information on the meaning of the functions please refer to the FLISP section on the previous pages.

### Comparison of ADS and FDT Dialog Box Functions

<b>ADS Dialog Box Function</b>	<b>FDT Dialog Box Function</b>
ads_action_tile()	Dlg_TileAction()
ads_add_list()	Dlg_ListAdd()
ads_client_data_tile()	Dlg_TileClientData()
ads_dimensions_tile()	Dlg_TileDimensions()
ads_done_dialog()	Dlg_DialogDone()
ads_done_positioned_dialog()	Dlg_DialogDonePositioned()
ads_end_image()	Dlg_ImageEnd()
ads_end_list()	Dlg_ListEnd()
ads_end_list()	Dlg_ListEnd()
ads_fill_image()	Dlg_ImageFill()
ads_get_tile()	Dlg_TileGet()
ads_mode_tile()	Dlg_TileMode()
ads_new_dialog()	Dlg_DialogNew()
ads_new_positioned_dialog()	Dlg_DialogNewPositioned()
ads_set_tile()	Dlg_TileSet()
ads_slide_image()	Dlg_ImageSlide() See also: Dlg_ImageLibrary()
ads_start_list()	Dlg_ListStart()
ads_term_dialog()	Dlg_DialogTerm()
ads_vector_image()	Dlg_ImageVector()

As described earlier (in the section that discussed the equivalent LISP functions), the FDT function to

start a dialog is required to pass a callback function as a third function. This contains the code to initialize the dialog. In contrast, ADS initialization is assembled between the `ads_new_dialog()` and `ads_start_dialog()` functions.

```
int ads_start_dialog(          int Dlg_DialogStart(
ads_hdlg hdlg,                const dlg_hdlg hdlg,
int *status);                 int* status,
                              const DLGINITFUNC init_dialog_callback);
```

## Programmable Dialog Boxes

DCL (Dialog Control Language) is used by AutoLISP and ADS to define the layout of dialog boxes in AutoCAD. The drawback to DCL is that Autodesk does not provide a visual tool to design the layout of dialog boxes.

As with AutoCAD, AllenCad allows you to create dialog boxes for your LISP and C applications. AllenCad DLG files contain dialog box descriptions, just as DCL files do. The format and syntax of these files, however, differs. AllenCad does not (yet) provide direct support of AutoCAD DCL syntax.

However, AllenCad comes with a tool that dramatically shortens the process of converting DCL files into DLG files. The following sections describe the differences between AutoCAD and AllenCad dialog boxes.

## Cad Support Files

In AllenCad, you use:

The same font files as in AutoCAD (*.shx*), including BigFonts.

The same linetype definition files as in AutoCAD (*.lin*).

The same hatch pattern definition files as in AutoCAD (*.pat*).

The FCOMPILE command compiles text font source files (*.shp*) to binary text font files used in AllenCad (*.fsh*).

## Menus and Toolbars

As you probably already know, AutoCAD defines the bulk of its user interface items in a single menu file.

AllenCad chose a different approach, where the program uses individual files to define different aspects of the user interface. Each file covers different menu types and user interface items in separate files. The advantages are:

- The maintenance and testing effort is reduced. To fix an error, you don't need to access to the entire menu system.
- Applications do not need to change a complete system when enhancements or changes are provided to the end user. For example, a third-party developer can add a single palette or icon menu to the system, without modifying other components of the menu system.

The following table lists the file types related to the menu types and user interface items utilized by AllenCad:

#### **File Extensions Used by AllenCad Support Files**

<b>File Extension</b>	<b>Comments</b>
.mnu	Pull-down menu. Located in \SUPPORT folder.
.mnp	Palettes (toolbars). Located in \SUPPORT folder.
.mni	Icon menu (image menu). Located in \SUPPORT folder.
.mnt	Tablet menu. Located in \SUPPORT folder.
Desktop.cfg	Vertical and horizontal toolbars of the program's desktop. Located in \cfg folder.

#### **Release Notes on Dialog Box Functions**

The FLISP functions Dlg\_ImageSlide, Dlg\_ImageLib, and Dlg\_ImagePrev (respectively Dlg\_ImageSlide(), Dlg\_ImageLibrary(), and Dlg\_ImagePreview() in FDT) have been added in version 4.0 of AllenCad.

FLISP 4.0 has implemented the following dialog box functions with AutoLISP-like names that are equivalent to existing FLISP function:

#### **New FLISP Dialog Box Function Names**

<b>AutoLISP Function</b>	<b>FLISP 4.0 Function</b>
Action_tile	dlg_tileaction
add_list	dlg_listadd
client_data_tile	dlg_tileclientdata
dimx_tile	dlg_tiledimx
dimy_tile	dlg_tiledimy
done_dialog	dlg_dialogdone
end_image	dlg_imageend
end_list	dlg_listend
fill_image	dlg_imagefill
get_tile	dlg_tileget
load_dialog	dlg_dialogload
mode_tile	dlg_tilemode
new_dialog	dlg_dialognew
set_tile	dlg_tileset
slide_image	dlg_imageslide

start_dialog	dlg_dialogstart
start_image	dlg_imagestart
start_list	dlg_liststart
term_dialog	dlg_dialogterm
unload_dialog	dlg_dialogunload
vector_image	dlg_imagevector

## Unsupported ADS Functions

These ADS functions are not supported in FDT:

### Unsupported ADS Functions

ADS Function	Comments
ads_get_attr()	Not supported, provided from struct dlg_callback_packet-member 'key'
ads_get_attr_string()	Not supported, provided from struct dlg_callback_packet-member 'key'

## Unique FDT Functions

The following table lists dialog box functions that are unique to FDT:

### Unique FDT Functions

FDT Function
Dlg_TileSetFont()
Dlg_ListSetTabStops()
Dlg_ListSetColumnWidth()
Dlg_SliderGet()
Dlg_SliderSet()
Dlg_ImageBmp()
Dlg_ImagePreviewBmp()
Dlg_ImageLibrary()
Dlg_ImageWmf()

## Supported AutoCAD Objects

The following table lists the geometric entities that are identical in AutoCAD and AllenCad:

<b>AutoCAD-AllenCad Object</b>	<b>Comments</b>
3DFACE	3D face entity.
ARC	Arc entity.
ATTDEF	Attribute definition.
ATTRIB	Attribute of a block insertion.
CIRCLE	Circle entity.
DIMENSION (R12 and earlier)	Dimension (complex entity).
BLOCKS & WBLOCKS	Block insertion (block reference, internal & external).
LINE	Line entity.
POINT	Point entity.
POLYLINE	Polyline (complex entity).
SOLID	Filled 2D face entity.
TEXT	Text entity.
VIEWPORT	Paper space viewport.
<i>Note:</i> Dimensioning and hatching is associative in AllenCad, as in AutoCAD	

#### Converted AutoCAD Objects

The following table lists DWG geometric entities that are converted by AllenCad:

<b>AutoCAD Object</b>	<b>AllenCad Entity</b>	<b>Comments</b>
TRACE (Introduced to AutoCAD with version 1)	3DFACE	In AutoCAD's drawing database definition, the TRACE object is identical to the 3DFACE object. When reading an AutoCAD drawing, AllenCad converts the TRACE object to a 3DFACE entity.

#### Ignored Objects

The following table lists AutoCAD geometric objects that are not displayed by AllenCad. When objects are not displayed in AllenCad, they are maintained in the .dwg file. This ensures that the objects remain intact when displayed by AutoCAD.

#### AutoCAD Objects Not Supported by AllenCad

<b>AutoCAD Object</b>	<b>Comments</b>
SHAPE (AutoCAD with version 1)	Shape entity used in shapes, complex linetypes, and tolerances.
3DSOLID (Release 13)	ACIS object.



BODY (Release 13)	ACIS object.
LEADER (Release 13)	Dimension: leader line and text.
MLINE (Release 13)	Multiline.
OLEFRAME (Release 13)	OLE (object linking and embedding) object.
ZOMBIE (Release 13)	ObjectARX-defined object.
PROXY (Release 14)	
RAY (Release 13)	Semi-infinite construction line.
REGION (Release 13)	ACIS object.
TOLERANCE (Release 13)	Dimension: geometric tolerance.
XLINE (Release 13)	Infinite construction line.
IMAGE (Release 14)	Raster image object.

## Unsupported Commands

The following groups of AutoCAD commands are not found in AllenCad:  
ACIS and 3DStudio commands. A 3D solid modeler is currently being developed for AllenCad.

