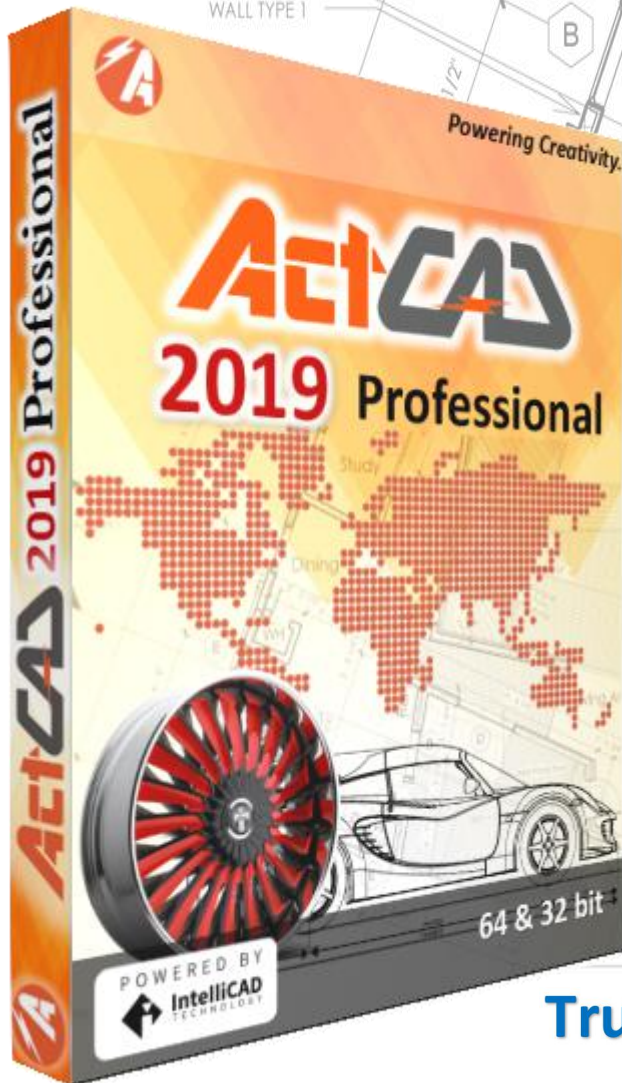


# ActCAD 2019 Quick Start Guide



**ActCAD**  
*Powering Creativity*

**Trusted by over 15000 Customers  
in over 45 Countries.**



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## Chapter 1 - Introduction

### Introduction to CAD

Computer Aided Design (CAD) is the application of computers for the purpose of design and drafting of engineering or non-engineering systems. Sometimes CAD is also referred as Computer Aided Design and Drafting (CADD). CAD is a process of creating or modifying the technical drawings, layouts, 3D models etc. using computers. The output of CAD software is in electronic format in the form of computer files. CAD Software uses either Vector based approach or Raster based approach to create the drawings. Apart from the shape and size, the CAD drawings also provide information on materials, processes, specifications, special instructions, indications etc.

Now-a-days, CAD is an important application widely used in Mechanical, Civil, Architectural, Aerospace, Chemical, Electrical, Railways, Ship Building, technical documentation, animations, renderings and many more fields. The manual drafting methods are almost obsolete due the advantages of CAD software tools.

Initial developments on CAD were carried out in the 1960s within the aircraft and automotive industries in the area of 3D surface construction and NC programming. Some of the mathematical description work on curves was developed in the early 1940s by Robert Issac. However, probably the most important work on polynomial curves and sculptured surface was done by Pierre Bézier (Renault), Paul de Casteljaou (Citroen), Steven Anson Coons (MIT, Ford), James Ferguson (Boeing), Carl de Boor (GM), Birkhoff (GM) and Garibedian (GM) in the 1960s and W. Gordon (GM) and R. Riesenfeld in the 1970s.

### Advantages of CAD Drawings

- ✓ Faster to create and modify the drawings.
- ✓ More accurate to draw and measure compared to manual drawings.
- ✓ Saving copies is much easier and effort will be less to create replicates.
- ✓ Easy to share the drawings in electronic format with others.
- ✓ Space requirement is less in design office as there is no need of bigger drafting machines.
- ✓ CAD software is inexpensive and there are few free versions too.
- ✓ Drawings look more professional in terms of line widths, text, fonts etc.
- ✓ Facilitate the file management using database.
- ✓ Computer files never lost or deteriorate unlike manual paper or sepia drawings.
- ✓ Standard functions and commands help to create special geometries much faster.
- ✓ Zooming can be done easily for enlarged or reduced view scale.
- ✓ Changing the Scale is very simple for the whole drawing, model or views.
- ✓ Drafting skill requirement is less in CAD.
- ✓ Repetitive tasks can be automated in the form of LISP, VBA, ARX programs.
- ✓ Standard elements like Title blocks, symbols (civil, mechanical, electrical, electronics, architectural etc) can be stored in the form of templates, blocks etc.

- ✓ Symbols and source files can be downloaded from internet without the need of developing them from scratch.
- ✓ Integration is possible from other computer documents like Excel, Word, PDF, JPEG, GIF etc.
- ✓ Easy to handle drawings as CAD software supports Layer concept. Unwanted elements in layers can be switched-off when not needed. This is not possible on manual drawings.

### Popular CAD Softwares and Developers

CAD Software	Developer
Unigraphics NX	Siemens Industry Software
I-DEAS	Siemens Industry Software
CATIA	Dassault Systemes
Solid Edge ST	Siemens Industry Software
Solid Woks	Dassault Systemes
Creo Elements	PTC
AutoCAD	AutoDesk
ActCAD	ActCAD.LLC

### What is IntelliCAD?

The IntelliCAD Technology Consortium (ITC) is an organization owned and governed by its members. ITC focuses on the development of the IntelliCAD CAD platform compatible with the DWG file format. ITC members focus on the customization, marketing and sales of CAD solutions based on the IntelliCAD technology. ITC Commercial Members have unlimited distribution rights for delivering the IntelliCAD technology to their customers.

The ITC is a non-profit organization and invests over 70% of revenue back to research and development. The ITC developers are seasoned programmers having expertise in 2D and 3D math and visualization.

### About ActCAD:

ActCAD is a 2D Drafting and 3D Modeling CAD software meant for Engineers, Architects and other technical consultants. ActCAD is a native DWG & DXF CAD software to create and edit drawings. ActCAD is based on the technologies from IntelliCAD Technology Consortium (ITC), ODA Teigha Libraries, ACIS 3D Kernel, Wintopo, ps2edit etc. ActCAD uses the most recently released version of IntelliCAD to offer the best performance and features to its users. ActCAD as many add-on commands and features supplementing core IntelliCAD making ActCAD more productive than its competition. In addition, ActCAD includes many other tools like Block Libraries (in the domains of Civil, Mechanical, Electrical, Architectural, Structural, P &IDs etc.), Batch file converter, Units converter and many other productivity enhancement tools.

ActCAD can open the .dgn file formats and saving them to dwg/dxf file formats. All dwg/dxf versions from R2.5 to 2018 are supported...

ActCAD is a light weight yet very powerful Professional Computer Aided Design (CAD) software utilizing the benefits of multi-threading and multi-core processing capabilities of recent computer systems. The technology is being enhanced by well experienced Developers working from all

over the World, around the clock. The core developers of ActCAD have over 40 years of experience in industry especially on CAD software. There are many other tools available in the markets which do not have an optimum balance between capability and price. Some of the capable tools are very expensive and on the other hand, some of the low cost tools are not capable of handling critical project requirements. ActCAD perfectly bridges this price performance gap offering great value to our customers.

### **Advantages of ActCAD**

- Very Low Price : With Full Features & Functions
- No Special Learning : Familiar interface, commands & Icons
- Best Performance : True power of 64bit & Multi-threading
- Perpetual Licenses : Global Validity & Self-License Transfer
- ActCAD Converter : Batch Auditing & File Conversions
- All DWG & DXF versions : From R2 to 2018
- PDF Export : Both 3D PDF & 2D PDF Export
- Dr.KLN Drawings Converter : PDF/PS to DXF/SVG/PCG Conversion
- Raster to Vector : Image/Scanned Drawings to DXF Conversion
- Express Tools & Add-on Features
- Content Management : Over 5000 Blocks included. Add your own too.
- Many Features & Improvements in addition to IntelliCAD 8.4b Engine.

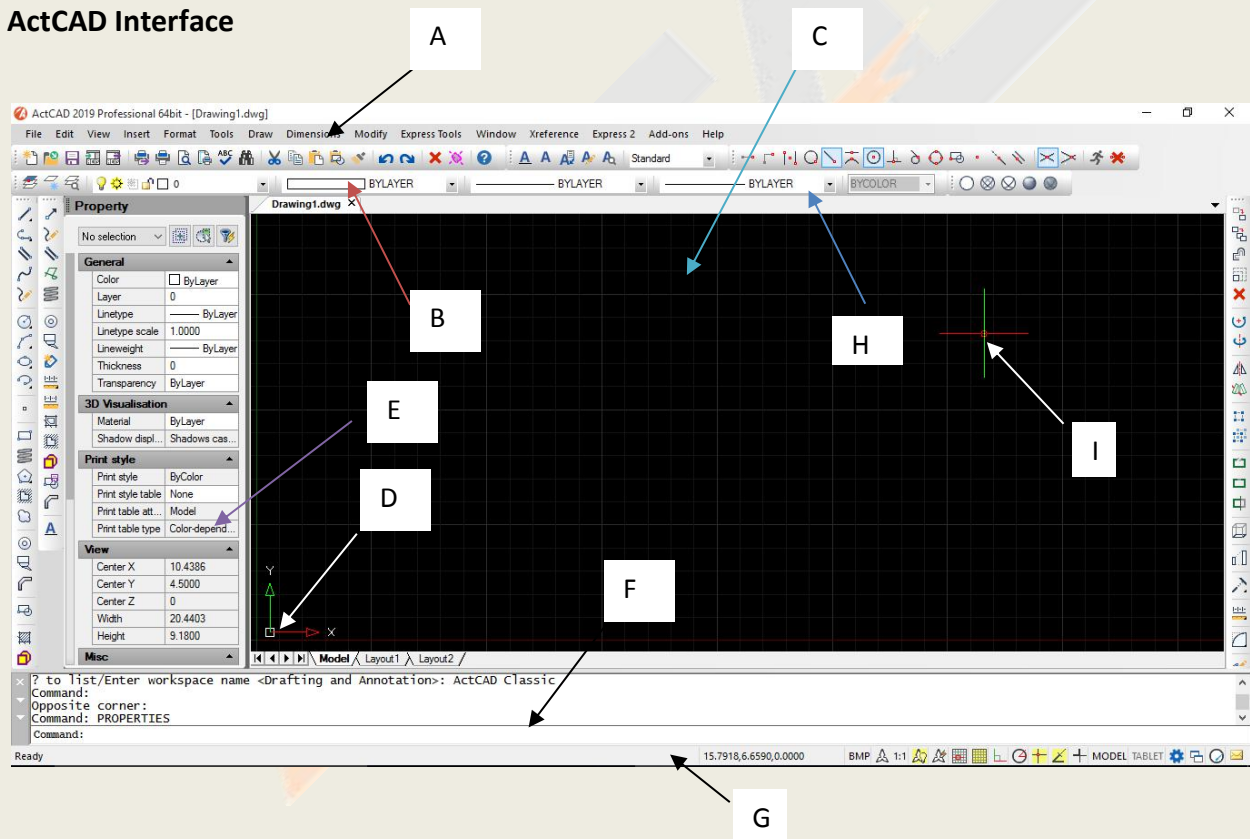
## Chapter 2: Interface Study & Menus

### Starting up ActCAD

Select the ActCAD option on the program menu or select the ActCAD icon on the desktop. Once the program is loaded into memory, the ActCAD drawing screen will appear on the screen.

Note that ActCAD automatically assigns generic names, Drawing X, as new drawings are created. In our example, ActCAD opened the graphics window using the default system units and assigned the drawing name Drawing1.

### ActCAD Interface

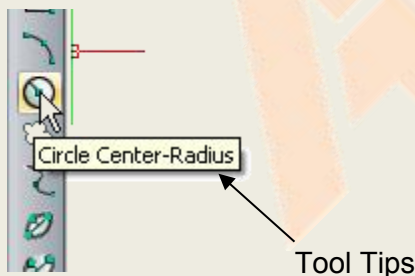


- |                           |  |                                       |
|---------------------------|--|---------------------------------------|
| <b>A:</b> Menu Bar        | <b>D:</b> UCS(User Co-ordinate System) | <b>G:</b> Status Bar                  |
| <b>B:</b> Toolbars        | <b>E:</b> Properties Window            | <b>H:</b> Floating or Docking ToolBar |
| <b>C:</b> Graphics Window | <b>F:</b> Command Bar                  | <b>I:</b> Cross Hair                  |

- A:** Menu bar can be customized at the top of the window
- B:** Toolbars can be customized by changing the appearance and arrangement of tools and adding required commands and macros.
- C:** Drawing window displays the drawings Created. It is the working area for the drawings to be created.
- D:** The user coordinate system (UCS) icon indicates the orientation of the drawing in three-dimensional space. It can be taken as reference point from which the cursor location Co-ordinates will be displayed in the Status bar
- E:** Layout bar contains one model and two layouts will be available which can be increased to a maximum of 256 layouts, Click a tab to switch between the drawing model and a print layout
- F:** Command bar is used to give the inputs to the drawing through typed commands. To reposition the command bar, drag it to another location on the screen.
- G:** The status bar displays information such as the name or purpose of a tool, the current cursor coordinates, layer name, and mode settings.
- H:** Floting or docking Toolbars-toolbars can be moved and docked at any location on the screen as desired.
- I:** The Cross Hair displays the cursor location and the orientation of X, Y axis of drawing.

## Tool Tips

**ToolTips** To find out what a specific tool on a toolbar does, pause the cursor over it for a moment. A ToolTip appears on the screen. with its name

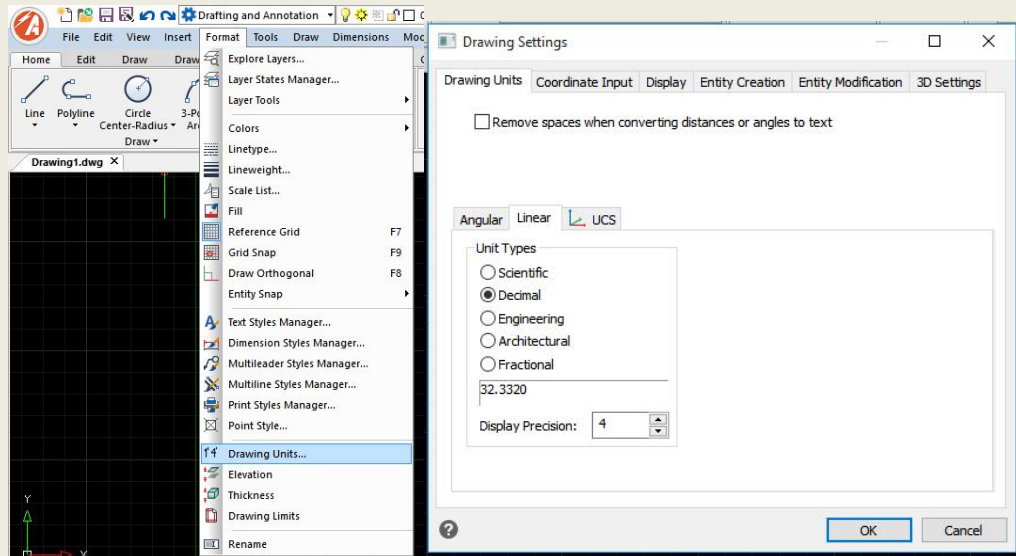


## Drawing Units Setup

Every object we construct in a CAD system is measured in units. We should determine the value of the units within the CAD system before creating the first geometric entities.

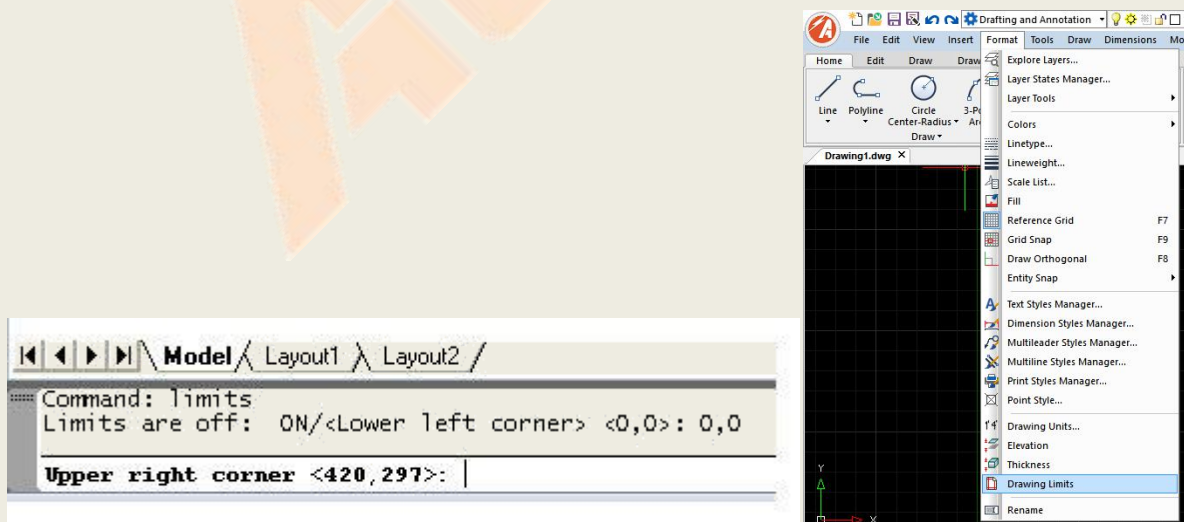
1. In the pull-down menus, select:[Format] [Units]
2. In the Drawing Units dialog box, set the Length Type to Decimal. This will set the measurement to the default Metric Units.

3. Set the Precision to two digits after the decimal point as shown in the above figure.
4. Pick OK to exit the Drawing Units dialog box.