ASE (AutoCAD SQL Extension) commands: AseAdmin, AseExport, AseLinks, AseRows, AseSelect, and AseSqlEd. AllenCad does not provide an interface to external database files.

Other commands, as listed in the Unsupported Command Chart have no direct command translation.

### **Note to Application Developers**

Some of the subtle differences of the more common AutoCAD commands and their equivalent AllenCad commands are listed in the following table:

## Subtle Differences Between Commonly Used AutoCAD and AllenCad Commands

AutoCAD Command	AllenCad Command	Comments
QSave	Qsave	In AutoCAD, if the drawing is named, the drawing is saved without requesting a filename. If the drawing is unnamed, the Save Drawing As dialog box comes up and allows to save the drawing under the file name the user specifies.
Array	Qarray	In AllenCad, the Array command is processed differently from AutoCAD's Array command. However, the QArray command corresponds to the AutoCAD command in operation and options.
Mirror	Qmirror	In AllenCad, the Mirror command is processed differently from AutoCAD's Mirror command. However, the QMirror command corresponds to the AutoCAD command in

		operation and options.
Offset	Offset	In AllenCad, the Offset command is processed different from AutoCAD's Offset command.
	Qoffset	The QOffset command corresponds to the AutoCAD command in operation and options.
Text	Qtext	In AllenCad, the Text command uses a dialog box. AllenCad's QText command corresponds to AutoCAD's Text command in operation and options.
AttDef	QattDef	The native AllenCad AttDef command uses a dialog box. QAttDef corresponds to the AutoCAD AttDef command in operation and options.
AttEdit	QattEdit	The native AllenCad command AttEdit uses a dialog box. QAttEdit corresponds to the AutoCAD AttEdit command in operation and options.
-Layer	Qlayer	The AllenCad QLayer command corresponds to AutoCAD R14's -Layer command in operation and options.
View	Qview	The AllenCad View command uses a dialog box. QView corresponds to AutoCAD's View command in operation and options.
-Style	Qfont	The AllenCad QFont command corresponds to AutoCAD R14's command - Style in operation and options.

## Unsupported Command Options

/3Ltscale

Edge Mode

Edgemode, Projectmode, Undo

**Unsupported Command Options**

<b>CHPROP</b>	<b>Ltscale</b>
<b>EXTEND</b>	Edge Mode
<b>LENGTHEN</b>	
<b>TRIM</b>	Edgemode, Projectmode, Undo

### "Round-Trip" Drawing Exchange Capabilities

AllenCad ensures that when you open an AutoCAD drawing, all unsupported AutoCAD entities are retained in the drawing database, even though the AllenCad program does not display them. This

capability ensures that you can exchange drawing files with AutoCAD 2000 workstations without disabling or removing unsupported AutoCAD entities from the drawing database.

"Round-Tripping", as this is called, assumes that the file is saved in AutoCAD 2000 format (.dwg or .dxf). When saving a drawing in an AutoCAD version format other than AutoCAD 2000, some difficulties may be encountered if the original source of the database information contains unsupported entities obtained from an AutoCAD 2000 drawing file. In this instance, users may wish to use AutoCAD's SAVEAS function to remove these entities, either in ALLENCAD's input or output file. Enhanced SAVEAS functionality to better handle unsupported ACAD 2000 entities with earlier versions, is planned for a later service pack release of the program

#### **"Round tripped" AutoCAD Entities:**

ARCALIGNEDTEXT  
 BODY  
 IMAGE  
 LEADER  
 MLINE  
 OLE2FRAME  
 PROXY  
 RAY  
 REGION  
 RTEXT  
 SOLID3D  
 TOLERANCE  
 WIPEOUT  
 XLINE  
 XRECORD-OBJECTS

#### **"Round tripped" AutoCAD Table Information:**

DICTIONARYWDFLTOBJTYPE  
 DICTVAR  
 IMAGEDEF  
 MLINestyle  
 OBJECTPOINTER  
 PLACEHOLDEROBJECT  
 VBAPROJECTOBJECT  
 WIPEOUTVAROBJECT

#### **Command Compatibility Chart**

The following AutoCAD commands are fully supported by AllenCad:

<b>AutoCAD 2000 Commands</b>	<b>AllenCad 2000 Commands</b>	<b>Alias</b>	<b>Command Description</b>
#NAME?	QXLINK	QX	Controls external references to drawing files (Command line version)
#NAME?	XREF	XR	Controls external references to drawing files (Command line version)
'GRID	See TGRID		Displays a dot grid in the current viewport
3D	3DOBJECTS (Application)	3O	Creates three-dimensional polygon mesh objects
3DFACE	3DFACE	3F	Creates a three-dimensional face

3DMESH	3DMESH	3M	Creates a free-form polygon mesh
3DPOLY	POLY3D	3P	Creates a polyline of straight line segments in three-dimensional space
'ABOUT	'ABOUT	AB	Displays information about the Program
'APPLOAD	'APPLOAD	AP	Loads and unloads applications and defines which applications to load at startup
ARC	ARC	A	Creates an arc
AREA	AREA	AR	Calculates the area and perimeter of objects or of defined areas
ARRAY	ARRAY	RA	Creates multiple copies of objects in a pattern
ATTEDIT	QATTEDIT	QAE	Changes attribute information independent of its block definition (Command line version)
ATTEXT	'ATTEXP (Application)	ATX	Extracts attribute data
BHATCH	BHATCH	BH	Fills an enclosed area with a hatch pattern
BLOCK	PARTDEF	PDF	Define a part (block)
BMAKE	BMAKE	BK	Defines a block using a dialog box
BMPOUT	BMPOUT	BO	Saves selected objects to a file in device-independent bitmap format
BOUNDARY	BPOLY	BP	Creates a region or a polyline from an enclosed area
BOX	3DBOX	3B	Creates a three-dimensional solid box
BREAK	BREAK	B	Erases parts of objects or splits an object in two
CHAMFER	CHAMFER	CH	Bevels the edges of objects
CHANGE	CHANGE	CV	Changes the properties of existing objects
CHANGE	PROPEDIT	PED	Changes the properties of existing objects
CHPROP	CHPROP	CHP	Changes the color, layer, linetype, linetype scale factor, and thickness of an object
CIRCLE	CIRCLE	CI	Creates a circle
COMPILE	DWFCOMPILE, FCOMPILE, OEMFCOMPILE		Compiles shape files and PostScript font files (Compiles AllenCad fnt font files and shp font files)
CONE	3DCONE (APPLICATION)	3DC	Creates a three-dimensional solid cone
COPY	COPY	C	Duplicates objects
COPYCLIP	COPYSELECT	CST	Copies objects to the Clipboard
CUTCLIP	CUTSELECT	CT	Copies objects to the Clipboard and erases the objects from the drawing
DDATTDEF	ATTDEF	AD	Creates an attribute definition
DDATTE	ATTXEDIT	AX	Edits the variable attributes of a block
DDATTEXT	ATTEXP	ATX	Extracts attribute data
DDCHPROP	PROPEDIT	PED	Changes the color, layer, linetype, and thickness of an object
DDCOLOR	'SETCOLOR	STC	Sets the color for new objects
DDEDIT	TEXTEDIT	ED	Edits text and attribute definitions
DDIM	DIMDRAW (Application)	DDR	Creates and modifies dimension styles
DDIM	DIMTYPE	DDR	Creates and modifies dimension styles
DDINSERT	INSERT	I	Inserts a block or another drawing
DDPTYPE	DRAWMODE	DR	Specify drawing modes for subsequent entities
DDRENAME	TABSERVICE	TB	Changes the names of named objects

'DDRMODES	SETTINGS	ST	Sets drawing aids
'DDSELECT	SETTINGS	ST	Sets object selection modes
'DDUNITS	SETUP or UNITSETUP	US	Sets up drawing units
DDVIEW	VIEW	V	Creates and restores views
DDVPOINT	3DVIEW	3V	Sets the three-dimensional viewing direction
DIMALIGNED	DIMALI	DAL	Creates an aligned linear dimension
DIMANGULAR	DIMA3P	D3P	Creates an angular dimension
DIMBASELINE	DIMBAS	DBA	Continues a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension
DIMCENTER	DIMCENT	CM	Creates the center mark or the center lines of circles and arcs
DIMCONTINUE	DIMCON	DCN	Continues a linear, angular, or ordinate dimension from the second extension line of the previous or a selected dimension
DIMDIAMETER	DIMDIA	DDA	Creates diameter dimensions for circles and arcs
DIMEDIT	DIMEDIT (Application)	DED	Edits dimensions
DIMLINEAR	DIMLIN	DL	Creates linear dimensions
DIMORDINATE	DIMORD	DOD	Creates ordinate point dimensions
DIMOVERRIDE	DIMTNEW	DN	Overrides dimension system variables
DIMRADIUS	DIMRAD	DRA	Creates radial dimensions for circles and arcs
DIMTEDIT	See DIMEDIT DIMTROT & DIMTMOVE		Moves and rotates dimension text
'DIST	'DIST	D	Measures the distance and angle between two points
DIVIDE	DIVIDE (Application)	DI	Places evenly spaced point objects or blocks along the length or perimeter of an object
DONUT	RING	RI	Draws filled circles and rings
DRAWORDER	DRAWORDER	DO	Changes the display order of images and other objects
DTEXT	DTEXT	DT	Displays text on screen as it is entered
DVIEW	DVIEW	DV	Defines parallel projection or perspective views
DXFIN	DXFIN	DXN	Imports a drawing interchange file
DXFOUT	DXFOUT	DXO	Creates a drawing interchange file of the current drawing
EDGE	EDGE (Application)	EG	Changes the visibility of three-dimensional face edges
EDGESURF	3DMESH	3M	Creates a three-dimensional polygon mesh
'ELEV	'ELEVATION	EL	Sets elevation properties of new objects
'ELEV	'THICKNESS	TH	Sets extrusion thickness properties of new objects
ELLIPSE	ELLIPSE	ELL	Creates an ellipse or an elliptical arc
ERASE	DELETE	DEL	Removes objects from a drawing
ERASE	ERASE	E	Removes objects from a drawing
EXIT	EXIT	X	Exit the program
EXPLODE	EXPLODE	EX	Breaks a compound object into its component objects
EXPORT	See SAVEAS		Saves objects to other file formats

EXTEND EXTEND EXTRUDE	EXPAND EXTEND, See THICKNESS	EXP ET	Extends an object to meet another object Extends an object to meet another object Creates unique solid primitives by extruding existing two-dimensional objects
FILLET GROUP HATCH HATCHEDIT 'HELP (F1) HIDE	FILLET GROUP HATCH HPEDIT HELP (F1) Fhide	F G H HED ? FHD	Rounds and fillets the edges of objects Creates a named selection set of objects Fills a specified boundary with a pattern Modifies an existing hatch object Displays online help Creates a second drawing file with a hidden line view
HIDE	HIDE	HD	Creates a second drawing file with a hidden line view
'ID	'ID	ID	Displays the coordinate values of a location
IMPORT	See OPEN		Imports various file formats into The program
INSERT	INSERT	I	Places a named block or drawing into the current drawing
INTERSECT	INTERSECT	IN	LIGHTEDIT Creates composite solids or regions from the intersection of two or more solids or regions
'LAYER LENGTHEN LIGHT 'LIMITS	LAYER LENGTHEN LIGHT 'LIMITS (Application)	LA LEN LIT LM	Manages layers and layer properties Lengthens an object Manages lights and lighting effects Sets and controls the drawing boundaries
LINE 'LINETYPE 'LINETYPE LIST	LINE LINETYPE, SETLINETYPE EINFO (Application)	L LT SLT LI	Creates straight line segments Creates, loads, and sets linetypes Creates, loads, and sets linetypes Displays database information for selected objects
LOGFILEOFF	LOGFILEOFF	LOF	Closes the log file opened by LOGFILEON
LOGFILEON 'MATCHPROP	LOGFILEON COPYPROP	LON CP	Writes the text window contents to a file Copies the properties from one object to one or more objects
MEASURE	MEASURE (Application)	MR	Places point objects or blocks at measured intervals on an object
MENU MENULOAD MINsert	MENU PULLDOWN See ARRAY	MU PN	Loads a menu file Loads partial menu files Inserts multiple instances of a block in a rectangular array
MIRROR MIRROR3D	MIRROR 3DMIR (Application)	MI M3	Creates a mirror image copy of objects Creates a mirror image of objects about a plane
MOVE	MOVE	M	Displaces objects a specified distance in a specified direction
MSLIDE MSPACE	SLDOUT 'TLMODE	SO TM	Creates a slide file of the current viewport Switches from paper space to a model space viewport
MTEXT MVIEW	MTEXT VIEWPORT	MT VP	Creates multi-line text Creates floating viewports and turns on existing floating viewports
NEW OFFSET	NEW DOFFSET	N DOF	Creates a new drawing file Creates concentric circles, parallel lines,

OOPS	UNDELETE	UD	and parallel curves
OPEN	OPEN	OP	Restores erased objects
OPTIONS	SETTINGS	ST	Opens an existing drawing file
'ORTHO	'TORTHO	TO	Customizes the Drawing settings
'OSNAP	'PRECPAR	PCP	Constrains cursor movement
			Sets running object snap modes and changes the target box size
'PAN	'PAN	P	Moves the drawing display in the current viewport
PASTECLIP	PASTE	PST	Inserts data from the Clipboard
PEDIT	POLYEDIT,	PE	Edits polylines and three-dimensional polygon meshes (VTXEDIT edits polyline vertices)
			Edits polylines and three-dimensional polygon meshes (VTXEDIT edits polyline vertices)
PEDIT	VTXEDIT	VX	Edits polylines and three-dimensional polygon meshes (VTXEDIT edits polyline vertices)
			Creates a three-dimensional polyface mesh vertex by vertex
PFACE	PFACE	PF	Displays the plan view of a user coordinate system
PLAN	PLAN	PLN	Creates two-dimensional polylines
PLINE	POLYLINE	PO	Plots a drawing to a plotter, printer, or file
PLOT	PRINT	PT	Creates a point object
POINT	POINT	POI	Creates an equilateral closed polyline
POLYGON	NGON	NG	Creates an equilateral closed polyline
POLYGON	POLYGON,	PG	Customizes the program settings
PREFERENCES	PRECPAR,	PCP	Customizes the program settings
PREFERENCES	'SETTINGS	ST	Switches from a model space viewport to paper space
PSPACE	'TLMODE	TM	Removes unused named objects, such as blocks or layers, from the database
			Saves the current drawing
PURGE	PURGE	PRG	Controls the display and plotting of text and attribute objects
			Exits the program
QSAVE	QSAVE	QS	Repairs a damaged drawing
'QTEXT	QTEXT	QT	Draws a rectangular polyline
			Restores the program internal commands overridden by UNDEFINE
QUIT	EXIT	X	Reverses the effects of the previous UNDO or U command
RECOVER	FLXRECOVER	REC	Refreshes the display of the current viewport
RECTANG	RECTANGLE,	RT	Regenerates the drawing and refreshes the current viewport
	RECTANG		Regenerates the drawing and refreshes all viewports
REDEFINE	REDEFINE	RE	Creates a region object from a selection set of existing objects
	(Application)		Changes the names of objects
REDO	REDO	RD	Creates a realistically shaded image of a three-dimensional wireframe or solid model
			Creates a rotated surface about a selected axis
'REDRAW	REDRAW	R	
REGEN	REGEN	RG	
REGENALL	REGENALL	RGA	
REGION	See GROUP		
RENAME	RENAME	RNM	
RENDER	RENDER	REN	
REVSURF	REVSURF	RS	

ROTATE	ROTATE	RO	Moves objects about a base point
ROTATE3D	3DROT (Application)	R3	Moves objects about a three-dimensional axis
RPREF	RCONFIG	RC	Sets rendering preferences
RULESURF	RULESURF	RSF	Creates a ruled surface between two curves
SAVE	SAVE	S	Saves the drawing under the current file name or a specified name
SAVEAS	SAVEAS	SA	Saves an unnamed drawing with a file name or renames the current drawing
SCALE	SCALE	SC	Enlarges or reduces selected objects equally in the X, Y, and Z directions
'SCRIPT	'MACRO	MA	Executes a sequence of commands from a script
SELECT	SELECT	SL	Places selected objects in the Previous selection set
'SETVAR	'SETVAR	SV	Lists or changes the values of system variables
SHADE	RENDER	REN	Displays a flat-shaded image of the drawing in the current viewport
'SNAP	'TSNAP	TS	Restricts cursor movement to specified intervals. Toggles snap modes on/off
SOLID	2DFACE	2F	Creates solid-filled polygons
SPHERE	3DSPHER (Application)	3DS	Creates three-dimensional solid sphere
SPLINE	SPLINE (Application)	SP	Creates quadratic or cubic spline (NURBS) curve
SPLINEDIT	SPLINE;D	SPD	Edits a pline object
STRETCH	STRETCH	SR	Moves or stretches objects
'STYLE	FONT	FT	Creates or modifies named styles and sets the current style for text in your drawing
TABLET	TABLET	TA	Calibrates, configures, and turns on and off an attached digitizing tablet
TABSURF	TABSURF	TF	Creates a tabulated surface from a path curve and a direction vector
TEXT	TEXT	TX	Creates a single line of text
'TEXTSCR	'TSCREEN	TN	Opens the text window
TOOLBAR	PALETTE, PALMAN (Application)	PA,PM	Displays, hides, and customizes toolbars
TORUS	3DTORUS	3T	Creates a donut-shaped solid
TRIM	TRIM	T	Trims objects at a cutting edge defined by other objects
U	U	U	Reverses the most recent operation
UCS	UCS	UC	Manages coordinate systems
UCSICON	TICON,	TI	Controls the visibility and placement of the UCS icon
UCSICON	UCSICON	UI	Controls the visibility and placement of the UCS icon
UNDEFINE	UNDEFINE (Application)	UF	Allows an application-defined command to override an internal command
UNDO	UNDO	U	Reverses the effect of commands
'UNITS	'UNITSETUP	US	Sets coordinate and angle display formats and precision