## Drawing Area Setup

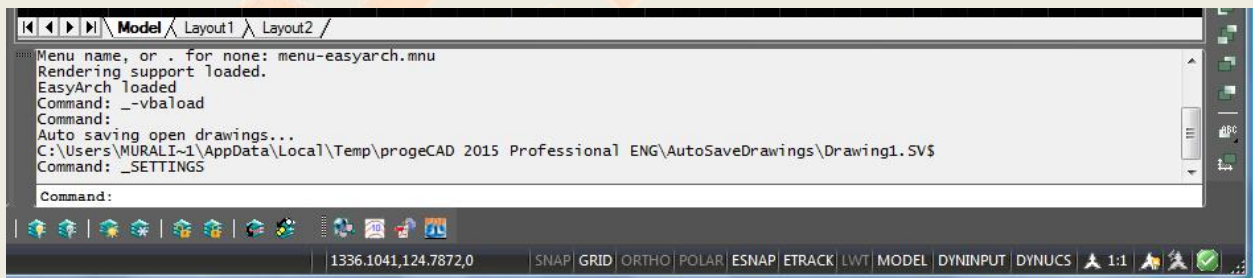
Next, we will set up the Drawing Limits; setting the Drawing Limits controls the extents of the display of the grid. It also serves as a visual reference that marks the working area. It can also be used to prevent construction outside the grid limits and as a plot option that defines an area to be plotted/printed. Note that this setting does not limit the region for geometry construction.



1. In the pull-down menus, select: [Format] [Drawing Limits]
2. In the command prompt area, near the bottom of the proge CAD drawing screen, the message "Reset Model Space Limits: Specify lower left corner or [On/Off]<0.00,0.00>:" is displayed. Press the Enter key once to accept the default coordinates <0.00,0.00>.
3. In the command prompt area, the message "Specify upper right corner <0.00,0.00>:" is displayed. Press the ENTER key once to accept the default coordinates <12.00,9.00>.
4. On your own, move the graphic cursor near the upper-right corner inside the drawing area and note that the drawing area is unchanged. (The Drawing Limits command is used to set the drawing area; but the display will not be adjusted until a display command is used.)
5. Move the graphic cursor near the upper right corner inside the drawing area and note that the display area is updated.

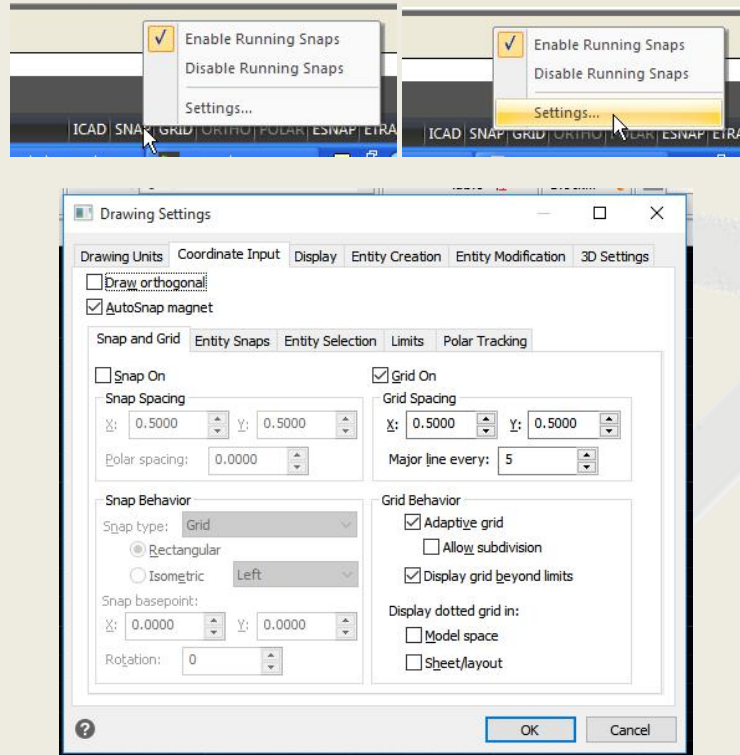
## GRID - ON/OFF:

1. Left-click the GRID button in the Status Bar to turn ON the GRID option. (Notice in the command prompt area, the message "<Grid on>" is also displayed.)
2. Right-click on GRID button in the Status Bar to change setting of GRID like Grid spacing and Grid angel.



**SNAP - ON/OFF:**

1. Left-click the SNAP button in the Status Bar to turn ON the SNAP option.
2. Right-click on SNAP button in status bar to change settings for snap distance.

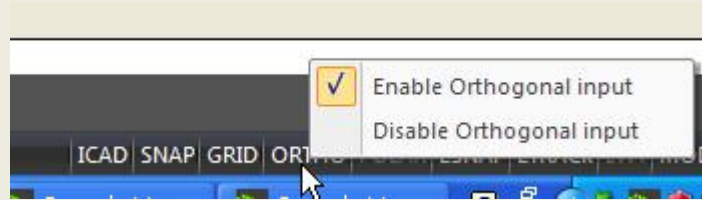
**ORTHO:**

ActCAD provides drawing and editing tools similar to the drafter's T-square. As you create or move objects, you can use Ortho mode to restrict the cursor to the horizontal or vertical axis. The orthogonal alignment depends on the current snap angle, UCS, or isometric grid and snap setting. Ortho works with commands that require you to specify a second point. You can turn Ortho on and off at any time during drawing and editing. You can use Ortho not only to establish vertical or horizontal alignments but also to enforce parallelism or to create regular offsets from existing objects.

By imposing orthogonal constraints, you can draw more quickly. For example, you can create a series of perpendicular lines by turning on Ortho mode before you start. Because the lines are constrained to be parallel to the horizontal and vertical axes, you know that the lines are perpendicular.

As you move the cursor, a rubber-band line that defines the displacement follows the horizontal or vertical axis, depending on which axis is nearest to the cursor. ActCAD overrides Ortho when you enter coordinates on the command line, use perspective views, or specify an object snap.

## To turn on or turn off Ortho mode



- On the status bar, click Ortho.

## POLAR TRACKING:

Polar tracking restricts cursor movement to specified angles. Polar Snap restricts cursor movement to specified increments along a polar angle.

When you are creating or modifying objects, you can use polar tracking to display temporary alignment paths defined by the polar angles you specify. In 3D views, polar tracking additionally provides an alignment path in the up and down directions. In that case, the tooltip displays a +Z or -Z for the angle.

Polar angles are relative to the orientation of the current user coordinate system (UCS) and the setting for the base angle convention in a drawing. The angle base direction is set in the Drawing Units dialog box (UNITS).

### To turn on and turn off polar tracking

1. Press F10, or click the polar button on the status bar.
2. To turn polar tracking on or off temporarily, hold down the F10 key while you work.

Note Turning on Ortho automatically turns off polar tracking.

## ESNAP:

The Entity Snaps (Esnaps for short) are drawing aids which are used in conjunction with other commands to help you draw accurately. Esnaps allow you to *snap* onto a specific object location when you are picking a point. For example, using Esnaps you can accurately pick the end point of a line or the center of a circle. Esnaps in ActCAD are so important that you cannot draw accurately without them. For this reason, you must develop a good understanding of what the Esnaps are and how they work.

There are three basic methods of accessing the Esnaps:

1. The Esnaps are also available on their own Object Snap toolbar. If this toolbar is not already displayed, you can display it using the TOOLBAR command, View ► Toolbars... from the pull-down menu. When the Toolbar dialogue box appears, simply check the box next to "Object Snap" in the toolbars list. Many ActCAD users work with the Object Snap toolbar permanently docked on their screen because it gives one-click access to all of the Esnaps, making drawing much more efficient.

2. You can also access the Esnaps from the cursor menu. Hold the Shift key down on the keyboard and right-click the mouse to bring up the cursor menu. The menu appears at the current cursor position.
3. Finally, you can also access the Esnaps from the keyboard by typing their abbreviated name. See the exercise above and the sections below for details.

### Endpoint :

Toolbar    Object Snap



Pull-down Shift + Right Click Endpoint

Keyboard END (when picking)



The Endpoint Esnap snaps to the end points of lines and arcs and to polyline vertices. This is one of the most useful and commonly used Esnaps.

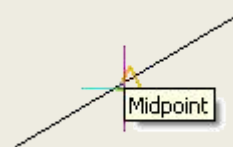
### Midpoint :

Toolbar    Object Snap



Pull-down Shift + Right Click Midpoint

Keyboard MID (when picking)



The Midpoint Esnap snaps to the mid points of lines and arcs and to the mid point of polyline segments.

### Intersection :

Toolbar    Object Snap



Pull-down Shift + Right Click Intersection

Keyboard INT (when picking)



The Intersection Esnap snaps to the physical intersection of any two drawing objects

### Extension :

Toolbar    Object Snap



Pull-down Shift + Right Click Extension

Keyboard EXT or EXTEN (when picking)

The Extension Esnap enables you to snap to some point along the imaginary extension of a line, arc or polyline segment.

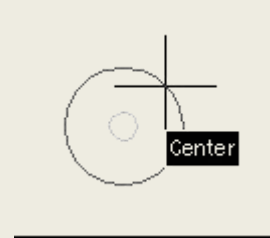
### Center :

Toolbar Object Snap



Pull-down Shift + Right Click Center

Keyboard CEN (when picking)



The Center Osnap snaps to the centre of a circle, arc or polyline arc segment. The cursor must pass over the circumference of the circle or the arc so that the centre can be found.

### Quadrant

Toolbar Object Snap



Pull-down Shift + Right Click Quadrant

Keyboard QUA or QUAD (when picking)

The Quadrant Osnap snaps to one of the four circle quadrant points located at north, south, east and west or 90, 270, 0 and 180 degrees respectively.



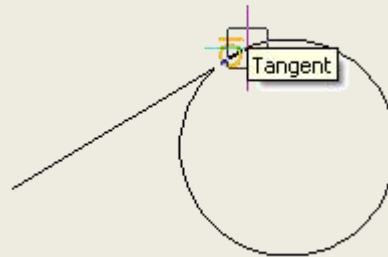
### Tangent :

Toolbar Object Snap



Pull-down Shift + Right Click Tangent

Keyboard TAN (when picking)



The Tangent Osnap snaps to a tangent point on a circle. This osnap works in two ways. You can either draw a line from a point to the tangent point (see illustration) or you can draw a line from a tangent point, the latter is referred to as the "Deferred Tangent" snap mode.

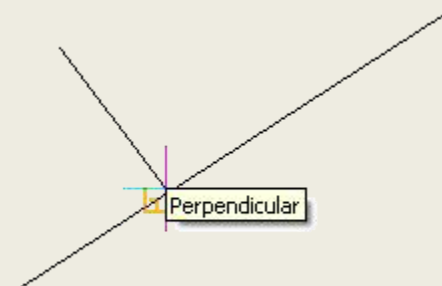
### Perpendicular :

Toolbar Object Snap



Pull-down Shift + Right Click Perpendicular

Keyboard PER or PERP (when picking)



The Perpendicular Osnap snaps to a point which forms a perpendicular with the selected object. As with the Tangent Osnap, Perpendicular can be used to draw a line to a perpendicular point, as in the illustration or from a perpendicular point, known as the "Deferred Perpendicular" snap mode.

parallel. You can now pick the second point of your line. The Snap Tip also includes a relative polar co-ordinate.

## Insert :

Toolbar Object Snap



Pull-down Shift + Right Click Insert

Keyboard INS (when picking)

The Insert Osnap snaps to the insertion point of a block, text or an image.

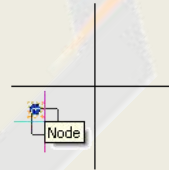
## Node

Toolbar Object Snap



Pull-down Shift + Right Click Node

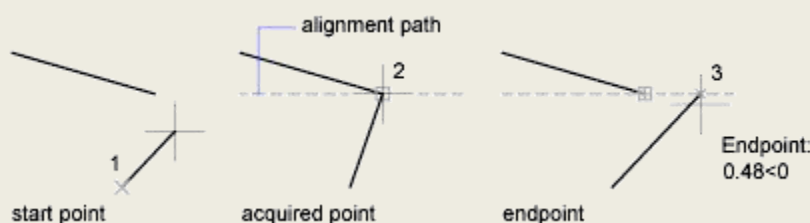
Keyboard NODE (when picking)



The Node Osnap snaps to the center of a Point object. This osnap can be useful if you have created a number of Points with the Measure or Divide commands. You could, for example insert a number of regularly spaced tree symbols (blocks) along a line by using the Node Osnap for the insertion point of each block

## ETRAK :

Use object snap tracking to track along alignment paths that are based on object snap points. Acquired points display a small plus sign (+), and you can acquire up to seven tracking points at a time. After you acquire a point, horizontal, vertical, or polar alignment paths relative to the point are displayed as you move the cursor over their drawing paths. For example, you can select a point along a path based on an object endpoint or midpoint or an intersection between objects.



## LWT :

Lineweights can be turned on and off in a drawing, and are displayed differently in model space than in a paper space layout.

In model space, a 0-value lineweight is displayed as one pixel, and other lineweights use a pixel width proportional to their real-unit value.

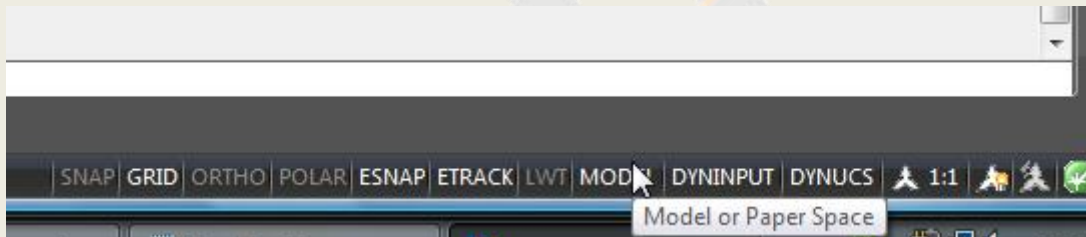
In a paper space layout, lineweights are displayed in the exact plotting width.

Regeneration time increases with lineweights that are represented by more than one pixel. Turn off the display of lineweights to optimize performance of the program.

You can turn the display of lineweights on or off by clicking LWT on the status bar. This setting does not affect the plotting of lineweights.

## MODEL:

In ActCAD you can switch between model and layout by using model button provided in status bar.



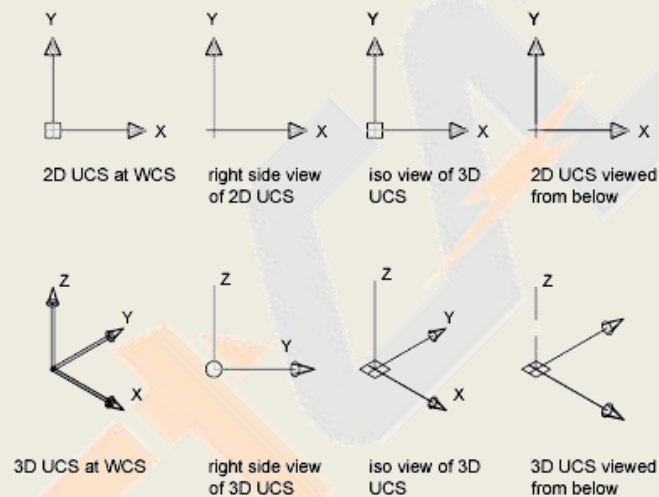
## CAD FILE DATA STRUCTURE:

Designs and drawings created in a CAD system are usually defined and stored using sets of points in what is called world space. In most CAD systems, the world space is defined using a three-dimensional Cartesian coordinate system. Three mutually perpendicular axes, usually referred to as the X, Y, and Z-axes, define this system. The intersection of the three coordinate axes forms a point called the origin. Any point in world space can then be defined as the distance from the origin in the X, Y and Z directions. In most CAD systems, the directions of the arrows shown on the axes identify the positive sides of the Coordinates.

A CAD file, which is the electronic version of the design, contains data that describe the entities created in the CAD system. Information such as the coordinate values in world space for all endpoints, center points, etc., along with the descriptions of the types of entities is all stored in the file. Knowing that ActCAD stores designs by keeping coordinate data helps us understand the inputs required to create entities.

## USER COORDINATE SYSTEM (UCS):

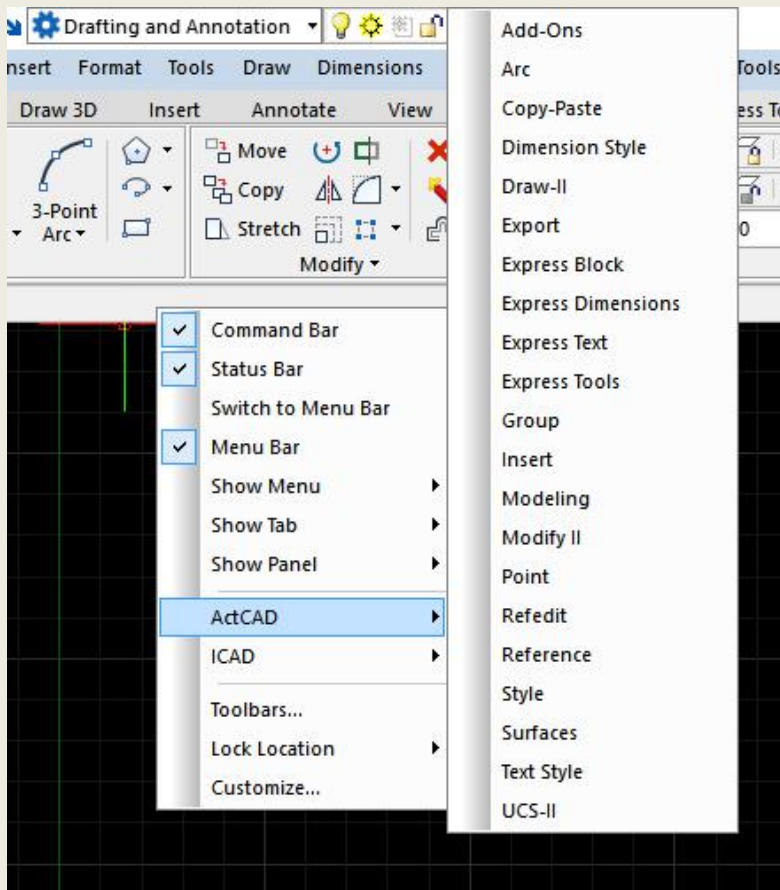
The icon near the bottom left corner of the default ActCAD Graphics window shows the positive X-direction and positive Y-direction of the coordinate system that is active. In ActCAD, the coordinate system that is used to create entities is called the User Coordinate System (UCS). By default, the User Coordinate System is aligned to the world coordinate system (WCS). The world coordinate system is a coordinate system used by ActCAD as the basis for defining all objects and other coordinate systems defined by the users. We can think of the origin of the world coordinate system as a fixed point being used as a reference for all measurements. The default orientation of the Z-axis can be considered as positive values in front of the monitor and negative values inside the monitor.



## SHOWING AND HIDING TOOLBARS:

Display a toolbar with one of the following methods

1. Right-click any toolbar and click a toolbar on the shortcut menu.
2. Click Tools ► Toolbars and click the toolbar to display.



## KEYBOARD SHORTCUT KEYS:

Shortcutkeys are key combinations that start commands. For example, you can press Ctrl+O to open a file and Ctrl+S to save a file, which is the same as clicking Open and Save on the Toolbar or File menu.

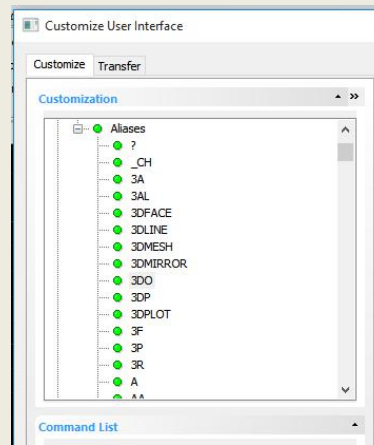
## CREATING AND MANAGING ALIASES:

A command alias is an abbreviation that you enter at the command prompt instead of entering the entire command name.

For example, you can enter c instead of circle to start the CIRCLE command. An alias is not the same as a keyboard shortcut, which is a combination of keystrokes, such as CTRL+S for SAVE.

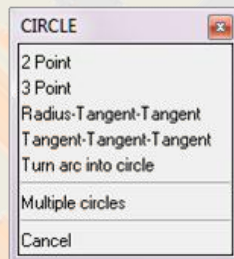
### To Create Aliases:

- Select Customize option from tools menu,
- Select menu customize, then you will get a dialog box namely Customize,
- In that dialog box select aliases tab to create or assign aliases to commands which are available in ActCAD
- In that dialog box you can import and export the aliases



## PROMPT BOXES:

ActCAD commands often provide several options. These options appear in the status bar or command bar and also in a prompt box (called a context menu in Autodesk AutoCAD) initially displayed in the upper right corner of the screen. You can move the prompt box by dragging it; future prompt boxes will appear where you last placed it. The options appear as menu selections. Choose the appropriate option by selecting it in the prompt box. To close a prompt box, click the Close box.



Command options appear in a prompt box

The prompt box isn't displayed by default. To turn on the display of prompt boxes, choose Tools > Options, click the Display tab, and then click the Display Prompt Boxes check box.

## MOUSE SHORTCUTS:

You can use your mouse, often combined with the keyboard, to start commands and perform other actions. The list of mouse shortcuts given below:

Mouse Shortcut	Action
Ctrl + Shift + Hold and drag left mouse button	Real-Time Zoom command
Ctrl + Shift + Hold and drag right mouse button	Real-Time Pan command
Ctrl + Hold and drag left mouse button	Real-Time Sphere command

Ctrl + Hold and drag right mouse button	Real-Time Z command
Shift + Left-click mouse	Deselect entities
Shift + Right-click mouse	Entity snap shortcut menu
Hold and drag left mouse button	Move selected entities
Ctrl + Hold and drag left mouse button	Copy and move selected entities
Right-click mouse	Display shortcut menu for the selected entity
Rotate mouse wheel	Zoom In and Zoom Out commands
Hold mouse wheel, and then move mouse	Pan command

## UNDO:

ActCAD tracks the commands you use and the changes you make. If you change your mind or make a mistake, you can undo, or reverse, the last action or several previous actions.

Toolbar: Standard



Command: undo

shortcut: ctrl+z

## REDO:

To reverse the effect of previous undo commands.

Toolbar: Standard



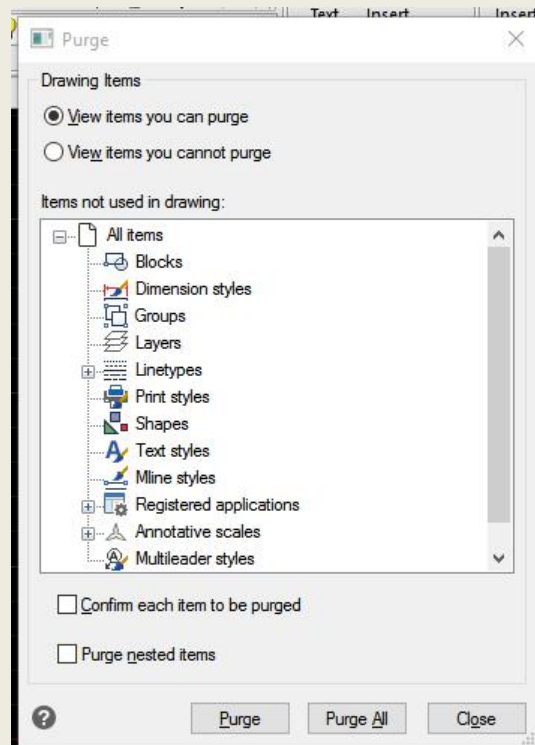
Command: redo

shortcut: ctrl+y

## PURGE:

Do one of the following to get purge dialog box:

- Choose File > Utility > Purge.
- Type purge and then press Enter.



Using the option buttons above the Items List window, you choose if you want to display the items that you can purge (items not used) or if you want to display those that you cannot purge. In the Item List area you can view the items that can be purged divided into categories. The plus symbol next to group names indicates that there are objects you can purge. Linetypes definitions are purged only from the current drawing, the definition in the file icad.lin remains unchanged. If you mark the check box Confirm each item to be purged, you will be asked to confirm before purging the items.

- **Purge nested items:** removes all the unused named objects from the drawing even if they are contained within or referenced by other unused named objects. The Confirm Purge dialog box is displayed and you can cancel or confirm the items to be purged.
- **Nested Blocks:** for example, you create a block from a circle and insert two instances of this block in the drawing. Next, you create a new block that consists of two previous circle blocks: in this way you will have a nested block in your drawing. Marking the check box Purge nested items you can purge in one session even nested blocks. It is recommended to leave active the position Confirm each item to be purged in order to be able to verify the blocks to be purged.
- **Purge:** purges the selected items.
- **Purge all:** purges all the unused items.

## ISOMETRIC SNAPPING :

You can use the Isometric Snap option to create two-dimensional isometric drawings. With the isometric option, you are simply drawing a simulated three-dimensional view on a two-

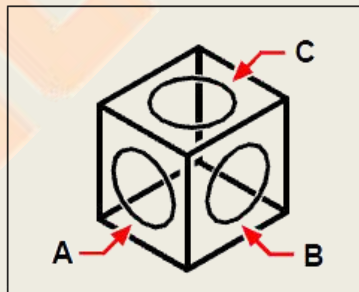
dimensional plane, much the same as you might draw on a piece of paper. Do not confuse isometric drawings with three-dimensional drawings. You create three-dimensional drawings in three-dimensional space. The isometric option always uses three preset planes, which are denoted as left, right, and top. You cannot alter the arrangement of these planes. If the snap angle is 0, the three isometric axes are 30 degrees, 90 degrees, and 150 degrees.

When you check the Isometric Snap option and select an isometric plane, the snap intervals, grid, and crosshairs align with the current plane. The grid is always shown as isometric and uses y-coordinates to calculate the grid spacing. If you click the Draw Orthogonal check box, the program restricts the drawing of entities to the current isometric planes.

- To toggle between isometric planes, press F5

### To turn the Isometric Snap:

1. Do one of the following:
  - Choose Tools > Drawing Settings.
  - Type settings and then press Enter.
2. Click the Coordinate Input tab.
3. Under Change Settings For, select Snap And Grid.
4. Click the Isometric Snap And Grid check box.
5. Under Isometric Crosshairs, click the option for the isometric plane you want (Top, Left, or Right).
6. Click OK.



Isometric planes left (A), right (B), top (C).

### ALIGNMENT:

When you create paragraph text, you can set the text alignment by specifying the attachment point location in relation to the paragraph text boundary and the direction in which text flows within the boundary. You can set the paragraph text alignment either in the prompt box displayed after you specify the first corner of the paragraph text boundary or from the Multiline Text dialog box. You can specify the attachment point at the top left, top center, top right, middle left, middle center, middle right, bottom left, bottom center, or bottom right. The paragraph text can flow left to right, right to left, top to bottom, or bottom to top.

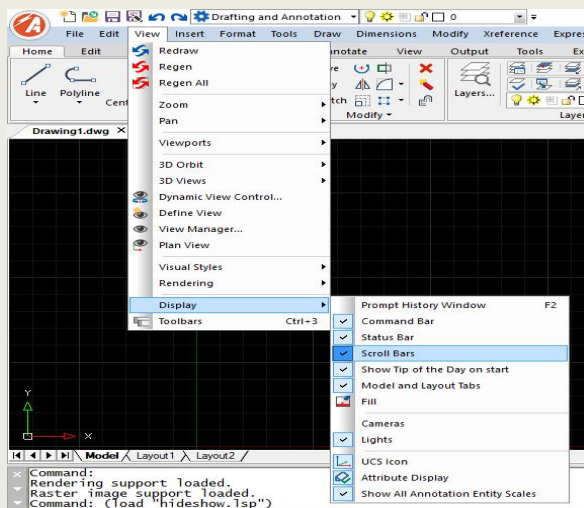
## CURSOR COORDINATE VALUES:

When your cursor rolling over Graphics window the location of your cursor will be shown on status bar, these values will be shown respective of your cursor location from UCS.



## SCROLL BARS:

To show or hide the scroll bars that display on the right side and bottom of the ActCAD window or viewport, select the Show Scroll Bars check box. You may want to hide the scroll bars if you only use the Pan command to scroll drawings

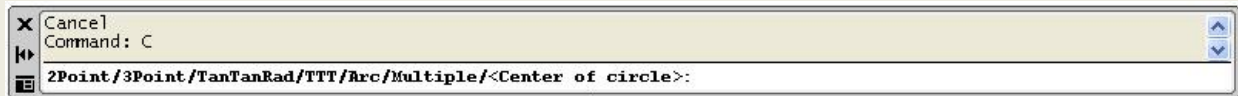


SCROLL BARS

## COMMAND BAR:

The command bar is a dock able window in which you type ActCAD commands and view prompts and other program messages. To display the command bar, choose View > Display > Command Bar. When initially displayed, the command bar is docked between the toolbars and the drawing. It displays the three most recent lines of prompts. You can move the command bar by dragging it.

When the command bar is floating, you can drag the top or bottom of the window to change the number of lines of text it displays. You can dock the command bar at the top or bottom of the drawing.



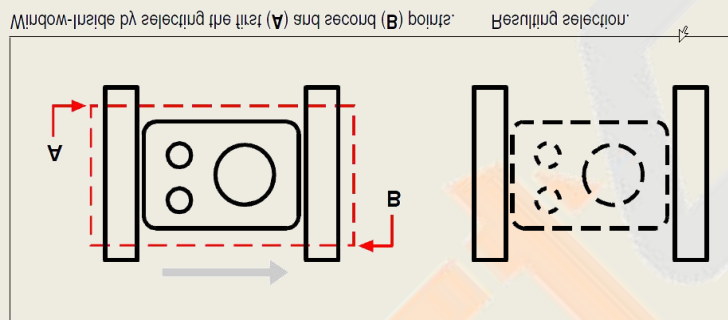
### SELECTION WINDOW OPTIONS:

When you choose a command that requires you to select entities (when you're deleting or changing entity properties, for example), you can use any of the following selection methods by choosing them in the prompt box or entering them in the command bar:

1. Window selection
2. Crossing selection

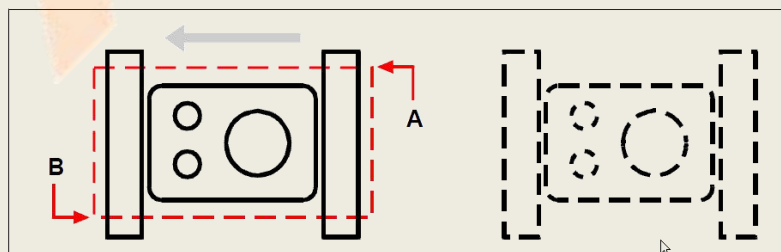
#### Window selection:

- Click to select a point in the drawing.
- Click to the right of the first point to select a second point in the drawing.



#### Crossing selection:

- Click to select a point in the drawing.
- Click to the left of the first point to select a second point in the drawing.



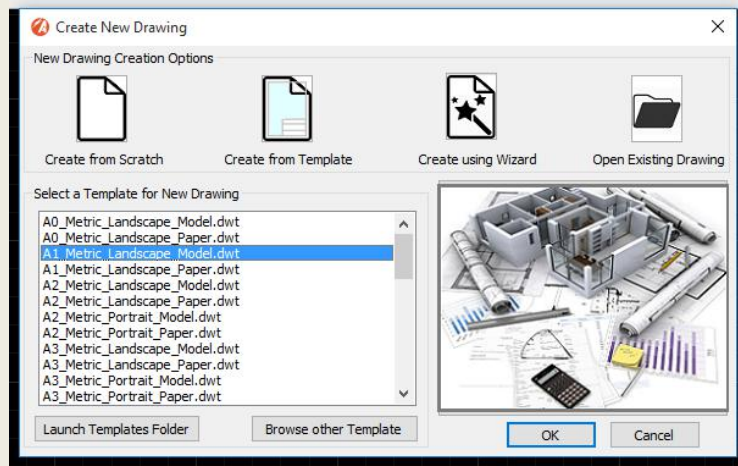
#### CALCULATOR:

ActCAD enables you a calculator to make calculations much fast simply type CAL in command window and press enter then you will get a windows calculator

## Chapter 3: Standard Commands

### DRAWING TEMPLATES:

- A drawing template file is used to provide consistency in the drawings that you create by providing standard styles and settings.
- Drawing template files have a .dwt file extension.
- When you create a new drawing based on an existing drawing template file and make changes, the changes in the new drawing do not affect the drawing template file.



A default set of drawing template (DWT) files is installed with ActCAD which are used to create 2D drawings and 3D models. Many of the default drawing templates are offered in two measurement types: imperial and metric. When using one of the default drawing templates you need determine if you will be working in 2D or 3D, and which measurement type best describes your work.

While the default templates provide for a quick way to start creating a new drawing, it is best to create drawing templates specific to your company and the drawings you create. You can use Initial Setup to select a drawing template that is close to your requirements, but you will still need to do some work to make the drawing template file work best for you.

### CREATING A NEW TEMPLATE:

When you need to create several drawings that use the same conventions and default settings, you can save time by creating or customizing a drawing template file instead of specifying the conventions and default settings each time you start. Conventions and settings commonly stored in template files include

- Unit type and precision
- Title blocks, borders, and logos

- Layer names
- Snap, Grid, and Ortho settings
- Grid limits
- Annotation styles (dimension, text, table and multileader)
- Line types

By default, drawing template files are stored in the *template* folder, where they are easily accessible.

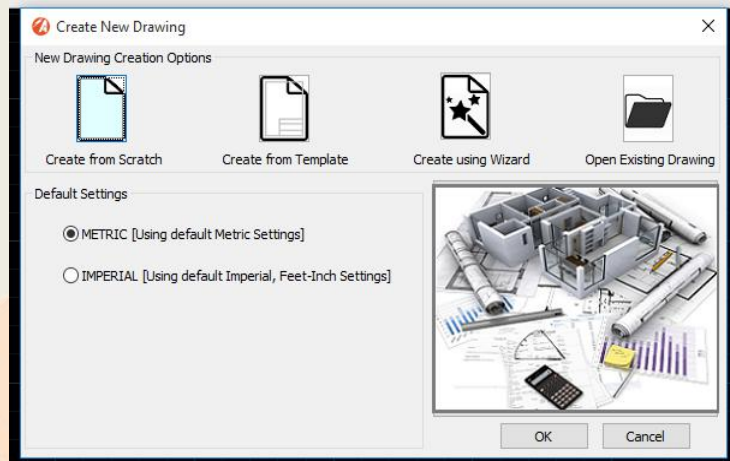
## CREATE A NEW FILE:

**Command NEW:** Creates a new drawing. ActCAD opens a START dialog box, in which you can select the Default settings like Unit system of your drawing either Metric or English. And you can use required templates

Menu: File » New

Keyboard: NEW

Shortcut key: N



## OPEN FILE:

You can open drawing (.dwg) files, Drawing Exchange Format (.dxf) files, Design Web Format™ (.dwf) files, and drawing template (.dwt) files.

You can also open and check drawings that you suspect are damaged.


## Opening an existing drawing

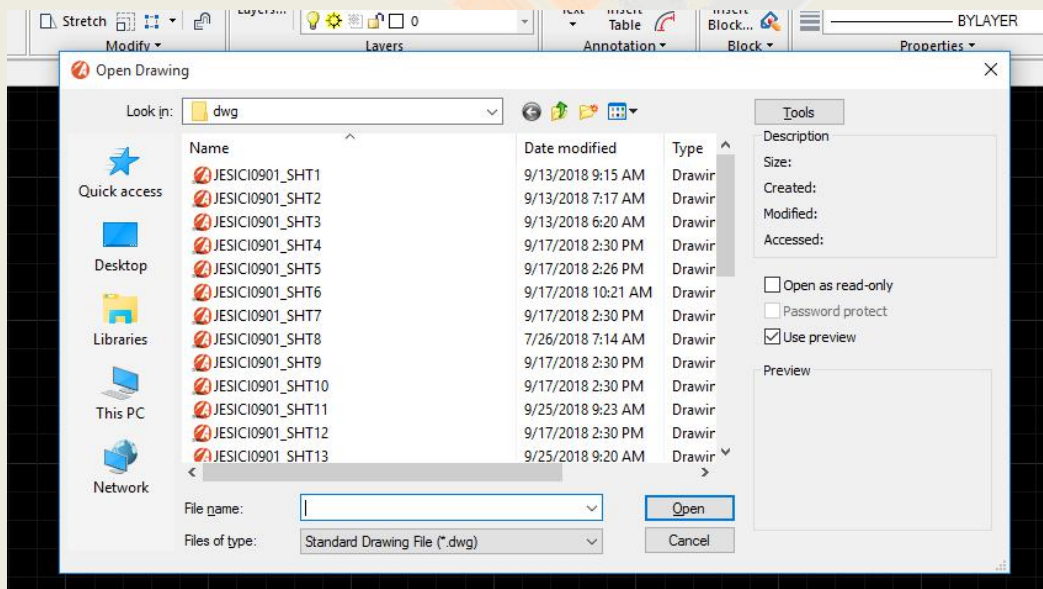
You can open any of these drawing files:

- Standard drawing files with a .dwg extension.
- In addition to your own drawing files, you can open and use one of the sample drawings that are included with ActCAD.