## Unsupported Command Chart

<b>AutoCAD 2000 Commands</b>	<b>Command Description</b>
3DARRAY	Creates a three-dimensional array
3DARRAY	Creates a three-dimensional array
3DCLIP	Invokes the interactive 3D view and opens the Adjust Clipping Planes window
3DCLIP	Invokes the interactive 3D view and opens the Adjust Clipping Planes window
3DCORBIT	Invokes the interactive 3D view and enables you to set the objects in the 3D view into continuous motion
3DCORBIT	Invokes the interactive 3D view and enables you to set the objects in the 3D view into continuous motion
3DDISTANCE	Invokes the interactive 3D view and makes objects appear closer or farther away
3DDISTANCE	Invokes the interactive 3D view and makes objects appear closer or farther away
3DORBIT	Controls the interactive viewing of objects in 3D
3DORBIT	Controls the interactive viewing of objects in 3D
3DPAN	Invokes the interactive 3D view and enables you to drag the view horizontally and vertically
3DPAN	Invokes the interactive 3D view and enables you to drag the view horizontally and vertically
3DSIN	Imports a 3D Studio File
3DSIN	Imports a 3D Studio File
3DSOUT	Exports to a 3D Studio file
3DSOUT	Exports to a 3D Studio file
3DSWIVEL	Invokes the interactive 3D view and simulates the effect of turning the camera
3DSWIVEL	Invokes the interactive 3D view and simulates the effect of turning the camera
3DZOOM	Invokes the interactive 3D view so you can zoom in and out on the view
3DZOOM	Invokes the interactive 3D view so you can zoom in and out on the view
ACISIN	Imports an ACIS file
ACISIN	Imports an ACIS file
ACISOUT	Exports AutoCAD solid objects to an ACIS file
ACISOUT	Exports AutoCAD solid objects to an ACIS file
ADCCLOSE	Closes AutoCAD DesignCenter
ADCCLOSE	Closes AutoCAD DesignCenter
ADCENTER	Manages content
ADCENTER	Manages content
ADCNAVIGATE	Directs the Desktop in AutoCAD DesignCenter to the file name, directory location, or network path you specify
ADCNAVIGATE	Directs the Desktop in AutoCAD DesignCenter to the file name, directory location, or network path you specify
ALIGN	Moves and rotates objects to align with other objects
ALIGN	Moves and rotates objects to align with other objects
AMECONVERT	Converts AME solid models to AutoCAD solid objects
ARX	Loads, unloads, and provides information about ARX applications
ASEADMIN	Performs administrative functions for external database commands
ASEEXPORT	Exports link information for selected objects
ASELINKS	Manages links between objects and an external database
ASEROWS	Displays and edits table data and creates links and selection sets
ASESELECT	Creates a selection set from rows linked to textual selection sets and graphic selection sets
ASESQLED	Executes Structured Query Language (SQL) statements
ATTREDEF	Redefines a block and updates associated attributes
AUDIT	Evaluates the integrity of a drawing
BACKGROUND	Sets up the background for your scene
BLOCKICON	Generates preview images for blocks created with Release 14 or earlier
BROWSER	Launches the default Web browser defined in your system registry

'CAL	Evaluates mathematical and geometric expressions
CAMERA	Sets a different camera and target location
CLOSE	Closes the current drawing
CONVERT	Converts 2D polylines and associative hatches to the optimized Release 14 format
COPYBASE	Copies objects with a specified base point
COPYHIST	Copies the text in the command line history to the Clipboard
COPYLINK	Copies the current view to the Clipboard for linking to other OLE applications
CYLINDER	Creates a three-dimensional solid cylinder
DBCCLOSE	Closes the dbConnect Manager
DBCONNECT	Provides an AutoCAD interface to external database tables
DBLIST	Lists database information for each object in the drawing
'DDGRIPS	Turns on grips and sets their color
DDMODIFY	Controls properties of existing objects
'DDPTYPE	Specifies the display mode and size of point objects
DDUCS	Manages defined user coordinate systems
DDUCSP	Selects a preset user coordinate system
'DELAY	Provides a timed pause within a script
DIM/DIM1	Both Access Dimensioning mode
DIMSTYLE	Creates and modifies dimension styles on the command line
DSETTINGS	Specifies settings for Snap mode, grid, and polar and object snap tracking
DSVIEWER	Opens the Aerial View window
DWFOUT	Exports a drawing web format file
DWGPROPS	Sets and displays the properties of the current drawing
DXBIN	Imports specially coded binary files
EXPRESSTOOLS	Activates the installed AutoCAD Express Tools if currently unavailable
'FILTER	Creates lists to select objects based on properties
FIND	Finds, replaces, selects, or zooms to specified text
FOG	Provides visual cues for the apparent distance of objects
'GRAPHSCR	Switches from the text window to the drawing area
HYPERLINK	Attaches a hyperlink to a graphical object or modifies an existing hyperlink
HYPERLINKOPTIO NS	Controls the visibility of the hyperlink cursor and the display of hyperlink tooltips
IMAGE	Inserts images in many formats into an AutoCAD drawing file
IMAGEADJUST	Controls the brightness, contrast and fade values of the selected image
IMAGEATTACH	Attaches a new image object and definition
IMAGECLIP	Creates new clipping boundaries for single image objects
IMAGEFRAME	Controls whether the image frame is displayed on the screen or hidden from view
INSERTOBJ	Inserts a linked or embedded object
INTERFERE	Finds the interference of two or more three-dimensional solids and creates a composite 3D solid from their common volume
'ISOPLANE	Specifies the current isometric plane
LAYOUT	Creates a new layout and renames, copies, saves, or deletes an existing layout
LAYOUTWIZARD	Starts the Layout wizard, in which you can designate page and plot settings for a new layout
LEADER	Creates a line that connects annotation to a feature
LSEDIT	Lets you edit a landscape object
LSLIB	Lets you maintain libraries of landscape objects
LSNEW	Lets you add realistic landscape items, such as trees and bushes, to your drawings
LWEIGHT	Sets the current lineweight, lineweight display options, and lineweight units
MASSPROP	Calculates and displays the mass properties of regions or solids
MATLIB	Imports and exports materials to and from a library of materials
MENUUNLOAD	Unloads partial menu files
MLEDIT	Edits multiple parallel lines
MLINE	Creates multiple parallel lines
MLSTYLE	Defines a style for multiple parallel lines
MODEL	Switches from a layout tab to the Model tab and makes it current