- Drawing Exchange Format files with a .dxf file extension.
- Design Web Format files with a .dwf file extension.
- Drawing templates with a .dwt file extension.

### To open an existing drawing

1. Use one of the following methods:
  - Choose File > Open.
  - On the Standard toolbar, click the Open tool (  ).
  - Type open and then press Enter.
2. In Files of Type, choose the type of file you want to open.
3. Choose the folder containing the desired file.
4. Do one of the following:
  - Choose the drawing you want to open and then click Open.
  - Double-click the drawing you want to open.



### CLOSE FILE:

Menu: File » Close      Keyboard: CLOSE

Closes the current drawing file.

Before executing CLOSE command, if the current drawing file is not saved after modification, the system will prompt the user whether to save or not. If current drawing file is unchanged or want to cancel modification, you can close the file directly.

**SAVE FILE:**

Keyboard: SAVE

Aliases: SA

Shortcut key: CTRL+S

Saves the current drawing under the current file name. The system displays the Save Drawing As dialog box, in which you can save the file with a new name or an existing name. The command line shows as follows:

File name (extension defines export type) <current path>: Enter a file name and extension or press ENTER

**SAVE AS:**

Menu: File » Save as

Keyboard: SAVE AS

Shortcut key: CTRL+SHIFT+S

Saves the current drawing under another name or another format. The system displays Save Drawing As dialog box, in which you can specify a new name and format to save the current drawing.

The prompts on the command line are as follows: File name (extension defines export type) <current path>: Enter a file name and extension or press ENTER

**CUT:**

Menu: Edit » Cut

Keyboard: CUTCLIP

Shortcut key: CTRL+X

Copies selected object to the clipboard and then deletes the object from current file.

Command line prompt: Select entities to cut to the clipboard: Select entities to cut After executing Cut command, the contents of the clipboard can be pasted into documents or drawing as OLE objects, detail procedures can reference in PASTECLIP command and PASTESPEC command. But if the object has links before cutting, the link will not exist after cutting.

**COPY:**

Menu: Modify » COPY

Keyboard: COPY

Alias: CO,CP

Shortcut key: CTRL+C

Copies object.

**PASTE:**

Menu: Edit » Paste

Keyboard: PASTE

Shortcut key: CTRL+V

The system pastes the entities copied to the clipboard into the drawing at the specified insertion point.

Command line prompt:

Insertion point: Specify a point to paste.

## PASTE AS BLOCK:

Menu: Edit » Paste as Block

Keyboard: PASTEBLOCK

The system pastes the entities copied to the clipboard into the drawing as a block at the specified insertion point.

Command line prompt:

Insertion point: Specify a point to paste a block

## PAN:

Menu: View » Zoom » Realtime

Keyboard: PAN

Shortcut key: P

Moves the view in the current viewport.

Entering PAN on the command line displays the following images while the cursor changes to a hand cursor. You can lock the cursor to the current position relative to the viewport coordinate system by holding down the pick button on the pointing device.



Press Esc or Enter to exit, or right-click to display shortcut menu...

## ZOOM OPTIONS:

Menu: View » Zoom

Keyboard: Zoom

Shortcut key: Z

Zooms to display the entire drawing in the current viewport. You can up and down scroll to zoom graphics while pressing SHIFT or CTRL.

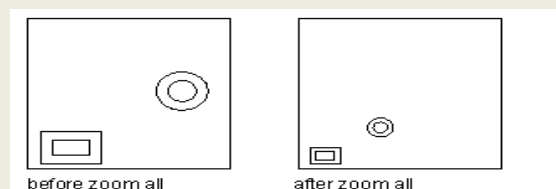
Command line prompt:

**ZOOM:** *In/Out/All/Dynamic/Center/Extents/Left/Previous/Right/Window/Objects/Scale(nX/nXP)/<Realtime>*: Enter a scale factor or an option or press ENTER

**In:** Zooms in to display the current graphics.

**Out:** Zooms out to display the current graphics.

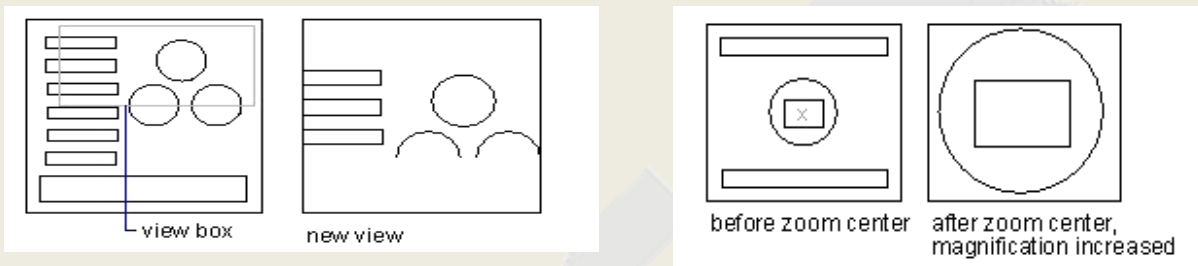
**All:** Zooms to display all the objects in the current viewport. It can display all the objects even if the



drawing extends outside the grid limits.

**Dynamic:** Zooms to display the generated portion of the drawing with a view box. We can size and position the view box. Positioning and sizing the view box pans or zooms to make sure the image inside the view box. It will display the view box once you use this option. Then drag it to the location you want and click, it will display the view box that you can zoom in or zoom out. Resize it and press ENTER to zoom, or click to return to the panning view box.

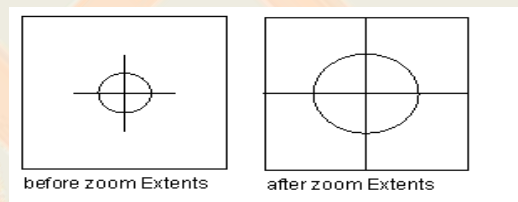
**Center:** Displays the drawing defined by a center point and a magnification value. Increases the magnification as the value of the height is smaller. Decreases the magnification as the value of the



height is larger.

Center of view <current>: Specify a point or press ENTER.

Enter zoom factor (nX/nXP), or view height <current>: Enter a scale value or specify the height or press ENTER.



**Extents:**

Zooms to display the drawing extents and displays all the objects as large as possible inside the viewport. In a 3D view, ZOOM All is equivalent to ZOOM Extents.

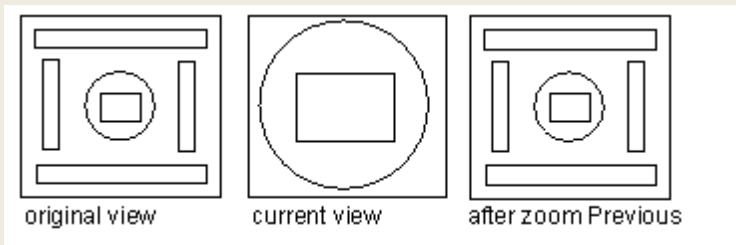
**Left:**

Specifies left-hand corner point and zoom radio or height to observe window.

Lower left corner: Specify a point

Enter zoom factor (nX/nXP), or view height:<current>: Enter a scale value or specify the height or press ENTER.

### Previous:



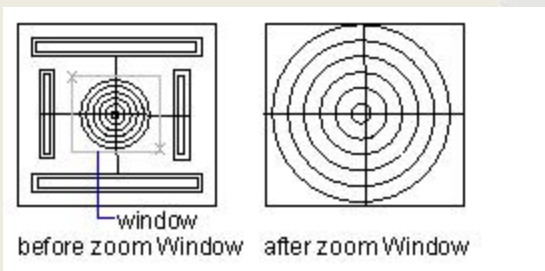
Zooms to display the previous viewport. You can restore up to 10 previous views by default.

**Right:** Specifies a upper right point and zoom ratio or height to observe window.

Upper right corner: specify a point

Enter zoom factor (nX/nXP), or view height: <current>: Enter a scale value or specify the height or press ENTER.

### Window:

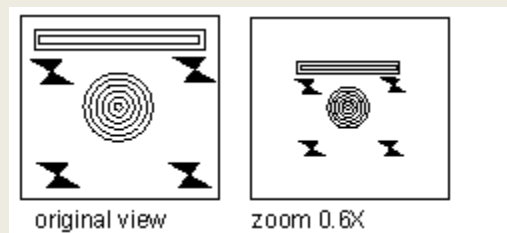


Zooms to display the specified rectangular window.

First point: Specify a point 1

Opposite corner: Specify a point 2

**Objects:** Scales specified objects so as to display them at the center of the drawing area as large as possible. Select entities to zoom: Select objects to be displayed in maximum size at the center of the drawing area, press ENTER when finish selecting.



Scale(nX/nXP):

Zooms to display the current drawing at a specified value.

**Realtime:** Pans on screen while pressing the pick button, it will zoom to display the current drawing in the logical extent. Press Esc or Enter to exit, or right-click to display shortcut menu... The cursor will changes to a magnifying glass with plus(+) signs.

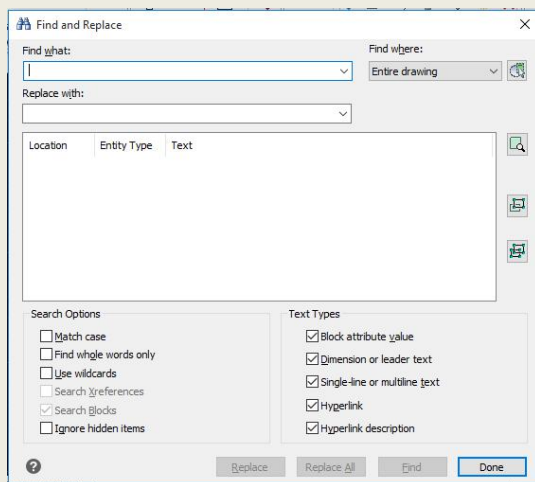


## Find and Replace:

This command allows you to find, replace, or select a text and it controls the results of the search.

### To find and replace a text

1. Do one of the following:
  - Select Edit > Find and Replace
  - Type find and then press Enter.
2. In the find and replace mask you can see these options:



- **Find Text String:** Indicates the text string to find. Enter a text string including any wild card characters, or choose one from the list of the six most recently used strings.
- **Replace With:** Indicates the text string you want to use to replace the found text. Enter a string or choose one from the list of the most recently used strings.
- **Search In:** Indicates whether to search the entire drawing or only the current selection. If something is already selected, Selection is the default value. If nothing is selected, the default value is Entire Drawing. You can use the Select Objects button to close temporarily the dialog box and create or modify the selection set. When selection is completed press Enter or right click to open the dialog box again.

- **Select Objects Button:** Closes temporarily the dialog box so that you can select objects in your drawing. Press ENTER to return to the dialog box. When you select objects, the Search In field displays Current Selection.
- **Options:** Displays the Find and Replace Options dialog box, in which you can define the type of words that you want to find.
- **Find:** Finds the text that you have entered in the Find Text String field. This option is not available if you have not entered any text in the Find Text String field. The found text is displayed in the Context area. Once you have found the first instance of the text, the Find option becomes Find Next, which you can use to find the next instance.
- **Replace:** Replaces the found text with the text that you entered in the Replace With field.
- **Replace All:** Finds all occurrences of the text that you entered in Find Text String and replaces it with the text in Replace With. The Search In setting controls whether to find and replace the text in the entire drawing or the text in the current selection. The status area confirms the replacements and indicates the number of replacements that have been made.
- **Select All:** Finds and selects all the objects containing instances of the text that you entered in the Find Text String field. This option is available only when you select objects in a drawing and the Search In control is set to Selection. When you choose Select All, the dialog box closes and a message in the command prompt shows the number of objects founded and selected. Select All does not replace the text.
- **Zoom To:** Displays the area in a drawing that contains the results of a find or replace search.

## EXIT:

The exit command is used for to close the entire cad drawing file window.

## Chapter 4: Draw Commands-2D

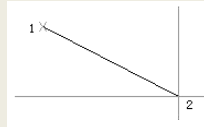
### LINE:

Draws lines at any length. You can enter x, y, z coordinates of the point to specify start and end point of 2D or 3D entities. When using LINE command, system will always prompt to enter point, before pressing ENTER to finish the command, users can specify series of points, but each line is an independent object.

Menu: Draw -> Line

Command: LINE

Aliases: L



```
Command: line
Start of line: 0,0,0
Angle/Length/<End point>: 100,0,0
Angle/Length/Follow/Undo/<End point>: 100,100,0
Angle/Length/Follow/Close/Undo/<End point>: 0,100,0
Angle/Length/Follow/Close/Undo/<End point>: 0,0
```

### MLINE:

Creates multiple parallel lines.

Menu: Draw -> Mline

Command: MLINE

Aliases: DL



```
Command: mline
Current settings: Justification = Top, Scale = 20.0000, Style = STANDARD
Specify start point or [Load style/Justification/Scale/Style]: 1
Current settings: Justification = Top, Scale = 20.0000, Style = STANDARD
Specify start point or [Load style/Justification/Scale/Style]: 100,0
Next point: 200,0
Specify next point or [Undo]: 200,100
Specify next point or [Close/Undo]: 0,0
```

### DRAW TRACES:

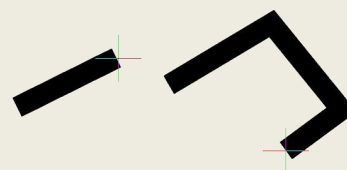
Creates line segments with required width

Menu: Draw -> Draw Traces

Command: -trace

Aliases: trace

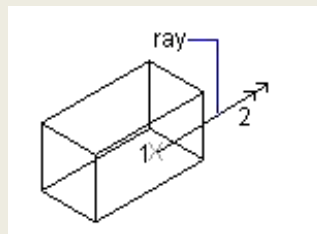
```
Command: trace
Width of trace <1>: 10
Start of trace: 150,0
Next point: @100<30
Next point: @100<90
Next point:
```



**RAY:**

Creates a ray extends infinitely toward the direction you specify.

Menu: Draw -> Ray      Command: RAY      Command: RAY

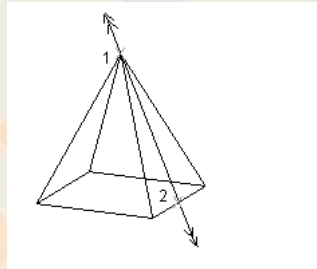


```
Command: ray
Infinite ray: Bisect/Horizontal/Vertical/Angle/Offset/<Start of Ray>: 0,0
Direction: @100<30
Direction: @100<45
```

**CONSTRUCTION LINE:**

Creates a construction line which extends infinitely.

Menu: Draw -> Construction Line      Command: XLINE      Aliases: XL

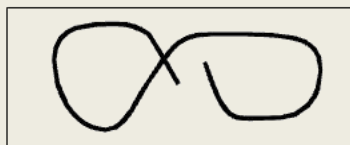


```
Command: xline
Infinite line: Bisect/Horizontal/Vertical/Angle/Offset/<Point along Line>: a
Reference/<Enter angle (121)>: 45
Location: 0,0
Location: 10,0
```

**FREEHAND:**

A freehand sketch consists of many straight line segments, created either as individual line entities or as a polyline. Before you begin creating a freehand sketch, you must set the length, or increment, of each segment. The smaller the segments, the more accurate your sketch, but segments that are too small can greatly increase the file size.

Menu: Draw -> Freehand      Command: Freehand      Aliases: Freehand



A freehand sketch consists of individual line entities or a polyline.

```

Command: freehand
Freehand: Length of segments <1>: 20
Press ENTER to end/Quit/Pen down/Erase/Write to drawing/:(Sketching...)
Writing freehand sketch to drawing...
20 lines recorded.

```

## POLYLINE:

With ActCAD, a polyline is a connected sequence of arcs and lines that is treated as a single entity. You can draw a polyline with any linetype and using a width that either remains constant or tapers over the length of any segment. When editing a polyline, you can modify the entire polyline or change individual segments.

Menu: Draw -> polyline    Command: polyline    Aliases: pl



Curved polyline with tapered width.

Polyline with straight segments.

Closed polyline.

```

Command: polyline
ENTER to use last point/Follow/<Start of polyline>:
Arc/Distance/Follow/Halfwidth/Width/<Next point>: w
Starting width <0>: 12
Ending width <12>: 10
Arc/Distance/Follow/Halfwidth/Width/<Next point>: @100<30
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: @100<120
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: c
Command:

```

## 3D POLYLINE:

Draws a three-dimensional polyline with straight segments. Although the 3D Polyline command does not have as many options as the Polyline command, you can apply editing changes, such as splining; with the Edit Polyline command after the three-dimensional polyline is drawn.

Menu: Draw -> 3D polyline    Command: 3Dpoly    Aliases: 3P



An example of a three-dimensional polyline.

```

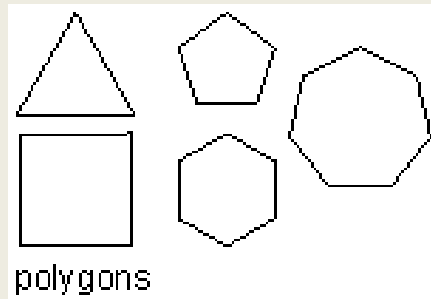
Command: 3dpoly
ENTER to use last point/Follow/<Start of polyline>: 100,100
Next point: @100,0,0
Undo/Next point: @0,0,100
Close/Undo/<Next point>: @0,100,0
Close/Undo/<Next point>:

```

## POLYGON:

Creates a polygon entity. A polygon entity is created based on the polylines.

Menu: Draw -> Polygon      Command: POLYGON      Aliases: POL

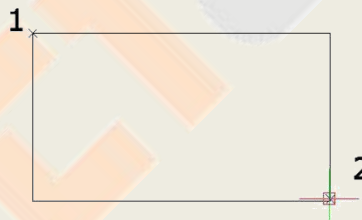


```
Command: pol
Polygon: Multiple/Width of line/<Number of sides> <4>: m
Polygon: Width of line/<Number of sides> <4>: w
Polyline width <0> :20
Polygon: Width of line/<Number of sides> <4>: 6
Specify center of polygon or [Edge]: 100,0
Enter an option [Inscribed in circle/Circumscribed about circle] <C>: c
Enter an option [Inscribed in circle/Circumscribed about circle] <C>:
```

## RECTANGLE:

Creates a rectangular polyline entities.

Menu: Draw -> Rectangle      Command: RECTANG      Aliases: REC

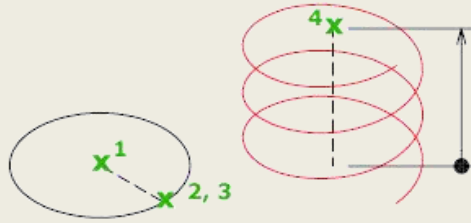


```
Command: rec
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: t
Thickness to use for all rectangles: 20
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: w
Width to use for all rectangles: 10
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: e
Elevation to use for all rectangles: 50
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: c
Defaults/First chamfer distance to use for all rectangles <0> : 15
Second chamfer distance to use for all rectangles <0>: 20
Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width/<Select first corner of rectangle>: 100,0
Specify other corner point or [Area/Dimensions/Rotation]: d
Enter rectangle length <10>: 50
Enter rectangle width <10>: 25
```

## HELIX:

Creates a 2D spiral or 3D spring.

Menu: Draw -> helix      Command: helix      Aliases: helix



```

Command: helix
Specify center point of base:100,0
Specify base radius or [Diameter] <1>: 50
Specify top radius or [Diameter] <1>: 30
Specify helix height or [Axis endpoint/Turns/turn Height/twist] <1>: t
Enter number of turns :10
Specify helix height or [Axis endpoint/Turns/turn Height/twist] <1>: 150
  
```

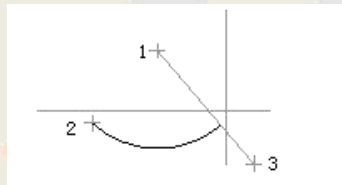
### ARC-VARIOUS OPTIONS:

Draws an arc.

Menu: Draw -> Arc

Command:ARC

Aliases:A



```

Command: a
Center/<Start of arc>: c
Center of arc: 100,100
Start Point: @100<30
Angle/Length of chord/<End point>: l
Length of chord: 100
  
```

### CIRCLE-VARIOUS OPTIONS:

Draws a circle of any size.

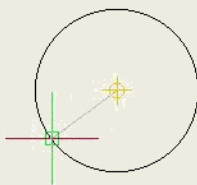
Predicted method of drawing circle is through center point and radius. there are many other ways to draw circle.

Menu: Draw -> Circle

Command: CIRCLE

Aliases:C

Selects Multiple option will draw multiple circles with the same setting.



```

Command: circle
2Point/3Point/TanTanRad/TTT/Arc/Multiple/<Center of circle>: m
2Point/3Point/TanTanRad/Arc/<Center of circle>: 100,100
Diameter/<Radius> <75>: d
Diameter of circle <150>: 150
Center of circle: 150,150
Center of circle:
  
```

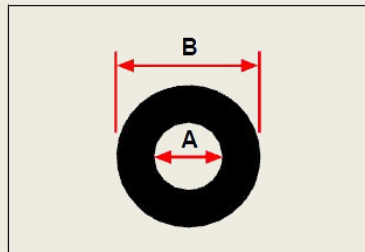
**DONUT:**

Donuts are solid, filled circles or rings created as closed, wide polylines. You can draw a donut using one of several methods. The default method is to specify the inside and outside diameters of the donut, and then specify its center. You can then create multiple copies of the same donut by specifying different center points until you press Enter to complete the command.

Menu: Draw -&gt;Donut

Command: Donut

Aliases: do



Inside diameter (A) and outside diameter (B).

```
Command: donut
2Point/3Point/RadTanTan/<Inside diameter of donut> <0.5>: 25
Outside diameter of donut <1>: 50
Center of donut: 100,0,0
Center of donut:
```

**PLANE:**

With the Plane tool, you can draw rectangular, triangular, or quadrilateral areas filled with a solid color. The default method is to specify the corners of the plane. After you specify the first two corners, the plane is displayed as you specify the remaining corners. Specify corner points in a triangular manner. The program prompts you for the third point and then the fourth point. If you continue specifying points, the third- and fourth-point prompts toggle until you press Enter to complete the command.

Menu: Draw -&gt;Plane

Command: plane

Aliases: plane or solid

```
Command:
_SOLID
Rectangle/Square/Triangle/<First point of plane>: 100,0
Specify second point: 100,100
Specify third point: 0,0
Specify fourth point or <exit>: 0,100
Specify third point:
```

**SPLINE:**

SPLINE fits a smooth curve to a series of points with the specified tolerance.

Menu: Draw -&gt;Spline

Command: Spline

Aliases: spl



```
Command: spline
Object/<First point for spline>: 0,0,0
Second point: 100,100
Close/Fit tolerance/<Next point>: 200,0,0
Close/Fit tolerance/<Next point>: f
Enter fit tolerance (in drawing units) <0.000000>: 1
Close/Fit tolerance/<Next point>: f
Enter fit tolerance (in drawing units) <1.000000>: 5
Close/Fit tolerance/<Next point>: c
Enter tangent:
```

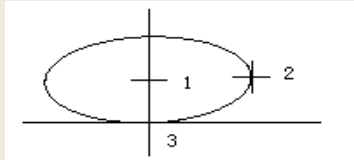
## ELLIPSE:

Creates an ellipse or an elliptical arc

Menu: Draw ->Ellipse

Command: Ellipse

Aliases: el



```
Command: ellipse
Arc/Center/<First end of ellipse axis>: 0,0,0
Second end of axis: @100<0
Rotation/<Other axis>: @25<50
```

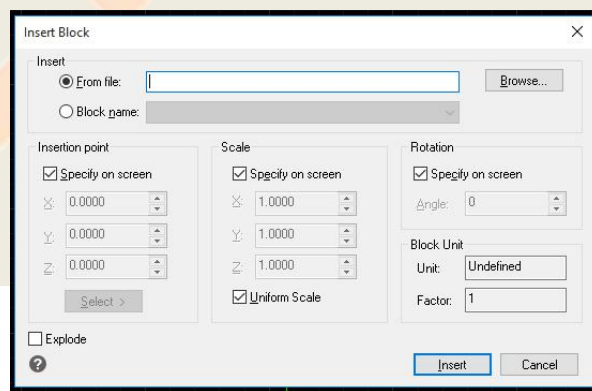
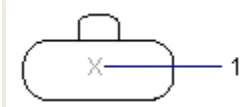
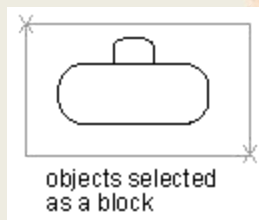
## BLOCK-VARIOUS OPTIONS:

You can combine any number of entities into a single block. After you create a block, you can insert copies of it into a drawing. Each block insertion is treated as a single entity; for example, you can rotate or scale each block when you insert it. The program adds the name of the new block you insert to the Block Name list in the ActCAD Explorer.

Menu: Draw ->Block

Command: Block

Aliases: B



Block Definition Dialog box

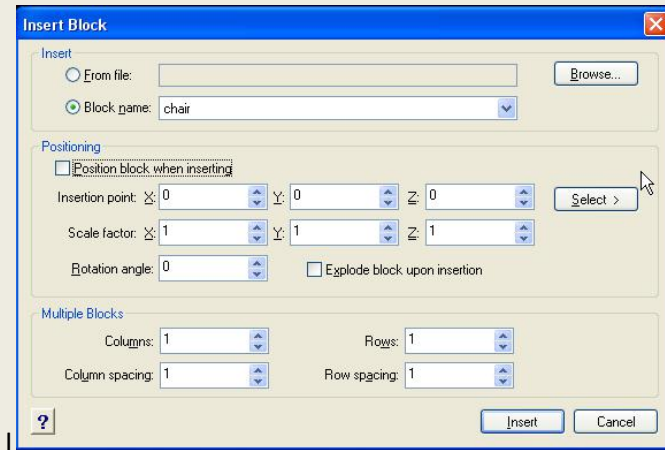
## Insert block:

Opens a Insert Block dialog box in this you can specify scale of block and location of block etc..

Menu: Insert-&gt;Block

Command: Insert

Aliases:



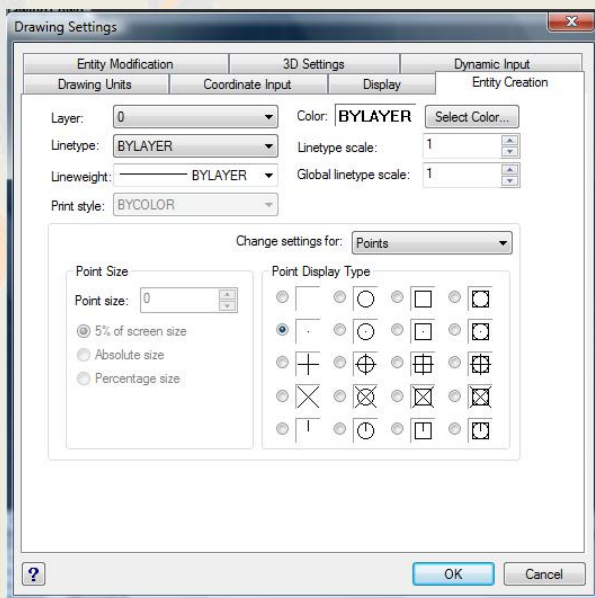
## POINT-VARIOUS OPTIONS:

Draws point entities.

Menu: Draw-&gt;Point-&gt;Single point or Multiple points,

Command: Point

Aliases: po



```
Command: point
Settings/Multiple/<Location of point>: m
Settings/<Location of point>: 100,100
Settings/<Location of point>: 200,200
Settings/<Location of point>: s
Settings/<Location of point>:
Command:
```

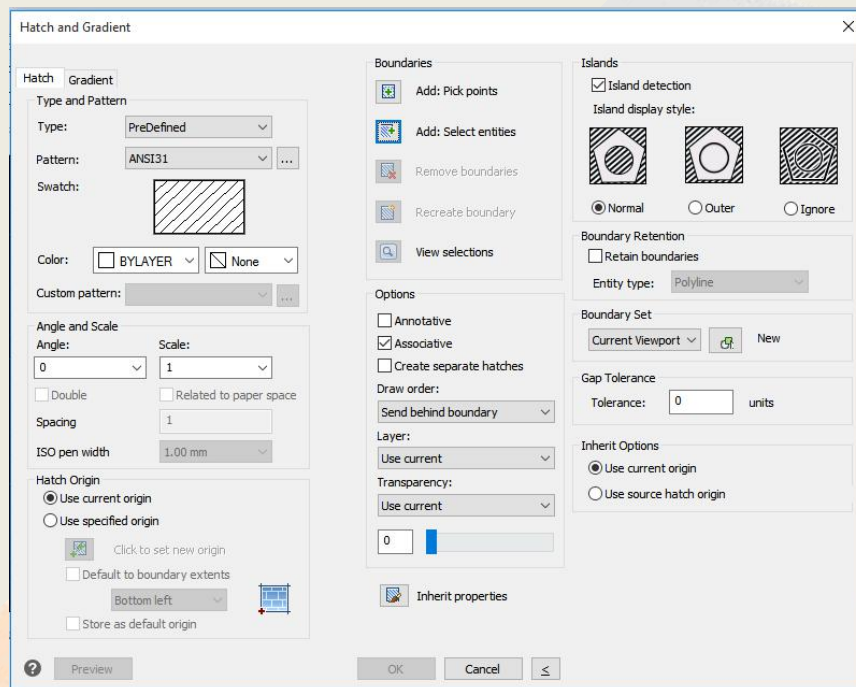
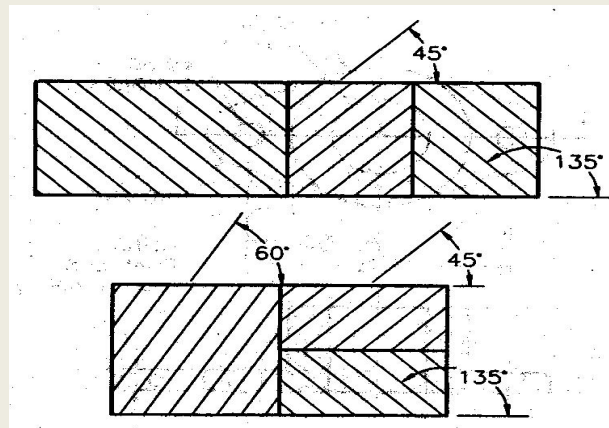
## HATCH:

Fills an enclosed area or selected objects with a hatch pattern or gradient fill.

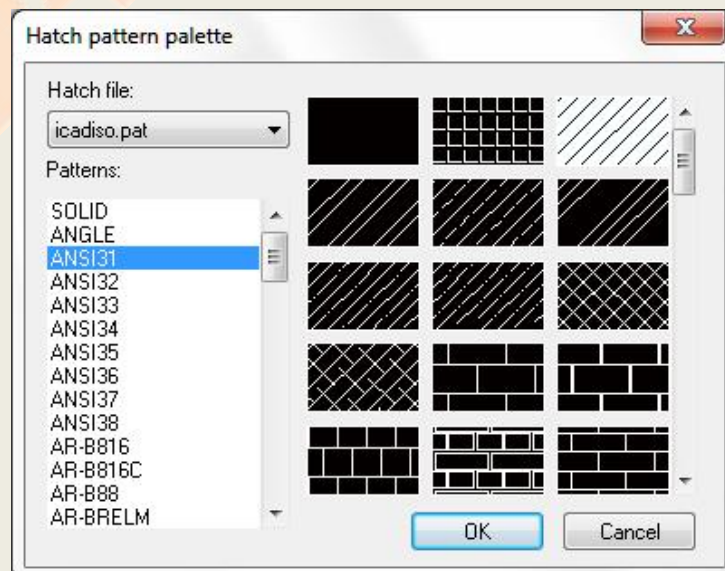
Menu: Draw -&gt;Hatch

Command: BHatch

Aliases: H

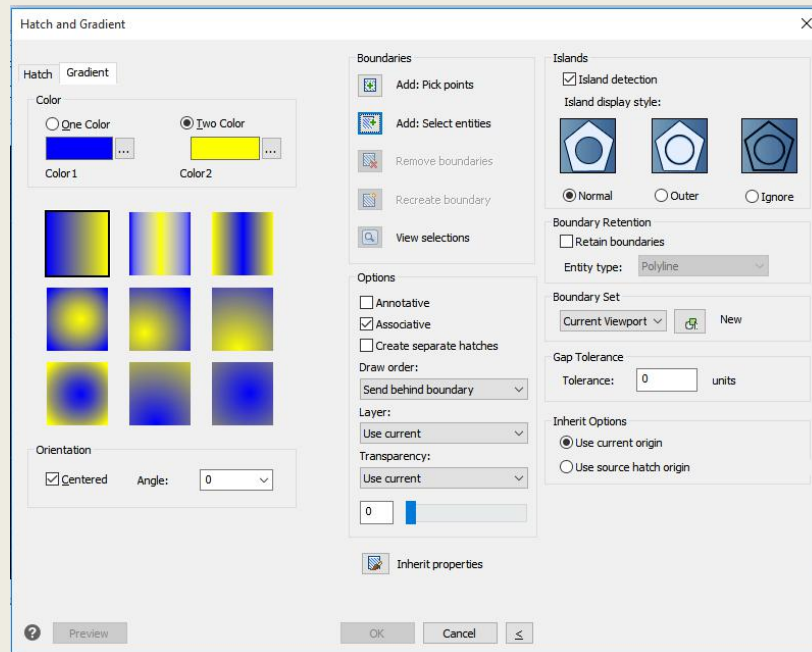


To Add patterns:



## GRADIENT HATCH:

Defines the appearance of the gradient fill to be applied.



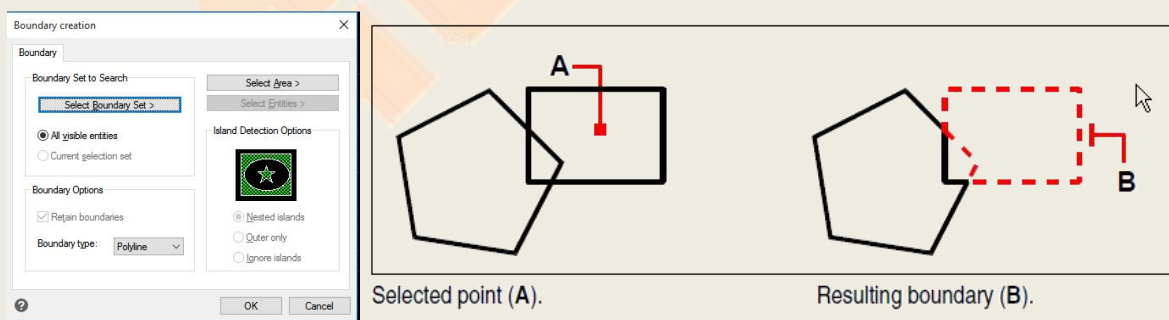
## BOUNDARY:

With the Boundary command, you can designate a specific area of a drawing for operations such as hatching and dimensioning. You create a boundary polyline by selecting an area inside a closed loop. The area you select can be bounded by a single closed entity or by multiple entities that intersect

Menu: Draw ->Boundary

Command: Bpoly

Aliases: Bpoly



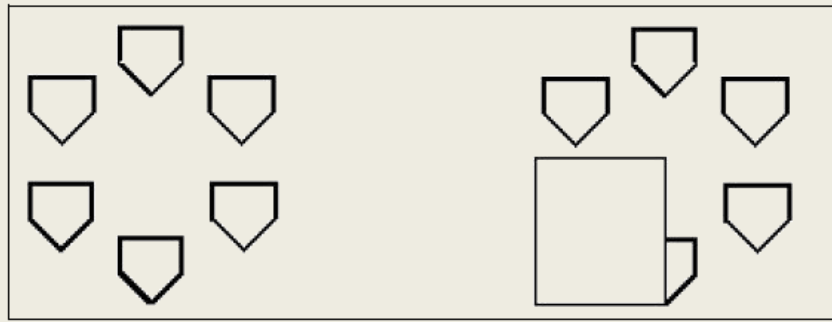
From boundary Dialog box click on Add Pick points to define your required boundary.