MRMRBLOCK	Creates a block definition from selected objects
MULTIPLE	Repeats the next command until canceled
MVSETUP	Sets up the specifications of a drawing
OLELINKS	Updates, changes, and cancels existing OLE links
OLESCALE	Displays the OLE Properties dialog box
PAGESETUP	Specifies the layout page, plotting device, paper size, and settings for each new layout
PARTIALLOAD	Loads additional geometry into a partially opened drawing
PARTIALOPEN	Loads geometry from a selected view or layer into a drawing
PASTEBLOCK	Pastes a copied block into a new drawing
PASTEORIG	Pastes a copied object in a new drawing using the coordinates from the original drawing
PASTESPEC	Inserts data from the Clipboard and controls the format of the data
PCINWIZARD	Displays a wizard to import PCP and PC2 configuration file plot settings into the Model tab or current layout
PLOTSTYLE	Sets the current plot style for new objects, or the assigned plot style for selected objects
PLOTTERMANAGER	Displays the Plotter Manager, where you can launch the Add-a-Plotter wizard and the Plotter Configuration Editor
PREVIEW	Shows how the drawing will look when it is printed or plotted
PROPERTIES	Controls properties of existing objects
PROPERTIESCLOSE	Closes the Properties window
PSDRAG	Controls the appearance of a PostScript image as it is dragged into position with PSIN
PSETUPIN	Imports a user-defined page setup into a new drawing layout
PSFILL	Fills a two-dimensional polyline outline with a PostScript pattern
PSIN	Imports a PostScript file
PSOUT	Creates an Encapsulated PostScript file
QDIM	Quickly creates a dimension
QLEADER	Quickly creates a leader and leader annotation
QSELECT	Quickly creates selection sets based on filtering criteria
RAY	Creates a semi-infinite line
REDRAWALL	Refreshes the display in all viewports
REFCLOSE	Saves back or discards changes made during in-place editing of a reference (an xref or a block)
REFEDIT	Selects a reference for editing
REFSET	Adds or removes objects from a working set during in-place editing of a reference (an xref or a block)
REINIT	Reinitializes the digitizer, digitizer input/output port, and program parameters file
RENDSCR	Redisplays the last rendering created with the RENDER command
REPLAY	Displays a BMP, TGA, or TIFF image
RESUME	Continues an interrupted script
REVOLVE	Creates a solid by revolving a two-dimensional object about an axis
RMAT	Manages rendering materials
RSCRIPT	Creates a script that repeats continuously
SAVEIMG	Saves a rendered image to a file
SCENE	Manages scenes in model space
SECTION	Uses the intersection of a plane and solids to create a region
SETUV	Lets you map materials onto geometry
SHADEMODE	Shades the objects in the current viewport
SHAPE	Inserts a shape
SHELL	Accesses operating system commands
SHOWMAT	Lists the material type and attachment method for a selected object
SKETCH	Creates a series of freehand line segments
SLICE	Slices a set of solids with a plane

SOLDRAW	Generate profiles and sections in viewports created with SOLVIEW
SOLIDEDIT	Edits faces and edges of 3D solid objects
SOLPROF	Creates profile images of three-dimensional solids
SOLVIEW	Creates floating viewports using orthographic projection to lay out multi- and sectional view drawings of 3D solid and body objects
'SPELL	Checks spelling in a drawing
STATS	Displays rendering statistics
'STATUS	Displays drawing statistics, modes, and extents
STLOUT	Stores a solid in an ASCII or binary file
STYLESMANAGER	Displays the Plot Style Manager
SUBTRACT	Creates a composite region or solid by subtraction
SYSWINDOWS	Arranges windows
'TIME	Displays the date and time statistics of a drawing
TOLERANCE	Creates isometric tolerances
TRACE	Creates solid lines
TRANSPARENCY	Controls whether background pixels in an image are transparent or opaque
'TREESTAT	Displays information about the drawing's current spatial index
UCSMAN	Manages defined user coordinate systems
UNION	Creates a composite region or solid
VBAIDE	Displays the Visual Basic Editor
VBALOAD	Loads a global VBA project into the current AutoCAD session
VBAMAN	Loads, unloads, saves, creates, embeds, and extracts VBA projects
VBARUN	Runs a VBA macro
VBASTMT	Executes a VBA statement on the AutoCAD command line
VBAUNLOAD	Unloads a global VBA project
VLISP	Displays the Visual LISP interactive development environment (IDE)
VPCLIP	Clips viewport objects
WHOHAS	Displays ownership information for opened drawing files
WMFIN	Imports a Windows metafile
WMFOPTS	Sets options for WMFIN
XCLIP	Defines an xref clipping boundary and sets the front or back clipping planes
XLINE	Creates an infinite line

#### System Variable Compatibility Chart

The following AutoCAD Sysvars have direct equivalents in AllenCad:

<b>AutoCAD 2000 Sysvars</b>	<b>AllenCad Sysvars</b>	<b>Global / Local</b>	<b>Function</b>
AFLAGS	AFLAGS	(L)	Sets attribute flags for ATTDEF bit-code
ANGBASE	ANGBASE	(L)	Sets the base angle 0 with respect to the current UCS
ANGDIR	ANGDIR	(L)	Sets the positive angle direction from angle 0 with respect to the current UCS
AREA	AREA	(G)	Stores the last area computed by AREA, LIST, or DBLIST
ATTDIA	ATTDIA	(L)	Controls whether -INSERT uses a dialog box for attribute value entry
ATTMODE	ATTMODE	(L)	Controls display of attributes
ATTREQ	ATTREQ	(L)	Determines whether INSERT uses default attribute settings during insertion of blocks
AUNITS	AUNITS	(L)	Sets units for angles
AUPREC	AUPREC	(L)	Sets the number of decimal places for angular units

BACKZ	BACKZ	(L)	Stores the back clipping plane offset from the target plane for the current viewport
BLIPMODE	BLIPMODE	(L)	Controls whether marker blips are visible
CECOLOR	CECOLOR	(L)	Sets the color of new objects
CELTSCALE	CELTSCALE	(L)	Sets the current object linetype scaling factor
CELTYPE	CELTYPE	(L)	Sets the linetype of new objects
CHAMFERA	CHAMFERA	(L)	Sets the first chamfer distance
CHAMFERB	CHAMFERB	(L)	Sets the second chamfer distance
CHAMFERC	CHAMFERC	(L)	Sets the chamfer length
CHAMFERD	CHAMFERD	(L)	Sets the chamfer angle
CHAMMODE	CHAMMODE	(L)	Sets the input method by which The program creates chamfers
CIRCLERAD	CIRCLERAD	(G)	Sets the default circle radius
CLAYER	CLAYER	(L)	Sets the current layer
CMDACTIVE	CMDACTIVE	(G)	Stores the bit-code that indicates whether an ordinary command, transparent command, script, or dialog box is active
CMDECHO	CMDECHO	(G)	Controls whether The program echoes prompts and input during the LISP (command) function
CMDNAMES	CMDNAMES	(L)	Displays the names of the active and transparent commands
CURSORSIZE	CURSORSIZE	(G)	Determines the size of the crosshairs as a percentage of the screen size
CVPORT	CVPORT	(L)	Sets the identification number of the current viewport
DATE	DATE	(G)	Stores the current date and time
DBMOD	DBMOD	(L)	Indicates the drawing modification status using bit-code
DIASSTAT	DIASSTAT	(G)	Stores the exit method of the most recently used dialog box
DIMALT	DIMALT	(L)	Controls the display of alternate units in dimensions
DIMALTD	DIMALTD	(L)	Controls the number of decimal places in alternate units
DIMALTF	DIMALTF	(L)	Controls the multiplier for alternate units
DIMAPOST	DIMAPOST	(L)	Specifies a text prefix or suffix (or both) to the alternate dimension measurement for all types of dimensions except angular
DIMASO	DIMASO	(L)	Controls the "associativity" of dimension objects
DIMASZ	DIMASZ	(L)	Controls the size of dimension line and leader line arrowheads
DIMBLK	DIMBLK	(L)	Sets the arrowhead block displayed at the ends of dimension lines or leader lines
DIMBLK1	DIMBLK1	(L)	Sets the arrowhead for the first end of the dimension line when DIMSAH is on
DIMBLK2	DIMBLK2	(L)	Sets the arrowhead for the second end of the dimension line when DIMSAH is on
DIMCEN	DIMCEN	(L)	Controls drawing of circle or arc center marks and centerlines by DIMCENTER, DIMDIAMETER, and DIMRADIUS
DIMCLRD	DIMCLRD	(L)	Assigns colors to dimension lines, arrowheads, and dimension leader lines
DIMCLRE	DIMCLRE	(L)	Assigns colors to dimension extension lines
DIMCLRT	DIMCLRT	(L)	Assigns colors to dimension text
DIMDLE	DIMDLE	(L)	Sets the distance the dimension line extends beyond the extension line when oblique strokes are drawn

			instead of arrowheads
DIMDLI	DIMDLI	(L)	Controls the spacing of dimension lines in baseline dimensions
DIMEXE	DIMEXE	(L)	Specifies how far to extend the extension line beyond the dimension line
DIMEXO	DIMEXO	(L)	Specifies how far extension lines are offset from origin points
DIMGAP	DIMGAP	(L)	Sets the distance around the dimension text when the dimension line breaks to accommodate dimension text
DIMLFAC	DIMLFAC	(L)	Sets a scale factor for linear dimension measurements
DIMLIM	DIMLIM	(L)	Generates dimension limits as the default text
DIMPOST	DIMPOST	(L)	Specifies a text prefix or suffix (or both) to the dimension measurement
DIMRND	DIMRND	(L)	Rounds all dimensioning distances to the specified value
DIMSAH	DIMSAH	(L)	Controls the display of dimension line arrowhead blocks
DIMSCALE	DIMSCALE	(L)	Sets the overall scale factor applied to dimensioning variables that specify sizes, distances, or offsets
DIMSE1	DIMSE1	(L)	Suppresses display of the first extension line
DIMSE2	DIMSE2	(L)	Suppresses display of the second extension line
DIMSHO	DIMSHO	(L)	Controls redefinition of dimension objects while dragging
DIMSOXD	DIMSOXD	(L)	Suppresses drawing of dimension lines outside the extension lines
DIMSTYLE	DIMSTYLE	(L)	Shows the current dimension style
DIMTAD	DIMTAD	(L)	Controls the vertical position of text in relation to the dimension line
DIMTFAC	DIMTFAC	(L)	Sets a scale factor used to calculate the height of text for dimension fractions and tolerances
DIMTIH	DIMTIH	(L)	Controls the position of dimension text inside the extension lines for all dimension types except ordinate
DIMTIX	DIMTIX	(L)	Draws text between extension lines
DIMTM	DIMTM	(L)	When DIMITOL or DIMLIM is on, sets the minimum (or lower) tolerance limit for dimension text
DIMTOFL	DIMTOFL	(L)	Controls whether a dimension line is drawn between the extension lines even when the text is placed outside
DIMTOH	DIMTOH	(L)	Controls the position of dimension text outside the extension lines
DIMTOL	DIMTOL	(L)	Appends tolerances to dimension text
DIMTP	DIMTP	(L)	When DIMITOL or DIMLIM is on, sets the maximum (or upper) tolerance limit for dimension text
DIMTSZ	DIMTSZ	(L)	Specifies the size of oblique strokes drawn instead of arrowheads for linear, radius, and diameter dimensioning
DIMTVP	DIMTVP	(L)	Controls the vertical position of dimension text above or below the dimension line
DIMTXT	DIMTXT	(L)	Specifies the height of dimension text, unless the current text style has a fixed height

DIMZIN	DIMZIN	(L)	Controls the suppression of zeroes in the primary unit value
DISTANCE	DISTANCE	(G)	Stores the distance computed by DIST
DRAGMODE	DRAGMODE	(L)	Controls the display of objects being dragged
DWGNAME	DWGNAME	(L)	Stores the drawing name as entered by the user
DWGPREFIX	DWGPREFIX	(L)	Stores the drive/directory prefix for the drawing
ELEVATION	ELEVATION	(L)	Stores the current elevation relative to the current UCS for the current viewport in the current space
EXPERT	EXPERT	(G)	Controls whether certain prompts are issued
FILEDIA	FILEDIA	(G)	Suppresses display of the file dialog boxes
FILLETRAD	FILLETRAD	(L)	Stores the current fillet radius
FILLMODE	FILLMODE	(L)	Specifies whether multilines, traces, solids, all hatches (including solid-fill), and wide polylines are filled in
FONTALT	FONTALT	(G)	Specifies the alternate font to be used when the specified font file cannot be located
FRONTZ	FRONTZ	(L)	Stores the front clipping plane offset from the target plane for the current viewport
GRIDMODE	GRIDMODE	(L)	Specifies whether the grid is turned on or off
GRIDUNIT	GRIDUNIT	(L)	Specifies the grid spacing (X and Y) for the current viewport
HIGHLIGHT	HIGHLIGHT	(G)	Controls object highlighting; does not affect objects selected with grips
HPANG	HPANG	(L)	Specifies the hatch pattern angle
HPDOUBLE	HPDOUBLE	(L)	Specifies hatch pattern doubling for user-defined patterns
HPNAME	HPNAME	(L)	Sets a default hatch pattern name
HPSCALE	HPSCALE	(L)	Specifies the hatch pattern scale factor
HPSPACE	HPSPACE	(L)	Specifies the hatch pattern line spacing for user-defined simple patterns
INSBASE	INSBASE	(L)	Stores insertion base point set by BASE
INSNAME	INSNAME	(L)	Sets a default block name for INSERT
LASTPOINT	LASTPOINT	(L)	Stores the last point entered
LENSLENGTH	LENSLENGTH	(L)	Stores the length of the lens (in millimeters) used in perspective viewing for the current viewport
LIMCHECK	LIMCHECK	(L)	Controls the creation of objects outside the drawing limits
LIMMAX	LIMMAX	(L)	Stores the upper-right drawing limits for the current space
LIMMIN	LIMMIN	(L)	Stores the lower-left drawing limits for the current space
LOGFILEMODE	LOGFILEMODE	(G)	Specifies whether the contents of the text window are written to a log file
LOGFILENAME	LOGFILENAME	(G)	Specifies the path and name of the log file
LTSCALE	LTSCALE	(L)	Sets the global linetype scale factor
LUNITS	LUNITS	(L)	Sets linear units
LUPREC	LUPREC	(L)	Sets the number of decimal places displayed for linear units
MIRRTEXT	MIRRTEXT	(L)	Controls how MIRROR reflects text
OFFSETDIST	OFFSETDIST	(G)	Sets the default offset distance
ORTHOMODE	ORTHOMODE	(L)	Constrains cursor movement to the perpendicular
OSMODE	OSMODE	(L)	Sets running object snap modes using bitcodes
PDMODE	PDMODE	(L)	Controls how point objects are displayed
PDSIZE	PDSIZE	(L)	Sets the display size for point objects
PERIMETER	PERIMETER	(G)	Stores the last perimeter value computed by AREA, LIST, or DBLIST

PICKFIRST	PICKFIRST	(G)	Controls whether you select objects before or after you issue a command
PICKSTYLE	PICKSTYLE	(L)	Controls the use of group selection and associative hatch selection
PLATFORM	PLATFORM	(G)	Indicates which platform of the program is in use
PLINETYPE	PLINETYPE	(G)	Specifies whether the program uses optimized 2D polylines
PLINEWID	PLINEWID	(L)	Stores the default polyline width
POLYSIDES	POLYSIDES	(G)	Sets the default number of sides for POLYGON
PSLTSCALE	PSLTSCALE	(L)	Controls paper space linetype scaling
QTEXTMODE	QTEXTMODE	(L)	Controls how text is displayed
REGENMODE	REGENMODE	(L)	Controls automatic regeneration of the drawing
SAVETIME	SAVETIME	(G)	Sets the automatic save interval, in minutes
SCREENMODE	SCREENMODE	(G)	Stores a bitcode indicating the graphics/text state of the The program display
SCREENSIZE	SCREENSIZE	(G)	Stores current viewport size in pixels (X and Y)
SNAPANG	SNAPANG	(L)	Sets snap and grid rotation angle for the current viewport
SNAPBASE	SNAPBASE	(L)	Sets the snap and grid origin point for the current viewport relative to the current UCS
SNAPISOPAIR	SNAPISOPAIR	(L)	Controls the isometric plane for the current viewport
SNAPMODE	SNAPMODE	(L)	Turns Snap mode on and off
SNAPSTYL	SNAPSTYL	(L)	Sets snap style for the current viewport
SNAPUNIT	SNAPUNIT	(L)	Sets the snap spacing for the current viewport
SPLFRAME	SPLFRAME	(L)	Controls the display of splines and spline-fit polylines
SURFTAB1	SURFTAB1	(L)	Sets the number of tabulations to be generated for RULESURF and TABSURF
SURFTAB2	SURFTAB2	(L)	Sets the mesh density in the N direction for REVSURF and EDGESURF
SURFU	SURFU	(L)	Sets the surface density for PEDIT Smooth in the M direction
SURFV	SURFV	(L)	Sets the surface density for PEDIT Smooth in the N direction
TEXTSIZE	TEXTSIZE	(L)	Sets the default height for new text objects drawn with the current text style
TEXTSTYLE	TEXTSTYLE	(L)	Sets the name of the current text style
THICKNESS	THICKNESS	(L)	Sets the current 3D solid thickness
TILEMODE	TILEMODE	(L)	Makes the Model tab or the last layout tab current
TRACEWID	TRACEWID	(L)	Sets the default trace width
TRIMMODE	TRIMMODE	(L)	Controls whether The program trims selected edges for chamfers and fillets
UCSFOLLOW	UCSFOLLOW	(L)	Generates a plan view whenever you change from one UCS to another
UCSICON	UCSICON	(L)	Displays the UCS icon for the current viewport
UCSNAME	UCSNAME	(L)	Stores the name of the current coordinate system for the current viewport in the current space
UCSORG	UCSORG	(L)	Stores the origin point of the current coordinate system for the current viewport in the current space
UCSXDIR	UCSXDIR	(L)	Stores the X direction of the current UCS for the current viewport in the current space
UCSYDIR	UCSYDIR	(L)	Stores the Y direction of the current UCS for the current viewport in the current space
UNDOCTL	UNDOCTL	(G)	Stores a bitcode indicating the state of the Auto and Control options of the UNDO command