## WIPEOUT:

Wipeouts are created using existing polygons, closed zero-width polylines made up of only line segments, or new polylines that you draw while using the Wipeout command.

Menu: Draw ->Wipeout Command: Wipeout

Aliases: Wipeout



Drawing without wipeout.

Drawing with wipeout.

```

Command: WIPEOUT
Specify first point or [Frames/Polyline] <Polyline>:
Angle/Length/<End point>:
Angle/Length/Follow/Undo/<End point>:
Angle/Length/Follow/Close/Undo/<End point>:
Angle/Length/Follow/Close/Undo/<End point>:
  
```

### Revision Cloud:

The revision cloud consists of a series of consecutive arcs of a polyline. It is used to draw attention to certain parts of a drawing during its revision.

Revising your drawings it is possible to use the Revision cloud command to mark out your revisions. REVCLLOUD creates a polyline of consecutive arcs making up an object in a cloud shape.

It is possible to set arc length values for a revision cloud. REVCLLOUD stores the last used arc length.

Menu: Draw -> Revision cloud    Command: Revision cloud

Aliases: Revcloud



```

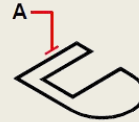
Command: revcloud
Arc length set at 14.85
Arc length/<Pick cloud starting point>: a
Arc length <14.85>: 10
Pick cloud start point:
Guide crosshairs along cloud path...
Finished.
  
```

**Region:**

You can convert a closed entity into a two-dimensional region. After you create a region, you can modify it using the various three-dimensional tools. For example, you can create a region from a square, and then extrude the square to create a three-dimensional cube.

Menu: Draw -> Region    Command: Region    Aliases: Rev

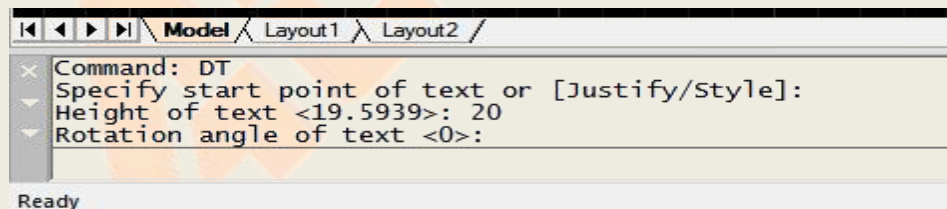
```
Command: reg
Select objects:
Opposite corner:
Entities in set: 15
Select objects:
2 loops extracted.
2 Regions created.
```

**Text :****Single-line Text:**

This option is used to create a single-line text entity. Each line of text is a separate entity and can be rotated, adjusted and resized. To begin another line, press ENTER after entering characters at the command line. As you type at the command line, the text you are typing is displayed on the screen. By applying a style to the text, you can use a variety of character patterns or fonts that you can stretch, compress, make oblique, mirror, or align in a vertical column.

Menu: Draw -> Single line text    Command: Single line text    Aliases:DT

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**Multi-line Text:**

MTEXT creates paragraphs that defines the width of the paragraph and its location in the drawing to display text you enter.

Menu: Draw -> Text -> Multiline Text    Command: TEXT    Aliases: MT

**Justification:**

Determines both text justification and text flow, for new or selected text, in relation to the text boundary. The current justification (Top Left, by default) is applied to new text.

You can get more details from the following images.]

```
Command: mlt  
Multiline Text: First corner for block of text: 100,100  
Justification/Line spacing/Rotation/Style/Height/Direction/Width/Columns<Opposite corner for block of text>:  
Command:
```

**Rotation:** Specifies a rotation angle for the multiline text entity.

Rotation angle for block of text <current>: Specify a point or enter a angle value or press ENTER

**Style:** Specifies the style for the multiline text.

Text style to use (or '?') <current>: Enter a style name or ? to list all available styles or press ENTER

**Height:** Specifies the height for the multiline text.

height <current>: Enter a value or press ENTER

**Direction:** Specifies the direction to create multiline text entity.

Specify drawing direction: Left-to-right/Right-to-left/Top-to-bottom/Bottom-to-Top/Style<current>: Enter an option or press ENTER

**Width:** Specifies the height of text box.

Width of text box: Specify a point or enter a value

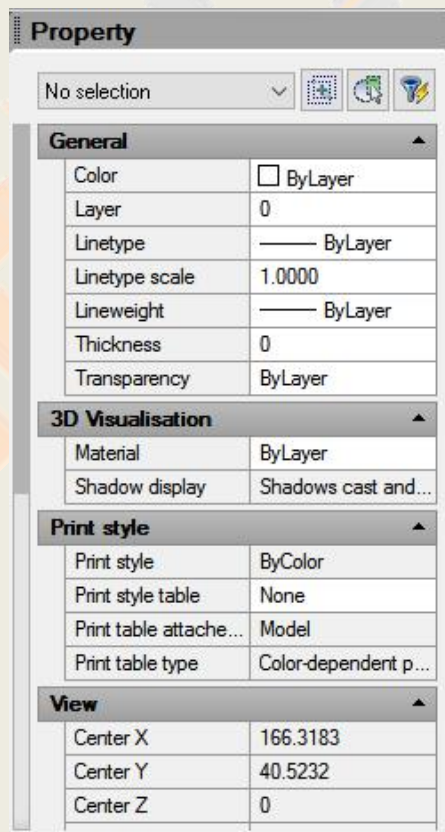
## Chapter 5 Modify Commands - 2D

### PROPERTIES:

You can change the layer, thickness, linetype, color, and linetype scale of one or more entities. Depending on the type of entity or entities you select, you can also change other properties, such as the start point and endpoint of lines, the center point and radius of circles, and the vertices of polylines.

The Properties palette is the main tool for viewing and modifying properties of ActCAD objects. You can enter commands and work in ActCAD while the Properties palette is open. When you select an object in the drawing area, the Properties palette displays the properties of that object. If you select multiple objects, the Properties palette displays all the properties they have in common.

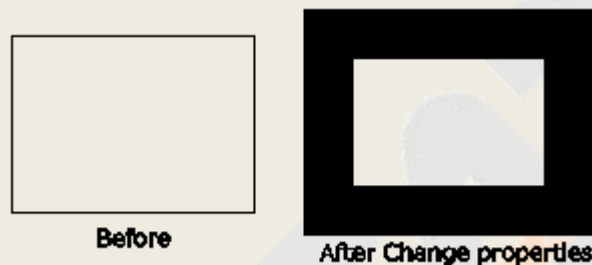
Menu: Modify -> Properties      Command: Properties      Aliases: entprop



**CHANGE:**

Changes the location, size, color, elevation, layer, linetype, linetype scale, lineweight, and three-dimensional thickness of selected entities. When you select a line, you can also change the location of the endpoints of the line. When you select a circle, you can also change the diameter of the circle. When you select a text entity, you can also change the insertion point, style, height, rotation angle, and the text of the entity. This command is especially useful for text entities.

Menu: Modify -> Change      Command: change      Aliases: Ch

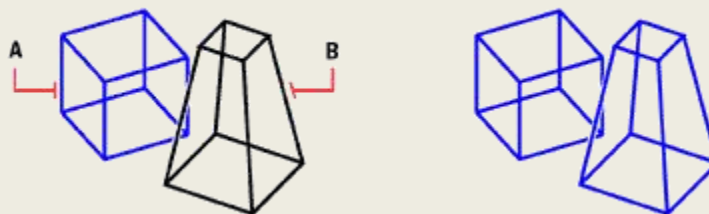


```
Command :change
Entities in set: 1
Change: Entities/Properties/<Change point>: p
Property to change: Color/Elevation/Layer/LType/LtScale/LWeight/Thickness: t
New thickness <0>: 20
Property to change: Color/Elevation/Layer/LType/LtScale/LWeight/Thickness: w
Unable to recognize entry. Please try again.
Property to change: Color/Elevation/Layer/LType/LtScale/LWeight/Thickness: lw
New Lineweight(-3 to 211) <-1>: 20
Property to change: Color/Elevation/Layer/LType/LtScale/LWeight/Thickness: '_help
Property to change: Color/Elevation/Layer/LType/LtScale/LWeight/Thickness:
```

**MATCH PROPERTIES:**

Menu: Modify -> Match properties      Command: Match Properties      Aliases: MA

Duplicates properties from one entity onto another. Only the properties are copied, not the entity itself. The properties copied are color, layer, linetype, thickness, text, dimensioning, and polyline width (if all segments of the source polyline are the same width).



```

Command : ma
Select entity to copy properties from:
Select entities to copy properties to:
Entities in set: 1
Select entities to copy properties to:

```

## CLIP:

### Image:

You can clip images so that only a portion of the image is visible in a drawing. The visible portion can be in the shape of a rectangle or polygon.

Image clipping can be turned on and off. If you turn off clipping for an image, the entire image is visible provided that the image is on a layer that is on and thawed. The clipping information is retained however, and you can turn clipping back on at any time.

Menu: Modify -> Clip->Image      Command: imageclip      Aliases: imageclip

```

Command: imageclip
Select image to clip:
Opposite corner:
On/Delete/<New>:
Command: _Redraw
Outside mode - Objects Outside boundary will be hidden.
Polygonal/Invert clip/<Rectangular>:p
First polygon point:
Pick polygon points, press Enter to close:
Pick polygon points, press Enter to close:

```

## DELETE:

Use to delete the objects from drawing.



Menu: Modify ->Delete      Command: Delete      Aliases: E

```

Command: e
FILter/<Select entities to delete>:
Opposite corner:
FILter/<Select entities to delete>:
Opposite corner:

```

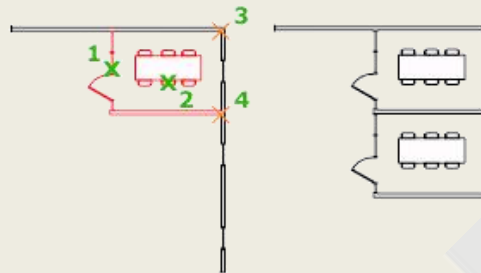
## COPY;

You can copy one or more entities, making one copy or multiple copies within the current drawing. You can also copy entities between drawings.

Menu: Modify ->Copy      Command: copy      Aliases: CP or CP

Use any of the following methods to copy entities within the current drawing:

- Create a copy at a location referenced from the original.
- Create a copy aligned parallel to the original.
- Create a copy as a mirror image of the original.
- Create several copies in a rectangular or circular pattern.



```
Command: cp
1 found
Current settings: Copy mode = Multiple
Specify base point or [Displacement/mOde] <Displacement>:
Specify second point or <use first point as displacement>:
Specify second point or <Exit>:
Specify second point or <Exit>:
```

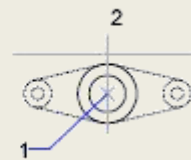
## MIRROR:

You can create a mirror image of an entity. You mirror the entity about a mirror line, which you define by specifying two points in a drawing. You can delete or retain the original entities.

Menu: Modify ->Mirror

Command: Mirror

Aliases: MI



```
Command : mi
Select entities to mirror:
Opposite corner:
Entities in set: 5
Select entities to mirror:
Start of mirror line:
End of mirror line:
Delete the original entities? <N> n
```

**OFFSET:**

You can use the offset feature to copy selected entities and align them parallel to the original entities at a specified distance. You can make parallel entities using arcs, circles, ellipses, elliptical arcs, lines, two-dimensional polylines, rays, and infinite lines.

Menu: Modify ->offset    Command: offset    Aliases: o



```
Command : offset
Parallel: ENTER for Through point/<Distance>: 20
Select entity:
Both sides/<Side for parallel copy>:
Select entity:
```

**ARRAY:**

You can copy an entity in a rectangular or polar (circular) pattern, creating an array. For a rectangular array, you control the number of copies in the array by specifying the number of rows and columns. You also specify the distance between each row and column. For a polar array, you control the number of copies that compose the array and whether to rotate the copies.

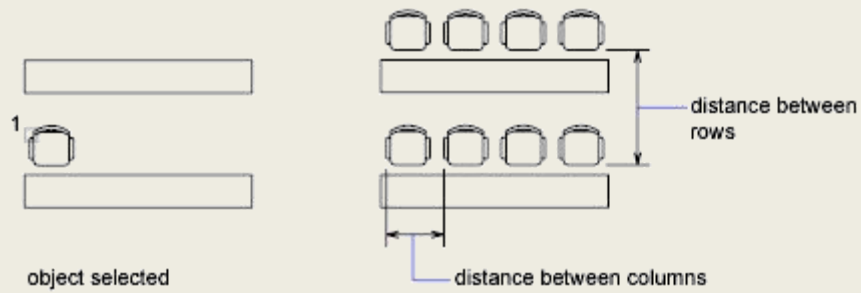
Menu: Modify ->Array    Command: Array    Aliases: AR

**Types Of Array:**

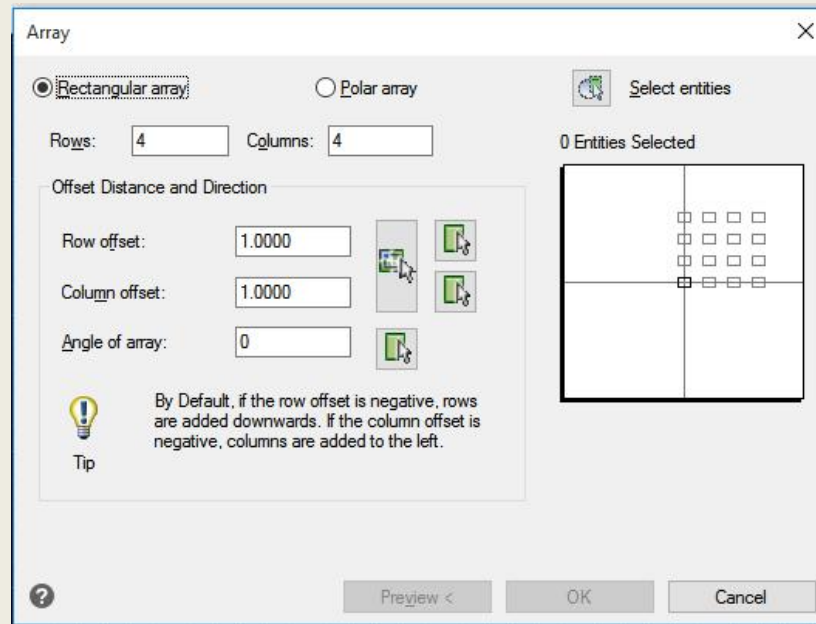
- Rectangular type
- Polar Array

**Rectangular Array:**

Creates an array of rows and columns of copies of the selected objects.



Rectangular Array Dialog Box:

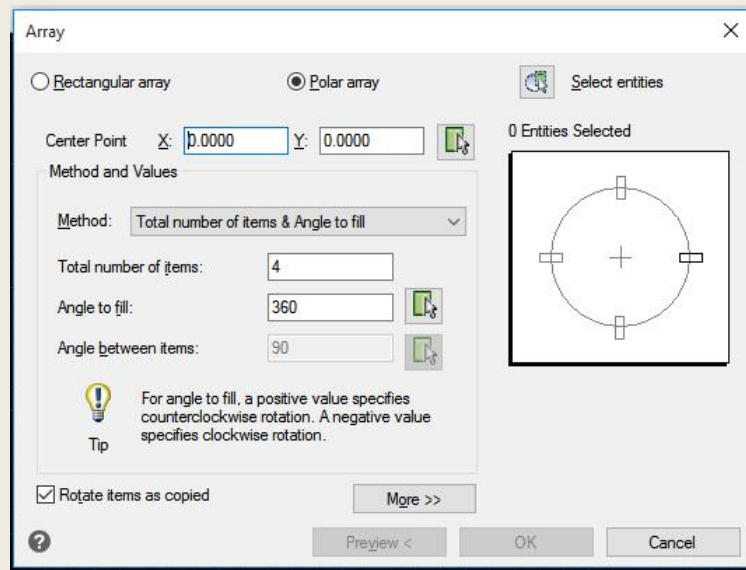


## Polar Array:

Creates an array by copying the selected objects around a specified center point.



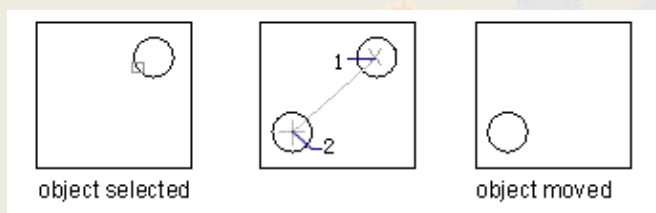
Polar Array Dialog Box:



## Move:

Moves the specified objects to a new place within a specified distance towards a specified direction.

Menu: Modify ->Move      Command: Move      Aliases: M

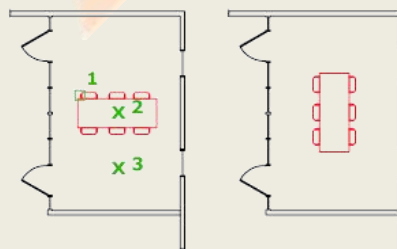


```
Command : move
Select entities to move:
Opposite corner:
Entities in set: 2
Select entities to move:
Vector/<Base point>:
Displacement point:
```

## Rotate:

Rotates objects about a specified point. The edges of the viewport are still parallel to the borders of the drawing area when you rotate an object on the viewport.