UNITMODE	UNITMODE	(L)	Controls the display format for units
USERI1-5	USERI1-5	(L)	Stores and retrieves integer values
USERR1-5	USERR1-5	(L)	Stores and retrieves real numbers
USERS1-5	USERS1-5	(L)	Stores and retrieves text string data
VIEWCTR	VIEWCTR	(L)	Stores the center of view in the current viewport
VIEWDIR	VIEWDIR	(L)	Stores the viewing direction in the current viewport
VIEWSIZE	VIEWSIZE	(L)	Stores the height of the view in the current viewport
VIEWTWIST	VIEWTWIST	(L)	Stores the view twist angle for the current viewport
VISRETAIN	VISRETAIN	(L)	Controls the visibility, color, linetype, linewidth, and plot styles (if PSTYLEPOLICY is set to 0) of xref-dependent layers and specifies whether nested xref path changes are saved
WORLDUCS	WORLDUCS	(L)	Indicates whether the UCS is the same as the WCS

The Title Bar

The Left Sidebar

The Menu Bar

The Command Window

The Status Bar

The Properties Bar

The Drawing (Edit) Window

The Cursor

The Coordinate Symbol

The Horizontal Toolbar

Roundtrip R2000 Drawings and DXF Files

Presently, AllenCad does not support a limited number of AutoCAD drawing elements including ACIS™ solids objects. However, these elements can be retained (but will not be displayed) by using the "Preserve unsupported AutoCAD entities" switch when opening existing AutoCAD DWG files.

Note that while being edited in AllenCad, unsupported AutoCAD 2000 entities remain unchanged and invisible the background. If the drawing is saved in AutoCAD 2000 drawing format again, all entities including these unsupported AutoCAD 2000 entities will be written back to the file.

Copy Selection to Clipboard



**Command: COPYSELECT**

**Alias CST**

**Located:**

- 1. Pull-down Menu: Edit – Cut Selection to Clipboard**
- 2. Top Function Bar**
- 3. Or Enter Command or Alias at the Command Line**

The command COPYSELECT copies selected objects from the drawing, placing them onto a AllenCad-specific clipboard. The selected objects remain in the drawing.

The command COPYSELECT is useful especially to copy objects from one drawing to other drawings. COPYSELECT places these objects onto the clipboard. Use the PASTE command to paste them from the clipboard into other drawings.

**To copy objects onto the clipboard:**

1. First specify the base point for later insertion of the objects.
2. Then select the objects to be copied to the clipboard.

**Command line example:**

```
> COPYSELECT  
Insertion base point:  
Select objects:
```

**Note:**

Do not confuse this command with the COPY command found in the *Modify* menu that makes copies of objects in the current drawing. The objects selected with COPYSELECT remain in the clipboard, ready to be placed in another drawing. They will remain on the clipboard until you place new objects on it with the COPYSELECT command.

---

**See also:**

Command PASTE

Command DGNIN

**Command: DGNIN****Alias DGI****Located:**

1. **Pull-down Menu: Tools – DGN Import**
2. **Or Enter Command or Alias at the Command Line**

This command is used to import Microstation DGN format drawing files into AllenCad. When you issue the command, the file is imported directly, without any prompts for insertion point, scale or rotation. If you require these capabilities, it is recommended that you import the DGN file into a new and empty drawing and then save it as an external part. Afterwards, you will be offered the opportunity to insert, scale and rotate to meet your requirements.

Command DGNOUT

**Command: DGNOUT****Alias DGO****Located:**

1. **Pull-down Menu: Tools – DGN Export**
2. **Or Enter Command or Alias at the Command Line**

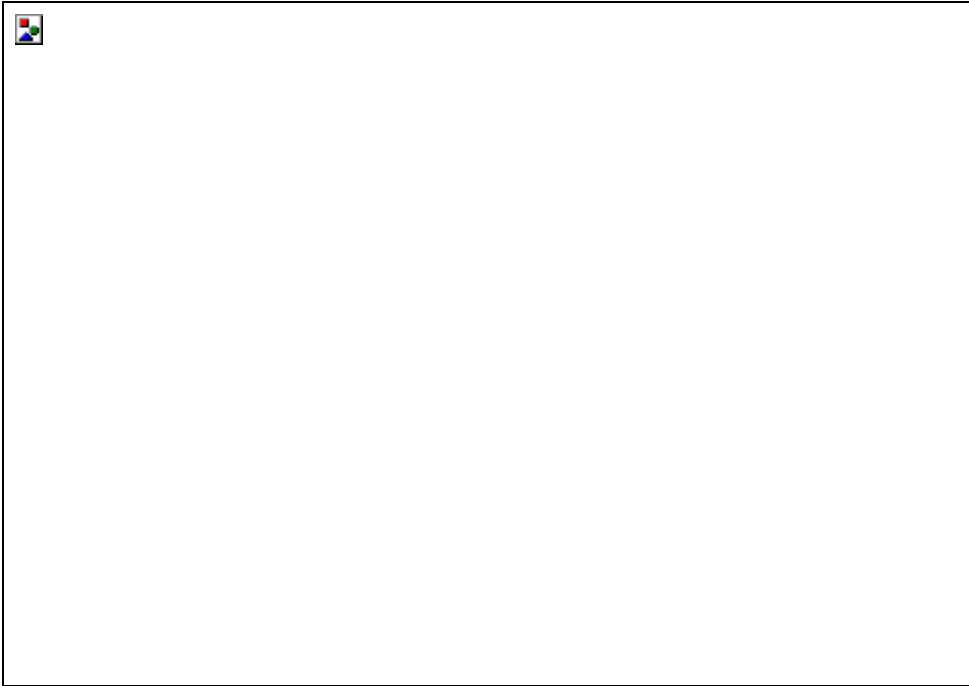
This command is used to export Microstation DGN format drawing files directly from AllenCad. When you issue the command, you are asked for a filename to assign to the new drawing file, then the program proceeds to save the current drawing in this format.

DIMLIM

0 = OFF (default) | 1 = ON

Determines that the string values of the variables DIMTP and DIMTM are used as dimension label to create limit dimensioning. When DIMLIM is set to 1 (on), DIMITOL is set to 0 (off).

The Desktop - Opening and Saving Drawings

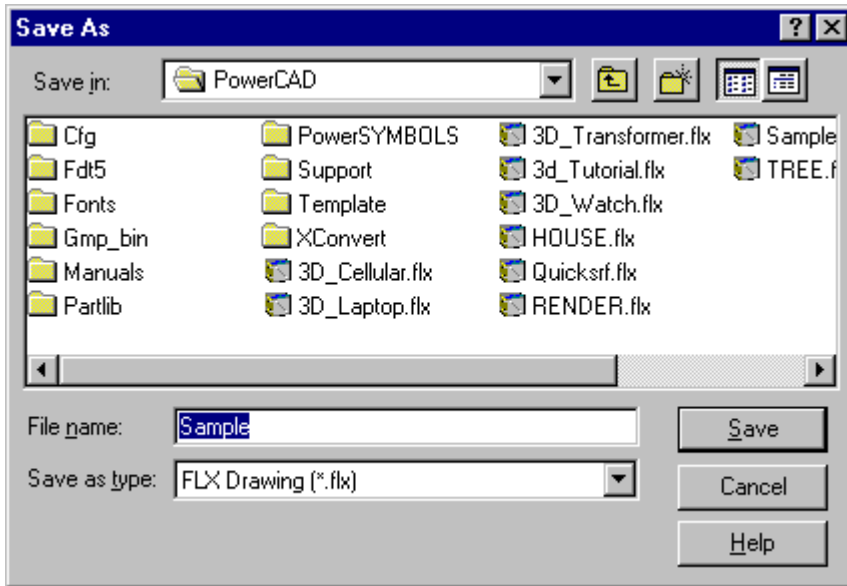


This short introduction will help you become familiar with the AllenCad Desktop pictured above. This is what you see when you first start the program. Click on any area of the screen image to see an explanation of what it is:

On The Desktop, you will see that AllenCad uses standard pull-down menus on the Menu Bar and standard tool and text bars. In addition, command icons are located on the vertical tool bar on the left side of the desktop. The area colored dark gray is the drawing edit area where you will construct and edit your drawing. **This area will remain gray and drawing commands will remain unavailable until either a new or existing drawing file has been opened.** Below the edit area, you will see the Command prompt area. Here, you may type in commands for execution.