Menu: Modify ->Rotate      Command: Rotate      Aliases: RO



```
Command : rotate
Select entities to rotate:
Opposite corner:
Entities in set: 2
Select entities to rotate:
Rotation point:
Base angle/<Rotation angle>: 90
```

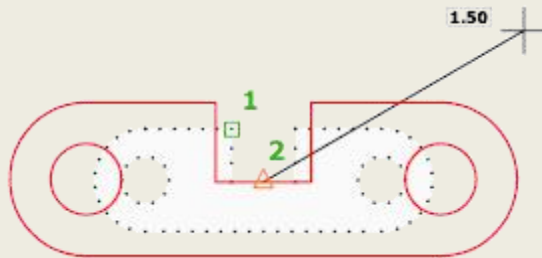
**SCALE:**

Enlarges or reduces selected objects, keeping the proportions of the object the same after scaling.

Menu: Modify ->Scale

Command: Scale

Aliases: SC



```
Command : scale
Select entities to scale:
Opposite corner:
Entities in set: 2
Select entities to scale:
Base point:
Base scale/<Scale factor>: 1.5
```

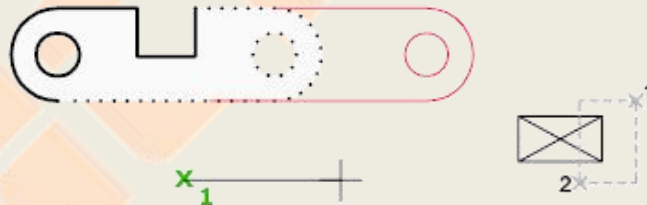
**STRETCH:**

It can changes the size of the object by stretching them in one direction. When you stretch entities, you must select the entities using either a crossing window or a crossing polygon. You then specify a displacement distance or select a base point and a displacement point. When we enter the stretch command in the commandbar it will displays as shown in below.

Menu: Modify ->Stretch

Command: Stretch

Aliases: S



```
Command : s
Select entities to stretch by crossing-window or crossing-polygon:
Opposite corner:
Entities in set: 2
Select entities to stretch by crossing-window or crossing-polygon:
Base point or displacement:
Second point of displacement:
```

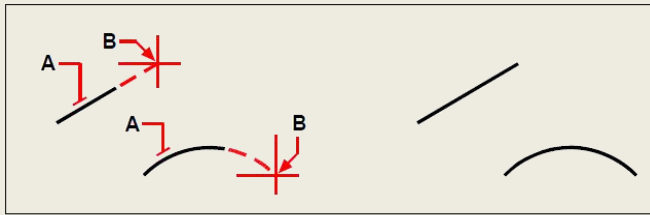
**EDIT LENGTH:**

You can change the length of entities or the included angle of arcs. Use any of the following methods to change the length of an entity:

- Dynamically drag the endpoint or angle.
- Specify an incremental length or angle measured from an endpoint.

- Specify the new length as a percentage of the total length or angle.
- Specify a new length or included angle.

Menu: Modify ->Edit Length      Command: Edit Length      Aliases: EditLen



Select the entity (A), and then select the new endpoint (B).

Result.

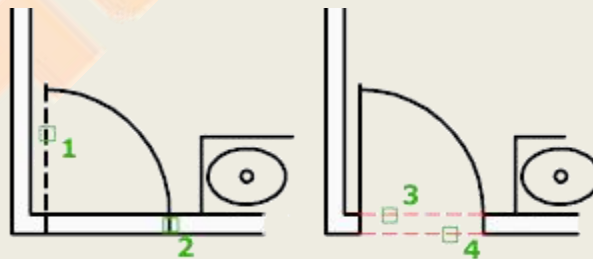
```
Command : _EDITLEN
Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>:
Current length: 297.704
Edit length: DYnamic/Increment/Percent/Total/<Select entity to list length>: dy
Mode/<Select entity to change>:
New end point for line:
Mode/Undo/<Select entity to change>:
```

## TRIM:

Trims objects to meet the edges of other objects.

Selects objects that define the cutting edges at which you want to trim an object, or press ENTER to select all objects as potential cutting edges. Valid cutting edge objects include 2D and 3D polylines, arcs, circles, ellipses, lines, layout viewports, rays, regions, splines, text, and xlines. TRIM projects the cutting edges and the objects to be trimmed onto the XY plane of the current user coordinate system (UCS).

Menu: Modify ->Trim      Command: Trim      Aliases: TR

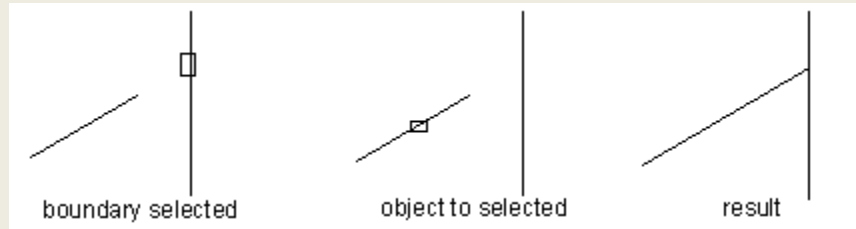


```
Command: trim
Select cutting entities for trim <ENTER to select all>:
Entities in set: 1
Select cutting entities for trim <ENTER to select all>:
Entities in set: 2
Select cutting entities for trim <ENTER to select all>:
Edge mode/Fence/Projection/Undo/<Select entity to trim or shift-select to extend>:
Edge mode/Fence/Projection/Undo/<Select entity to trim or shift-select to extend>:
```

**EXTEND:**

Extends objects to meet the edges of other objects.

Menu: Modify ->Extend      Command: Extend      Aliases: Ex

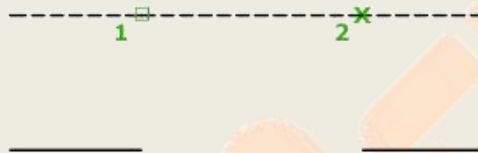


```
Command: extend
Select boundary entities for extend <ENTER to select all>:
Entities in set: 1
Select boundary entities for extend <ENTER to select all>:
Edge mode/Fence/Projection/Undo/<Select entity to extend or shift-select to trim>:
Edge mode/Fence/Projection/Undo/<Select entity to extend or shift-select to trim>:
```

**BREAK:**

Breaks the selected object between two points.

Menu: Modify ->Break      Command: Break      Aliases: BR

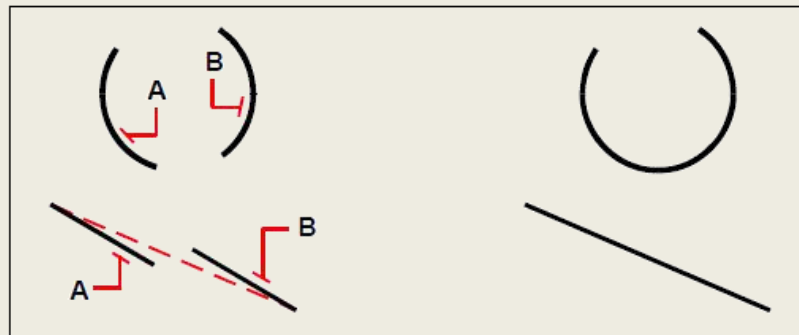


```
Command : br
Select entity to break:
First break point/<Second break point>: f
First break point:
Second break point: @100<0
```

**JOIN:**

Joins similar objects to form a single, unbroken object. You can join two entities into a single entity. You can join either two lines or two arcs. The two lines must be parallel; the two arcs must share the same center point and radius.

Menu: Modify ->JOIN      Command: Join      Aliases: JOIN



Select the first arc or line (A), and then select the second arc or line (B).

Result.

```
Command : join
Select arc or line for joining:
Select lines for attempted join:
Entities in set: 1
Select lines for attempted join:
1 object merged.
```

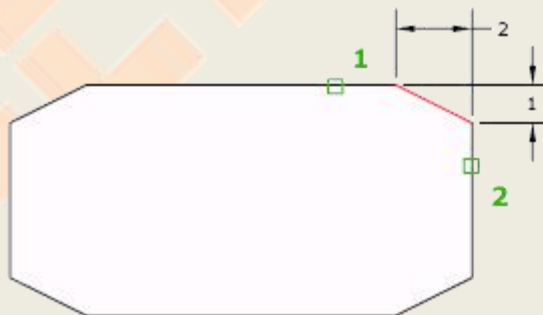
## CHAMFER:

Establishes chamfer on lines which are crossed, radiating or unlimited long. If the objects to be chamfered do not cross, the system will automatically trim or extend the object to make the chamfering possible.

Menu: Modify -> Chamfer

Command: Chamfer

Aliases: CHA

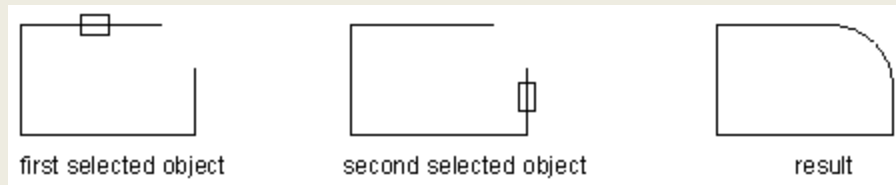


```
Command : cha
Chamfer (dist1=0, dist2=0): Distance/Settings/Polyline/<Select first entity>: d
Chamfer distance on first entity <0>: 20
Chamfer distance on second entity <20>: 25
Chamfer (dist1=20, dist2=25): Distance/Settings/Polyline/<Select first entity>:
Select second entity:
```

**FILLET:**

FILLET Creates fillets round the edges of two arcs, circles, elliptical arcs, lines, polylines, rays, splines, or xlines with an arc of a specified radius. FILLET also rounds the edges of 3D solids.

Menu: Modify ->Fillet    Command: Fillet    Aliases: F



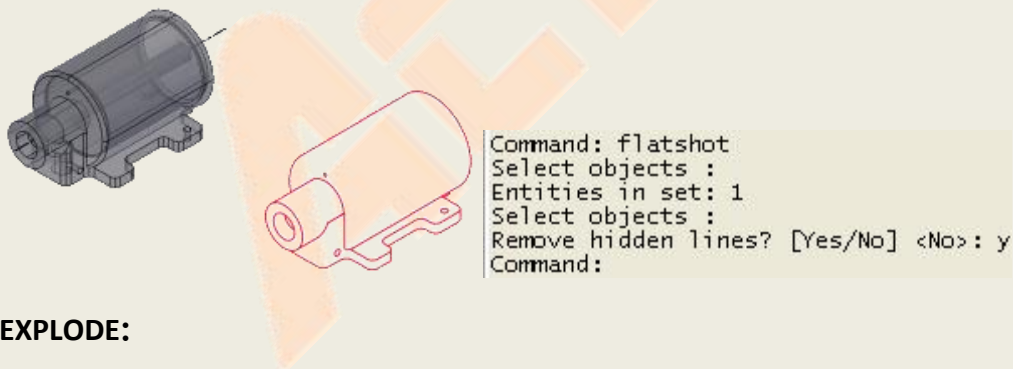
```
Command : f
Fillet (radius=0): Radius/Settings/Polyline/<Select first entity>: r
Fillet radius <0>: 20
Fillet (radius=20): Radius/Settings/Polyline/<Select first entity>:
Select second entity:
```

**FLATSHOT:**

Creates a 2D representation of all 3D objects based on the current view.

The edges of all 3D solids, surfaces, and meshes are projected line-of-sight onto a plane parallel to the viewing plane. The 2D representations of these edges are inserted as a block on the XY plane of the UCS. This block can be exploded for additional changes.

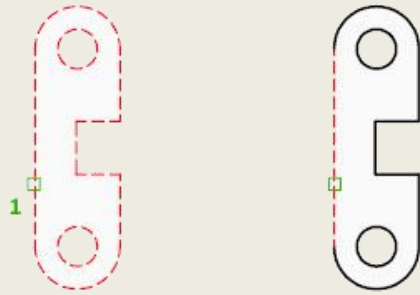
Menu: Modify ->Flatshot    Command: Flatshot    Aliases: Flatshot

**EXPLODE:**

Breaks a compound object into its component objects.

Explodes a compound object when you want to modify its components separately. Objects that can be exploded include blocks, polylines, and regions, among others.

Menu: Modify ->Explode    Command: Explode    Aliases: X

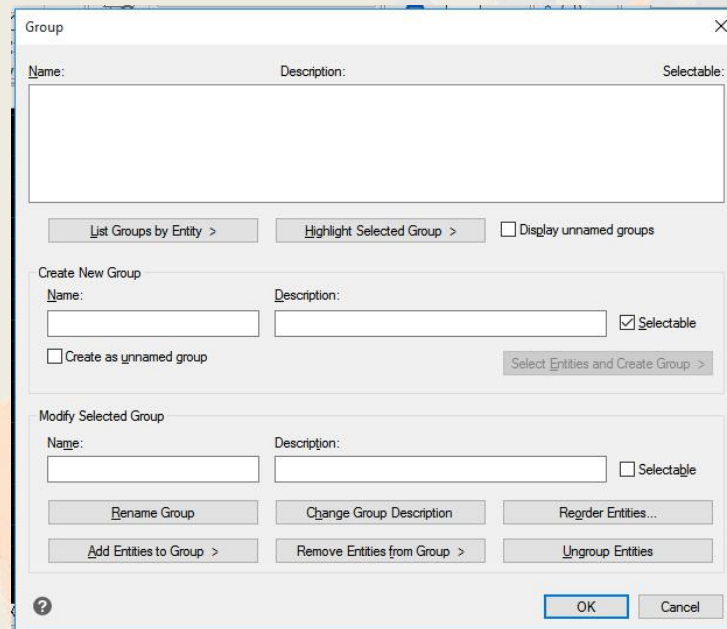


```
Command: _EXPLODE
Filter/<Select entities to explode>:
Entities in set: 1
Filter/<Select entities to explode>:
1 found
```

## GROUP:

A group is a collection of entities saved together that you can select and edit as a unit. Groups allow to easy combine drawing elements that you need to manage as a unit. For example, groups are useful in associating 3D solids when you do not want to combine them with a Boolean operation.

Menu: Modify ->Group    Command: Group    Aliases: Group



Group Dialog Box

## Creating groups:

When you create a group, you enter a group name and description, and then select the entities for the group.

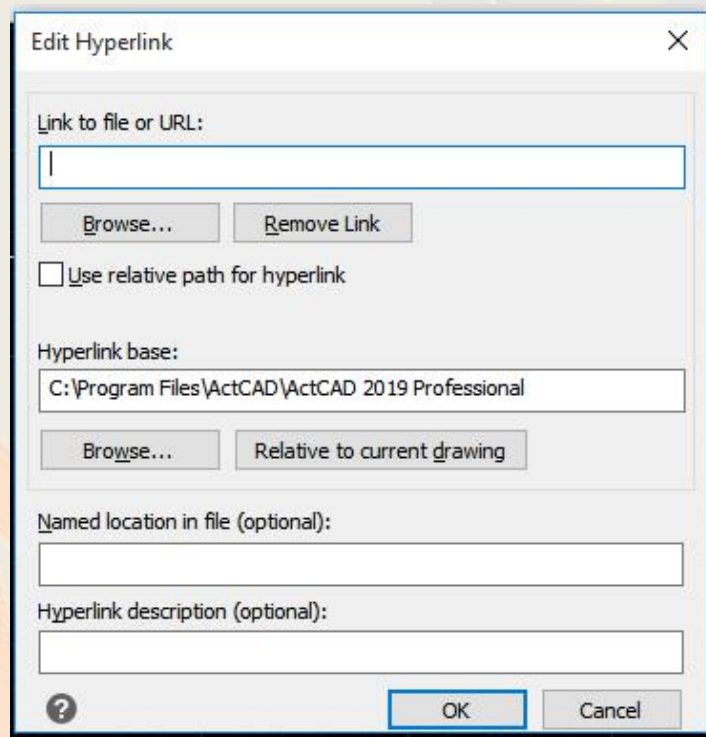
### To create a group

1. Do one of the following:
  - Choose Modify > Group.
  - Type group and then press Enter.
  - Click the button Groups ( ) and follow number 5

2. Under Create New Group, enter the name and description of the group.
3. Click Selectable if you want all entities in the group to be selected when you select one entity of the group in the drawing.
4. Click Select Entities and Create Group.
5. Select the entities for the group, and then press Enter.
6. In the Group dialog box, click OK.

## HYPERLINK:

In your ActCAD drawings, you can include hyperlinks, which are pointers that take you to another location, such as a Web address or a file on a particular computer. You can attach a hyperlink to any entity in your drawing. Then, when you select that entity, you can open the link and jump to the specific Web address or file location. You can create absolute hyperlinks, which store the full path to a file, or relative hyperlinks, which store a partial path relative to a base folder or URL.



## To create a hyperlink:

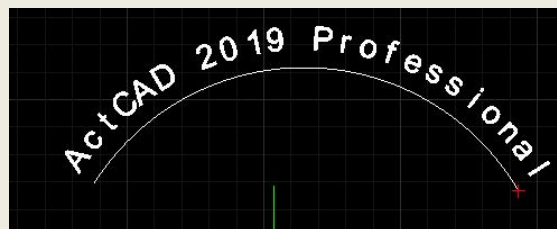
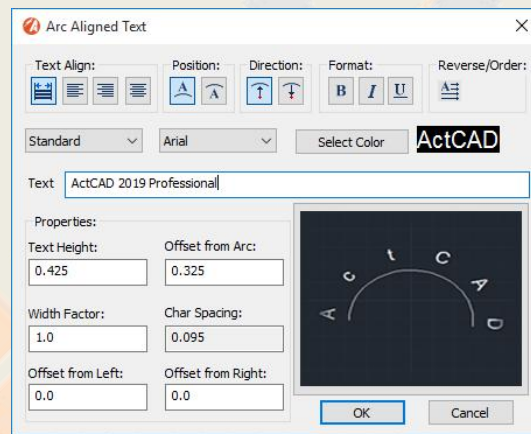
1. Type hyperlink and then press Enter.
2. Select an entity or entities that you would like to associate with a hyperlink; then press Enter to display the Edit Hyperlink dialog box.
3. Do one of the following:

- Click Browse to specify a file.
  - In the Link to File or URL box, type a Web address.
4. If you wish to use a common path for all hyperlinks in the drawing, check the Use relative path for hyperlinks box. Relative paths provide flexibility if you move files to a different folder, since you can change multiple hyperlink paths at once, rather than change them all individually.
  5. Click OK to close the Edit Hyperlink dialog box.