To open a drawing file you will click in the **File** pull-down menu and click on either **New** to begin a new drawing or **Open** to open an existing drawing.

The commands for saving a drawing are also located on the **File** pull-down menu. To simply save your drawing file in the default location using the filename and file type that you specified when creating the new file, you simply click on **Save**. If you wish to change any of these parameters, you click on **Save As**. This presents you with the following dialog box.



Here, you may change the directory to store the file in, rename the file, or elect to save the file as and AutoCAD format .dwg or .dxf drawing, instead of the native AllenCad format.

Another related command on the File pull-down menu is **"SaveAll"**. You will use this command (in a similar manner to **"Save"**), when you have multiple drawings open and you wish to save them all in a single operation.

### Command Entry

AllenCad has been designed to be flexible and versatile, enabling it to meet the needs of users who might wish to work in different ways. In all, AllenCad offers four different ways of entering commands:

1. Through pull-down menus
2. Selecting command icons, located on toolbars and palettes
3. "Alias" or direct command entry at the command line.
4. Automated command entry through the use of LISP programs and Macros (Scripts)

Many new users prefer to use the pull-down menus to enter their commands and may later switch to using command icons or typing their commands, once they become more familiar with the program.

Some command icons are visible on the desktop toolbars. Additional toolbars or palettes containing other related command icons can also be placed on the desktop as needed. Each icon represents an individual drawing command. If you place your cursor over any icon, you will see a pop-up "ToolTip" that identifies the related drawing command.

### Command Options

While you are drawing, AllenCad monitors the commands you are using and tries to anticipate your next moves. In doing this, the program offers you a selection of available options for each command.

How these options are presented is a matter of personal preference and how you have configured your desktop. There are three options for Option display:

1. Experienced operators who are very familiar with the commands and their options may elect to turn option display off, altogether and to indicate their choice of option, using typed in option aliases.
2. Options may be displayed in a dialog box that appears next to the cursor, to be available as you edit your drawing and
3. Options may be presented on an option bar that the program inserts below the horizontal toolbar whenever this display mode is selected.

Simply pick the option you want with a left-mouse click. Alternatively, You can specify the option by entering the appropriate option input value at the command line.

### Transparent Commands

Transparent commands are those that can be used while other commands are currently active. For instance, you can interrupt your drawing of a line to issue a **Zoom** or **Pan** command to change your view of the drawing. You will be switched back to completing your line, after the transparent command is completed. Please note that the **UNDO** command does not operate with transparent commands, but that other commands such as **ZOOMBACK** must be used to reverse commands.

To use these commands transparently, simply place an apostrophe before the command IE. 'DPAN, 'ID, etc.

### Using Palettes



- The Edit palette

Palettes are simply collections of associated commands that are made available as command icons on palette toolbars. They can be called when needed from the individual palette icons on the horizontal toolbar or from selections in the pull-down menus. They can be dragged and dropped in suitable locations on the drawing board, moved around as you work and closed when the commands are no longer required for use.



### Aligned Dimensioning



Aligned Dimensioning measures and labels the absolute distance between two points, regardless of the relative position of the axes. This makes it possible to dimension lines, edges and spaces etc. that are not located on the principal axes of the coordinate system (IE. not parallel to x, y or z-axes).

The user will be asked to define two points of measurement. Once the points of measurement have been chosen, the user will be asked to define a position for the dimensioning line. This can be done either by entering the relevant coordinates, or with the cursor. The absolute measurement between the two points will be displayed. Pressing the RETURN key will accept the calculated value. The dimension will be inserted into the drawing and shown on a horizontal plain.

IE.

Pick 1st point: P1

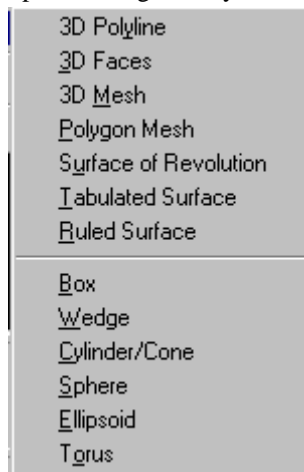
Pick 2nd point: P2

Select dimension line positioning: P3

Verify or modify dimension text <3.05>: RETURN

### 3D Commands

Selecting the 3D option on the Draw pull-down menu results in a presentation of the following option dialog. Here you can select which 3-dimensional object you wish to draw.

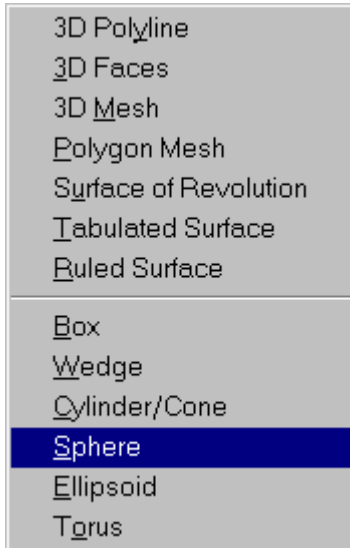


See the following topics for a description of each option. Remember, when you are drawing 3D objects that the length is along the x-axis and width is along the y-axis. Height is along the z-axis

IE into the drawing, so you may have to change your viewpoint with the 3DVIEW command to visualize the height or depth of an object.

### 3D Shapes

AllenCad offers the user the ability to simply draw 3 dimensional wire frame shapes directly from the Draw pull-down menu. When **3D** is selected on this menu, the following option box appears:



The lower section of this box lists the shapes that can be drawn using this option. These can be used in combination, to construct more complex entities that will comprise your 3D model. If solid surfaces are required, 3Dfaces can be subsequently drawn, using the wireframe model as input. The only stipulation here is that each face be in one plane (IE not bent).

Before beginning to draw with these objects, be sure to read and understand the lessons on:

- Layers
- Elevation
- Hide and Fast Hide
- Object Snaps

Understanding and applying these concepts will greatly simplify 3 dimensional drawing.

### Line Definition

A line is an entity that runs directly from a start to an endpoint. When arranged in a continuous fashion, lines may also be referred to as line segments. Each segment is treated by AllenCad as a separate entity

### Polygon Definition



A polygon is defined as a plane figure enclosed by five or more straight lines.

### Polyline Definition

Polylines, whether 3 or 2 dimensional (IE having or not having an assigned elevation) are a series of contiguous (joined end to end) lines or arcs that are treated by AllenCad as a single entity.

### Pull-Down Menu Bar

The Pull-Down Menu Bar contains most of the commands that you will use in creating and managing your drawing file. For more immediate access to these commands, you may choose to use command icons located on toolbars or palettes, or else you may enter the command directly by typing the command name at the command prompt near the bottom of the screen.

### The Command Prompt Area

The Command Prompt Area is another area where the program and the user communicate with each other. Here you may enter commands, options and values directly and this is also where the program posts messages to the user.

### The Drawing Editor

This is the area where you will create and edit your drawing. When you have made a file active, either by selecting from the pull-down menu either File – New or File – Open (for existing drawing files), this area will become active and you will be able to enter drawing commands.

### The Left Sidebar

The Left Side Bar contains commonly used command icons grouped together in clusters. If you find yourself not using these icons and needing more space for displaying your drawing, this sidebar can be switched off using File – Desktop Manager.

### The Management Toolbar

The Management Toolbar makes immediately available, some commonly used file and drawing management command icons that are also available in the Pull-Down Menu.

### The Mode Toolbar

The Mode Toolbar contains icons that toggle on/off Orthogonal Mode, Grid Mode and Snap Mode.

### The Pull-Down Menu Bar

Most of your drawing and file management commands are available to you in the pull-down menus on this toolbar. Alternately, you can initiate a command by clicking the command icon located on the Left Side Bar or typing in the command at the Command Prompt (below).

### The Status Bar

The Status Bar indicates the current user or world coordinate system, as well as cursor location in X,Y,Z coordinate display and relative distance/angle from the coordinate system origin.

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