### Arc Aligned Text:

ActCAD has ARCTEXT command to create and edit curved text in your drawing. You can apply many text and alignment styles in this dialog. You can double click on existing ARCTEXT and change its parent arc apart from editing other parameters. A text preview is provided on the dialog to see the text before actual placement into drawing.

At the Command prompt, enter ARCTEXT or select ArcAligned on the Express Tools tab, Text panel. Select an existing arc. Enter the text in the ArcAlignedText Workshop – Create dialogbox. This creates a special ARCALIGNEDTEXT object.



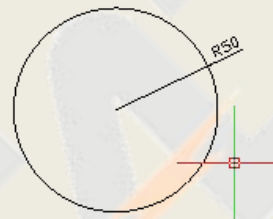
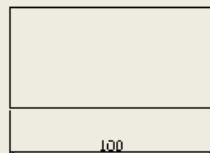
## Chapter 6: Dimension Commands

### QUICK DIMENSION:

To create and modify quickly a series of dimensions you can use the QDIM command. This command is particularly useful when you create a series of baseline or continued dimensions or when you dimension a series of circles and arcs.

Menu: Dimension -> Quick Dimension. Command: Quick Dimension

Aliases: Quick Dimension;

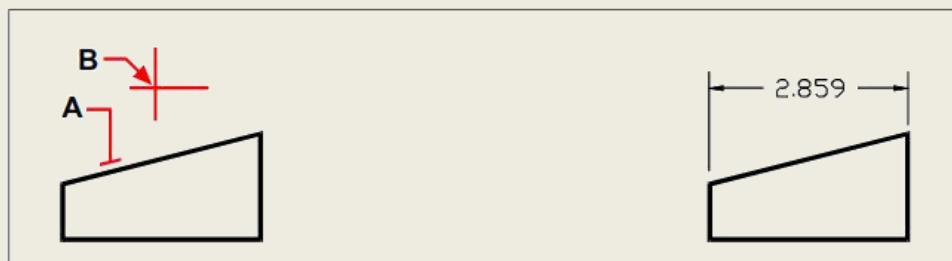


```
Command:
_QDIM
Filter/<Select entities>:
Entities in set: 1
Filter/<Select entities>:
Specify dimension line position, or
[Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/datumPoint/Edit] <Radius>:
```

### LINEAR:

Linear dimensions annotate linear distances or lengths and can be oriented horizontally, vertically, or aligned parallel to an existing entity or to the selected extension origin points. After you create a linear dimension, you can add a baseline dimension or a continued dimension. A linear baseline dimension inserts an additional dimension from a common first extension line origin of a previous linear dimension. A linear continued dimension continues a linear dimension from the second extension line of a previous linear dimension.

Menu: Dimension -> Linear. Command: Linear Dimension Aliases: DLI



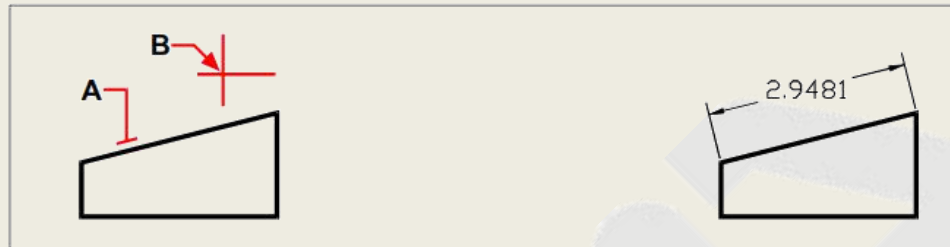
```

Command: _DIMLINEAR
ENTER to select entity/<Origin of first extension line>:
Origin of second extension line:
Angle/Mtext/Text/Orientation of dimension line: Horizontal/Vertical/Rotated:
Dimension Text = 2.859

```

## ALIGNED DIMENSION :

Menu: Dimension -> Aligned. Command: Aligned Dimension Aliases: AL



```

Command: _DIMALIGNED
ENTER to select entity/<Origin of first extension line>:
Origin of second extension line:
Angle/Mtext/Text/<Location of dimension line>:
Dimension Text = 2.948

```

## COORDINATE DIMENSIONS :

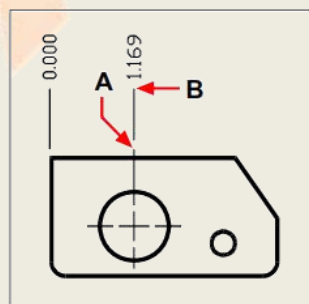
An ordinate dimension annotates the perpendicular distance from an origin or base Point (the origin of the current user coordinate system [UCS]). Ordinate dimensions consist of an x- or y-coordinate and a leader. An x-ordinate dimension measures distances along the x-axis; a y-ordinate dimension measures distances along the y-axis.

As you select ordinate points, the program automatically determines whether the Point is an x- or y-ordinate based on which direction you drag the second point. You Can Also specify whether the ordinate represents an x- or y-ordinate. Ordinate dimension text is always aligned with the ordinate leader lines, regardless of the text orientation specified by the current dimension style.

Menu: Dimension -> Coordinate

Command: Coordinate Dimension

Command: DOR



Select the ordinate point (A), and then specify the ordinate leader endpoint (B).

```

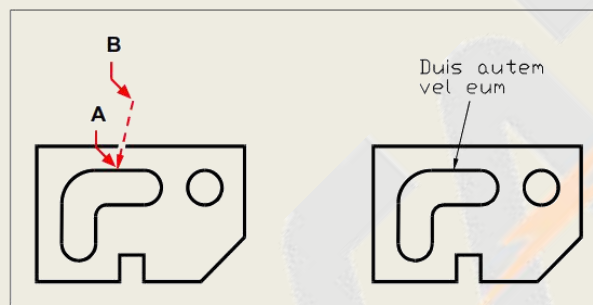
Command: _DIMORDINATE
Select point for ordinate dimension:
Text/Angle/X datum/Y datum/<Leader endpoint>:
Dimension Text = 0

```

## Leader:

Leaders consist of a line or series of lines that connects a feature in a drawing to an Annotation. Generally, you place an arrowhead at the first point. An annotation, created as dimension text, is placed immediately adjacent to the last point. By default, the text placed at the end of the leader line consists of the most recent dimension. You can also type an annotation as a single line of text.

Menu: Dimension -> Leader.    Command: Leader    Aliases: LE



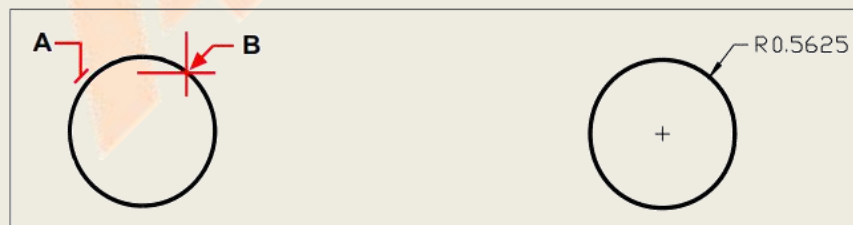
Specify the starting point of the leader (A) and the endpoint of the leader line segment (B).

Result.

```
Start of leader:
Next point:
To point: Format/Undo/<Annotation>:
To point: Format/Undo/<Annotation>:
First line of annotation text/<options>:
Dimension text options: Block/Copy/None/Tolerance/<Mtext>:
```

## RADIUS:

Menu: Dimension -> Radius    Command: Radius    Aliases: DR



Select the circle (A), and then specify the dimension line location (B).

Result.

```
Command: _DIMRADIUS
Select arc or circle:
Angle/Mtext/Text/<Location of dimension line>:
Dimension Text = R0.562
```

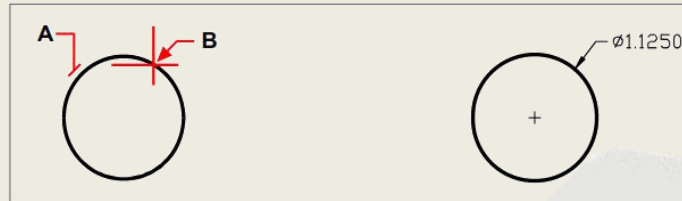
**DIAMETER:**

Diameter and radius dimensions annotate the radii and diameters of arcs and circles. You can optionally include centerlines or center marks.

Menu: Dimension -> Diameter

Command: Diameter

Aliases: DDI



Select the circle (A), and then specify the dimension line location (B).

Result.

```
Command: _DIMDIAMETER
Select arc or circle:
Angle/Mtext/Text/<Location of dimension line>:
Dimension Text = 1.125
```

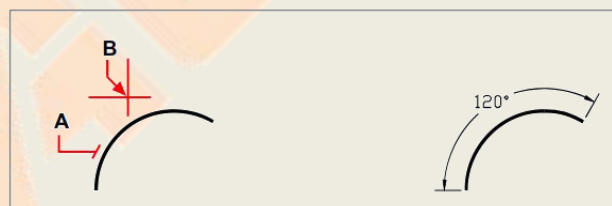
**ANGULAR:**

Angular dimensions annotate the angle measured between two lines. You can also dimension an angle by selecting an angle vertex and two endpoints. After you create an angular dimension, you can add a baseline dimension or a continued dimension. An angular baseline dimension inserts an additional dimension from a common first extension line origin of a previous angular dimension. An angular continued dimension continues an angular dimension from the second extension line of a previous angular.

Menu: Dimension -> Angular.

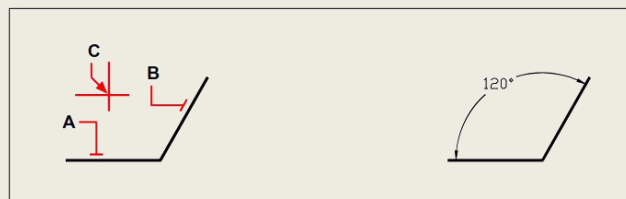
Command: Angular.

Aliases: DAN



To dimension the angle subtended by an arc, select the arc (A), and then specify the dimension arc location (B).

Result.



Select one line (A), select the other line (B), and then specify the dimension line location (C).

Result

```
Command: _DIMANGULAR
Press ENTER to specify angle/<Select line, arc, or circle>:
Other line for angular dimension:
Angle/Text/<Location of dimension line>:
Dimension Text = 120°
```

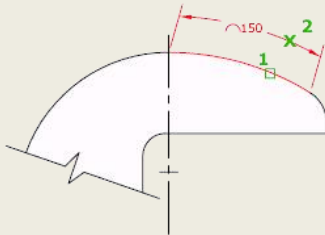
## ARC LENGTH:

Extension lines of an arc length dimension can be orthogonal or radial. An arc symbol is displayed either above or preceding the dimension text. Arc length dimensions measure the distance along an arc or polyline arc segment. The

Menu: Dimension -> Arc Length

Command: Arc Length

Aliases: Dimarc



```
Command: _DIMARC
Select arc or polyline arc segment:
Partial/Angle/Text/<Location of dimension arc>:
Dimension Text = 150
```

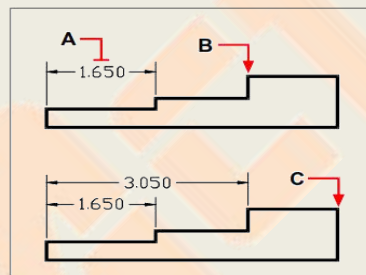
**BASELINE:** Creates a base line dimension from a specified Baseline

**Note:** Before you can use this procedure, you must first create a dimension

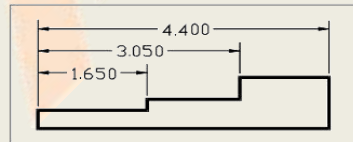
.Menu: Dimension -> Baseline.

Command: Dimbaseline.

Aliases: DBA



To add a baseline dimension to an existing linear dimension, select the existing dimension (A), select the next extension line origin (B), and select as many additional points as you want (C).

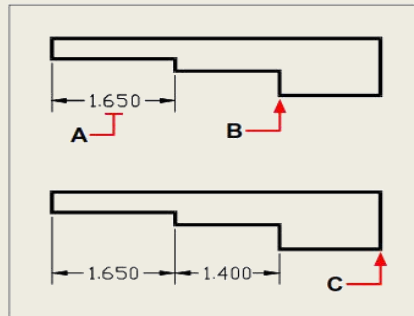


```
Command: _DIMLINEAR
ENTER to select entity/<Origin of first extension line>:
Origin of second extension line:
Angle/Mtext/Text/Orientation of dimension line: Horizontal/Vertical/Rotated:
Dimension Text = 1.65
Command: _DIMBASELINE
Baseline: ENTER to select starting dimension/Undo/<Origin of next extension line>:
Dimension Text = 3.05
Baseline: ENTER to select starting dimension/Undo/<Origin of next extension line>:
Dimension Text = 4.4
Baseline: ENTER to select starting dimension/Undo/<Origin of next extension line>:
```

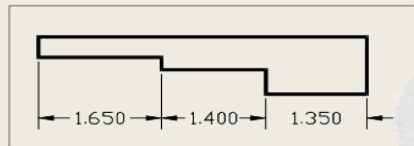
## Continue: Creates

Note: Before you can use this procedure, you must first create a dimension

Menu: Dimension -> Continue.      Command: Dimcontinue      Aliases: DCO



To add a continued dimension to an existing linear dimension, select the existing dimension (A), select the next extension line origin (B), and select another extension line origin (C).



```
Command: _DIMLINEAR
ENTER to select entity/<Origin of first extension line>:
Origin of second extension line:
Angle/Mtext/Text/Orientation of dimension line: Horizontal/Vertical/Rotated:
Dimension Text = 1.65
Command: _DIMCONTINUE
Continue: ENTER to select starting dimension/Undo/<Origin of next extension line>:
Dimension Text = 1.4
Continue: ENTER to select starting dimension/Undo/<Origin of next extension line>: and
```

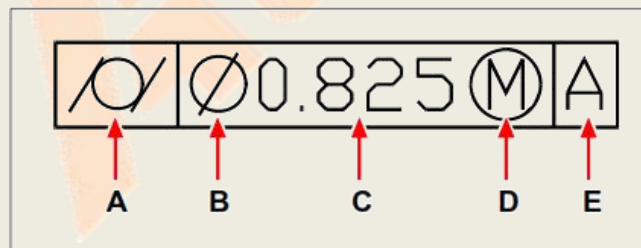
## GEOMETRIC TOLERANCE:

Geometric tolerances indicate the maximum allowable variations in the geometry defined by a drawing. ActCAD draws geometric tolerances using a feature control frame, which is a rectangle divided into compartments. Each feature control frame consists of at least two compartments. The first compartment contains a geometric tolerance symbol that indicates the geometric characteristic to which the tolerance is applied, such as location, orientation, or form. For example, a form tolerance may indicate the flatness or roundness of a surface.

Menu: Dimension -> Tolerance      Command: Tolerance      Aliases: TOL

- A Use this row to include composite tolerances.
- B Type a projected tolerance height value.
- C Type a datum identifier.
- D Click to include a projected tolerance symbol.
- E Click to select the material condition symbol for tertiary data.
- F Type the tertiary datum reference letter.
- G Click to select the material condition symbol for secondary datum.
- H Type the secondary datum reference letter.
- I Click to select the material condition symbol for primary datum.
- J Type the primary datum reference letter.
- K Click to select the material condition symbol for the second tolerance value.
- L Type the second tolerance value.
- M Click to include a diameter symbol for the second tolerance value.
- N Click to select the material condition symbol for the first tolerance value.
- O Type the first tolerance value.
- P Click to include a diameter symbol for the first tolerance value.
- Q Click to select a geometric tolerance symbol.

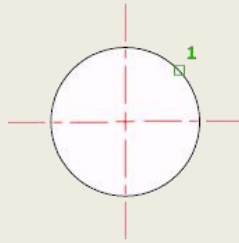
Command: tol  
 Select location for tolerance:  
 Substituting font [ic-gdt.shx] for font [gdt].  
 Substituting font [ic-gdt.shx] for font [gdt].



- A Geometric characteristic symbol.
- B Diameter symbol.
- C Tolerance value.
- D Material condition symbol.
- E Datum reference.

## CENTER MARK:

The default sizes of the center mark components can be set from the Dimension Style Manager, Symbols and Arrows tab, Center Marks (DIMCEN system variable).



You can choose between center marks and centerlines and specify their size when you set up the dimension style. See DIMSTYLE. You can also change center mark settings using the DIMCEN system variable.

Menu: Dimension -> Center Mark      Command: dimcenter

Aliases: dimcenter



center mark



centerlines

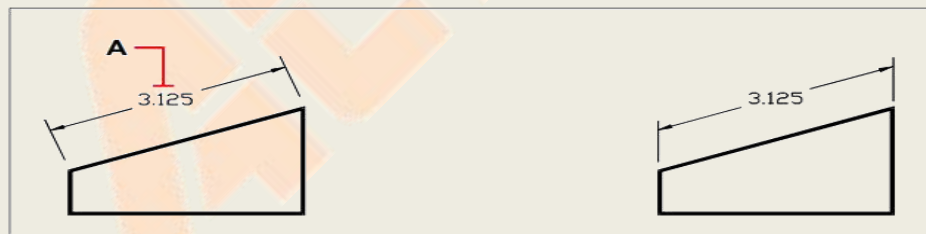
## MAKE OBLIQUE:

Extension lines are normally created at a perpendicular angle to the dimension line. You can change the angle of the extension lines, however, so that they tilt relative to the dimension line.

Menu: Dimension -> Make Oblique.

Command: Dimedit

Aliases: Dimedit



Select the dimension to be made oblique (A), and then type the obliquing angle.

Result.

```
Command: _DIMEDIT
Enter type of dimension editing: Home/New/Rotate/Oblique/<Home>: _OBLIQUE
Select linear dimensions to make oblique:
Entities in set: 1
Select linear dimensions to make oblique:
Enter obliquing angle: 30
```

## EDIT DIMENSION TEXT:

You can rotate the text of an existing dimension, move the dimension text to a new position, or replace selected dimension text with new text. You can also restore dimension text to its original position as defined by the current dimension style.

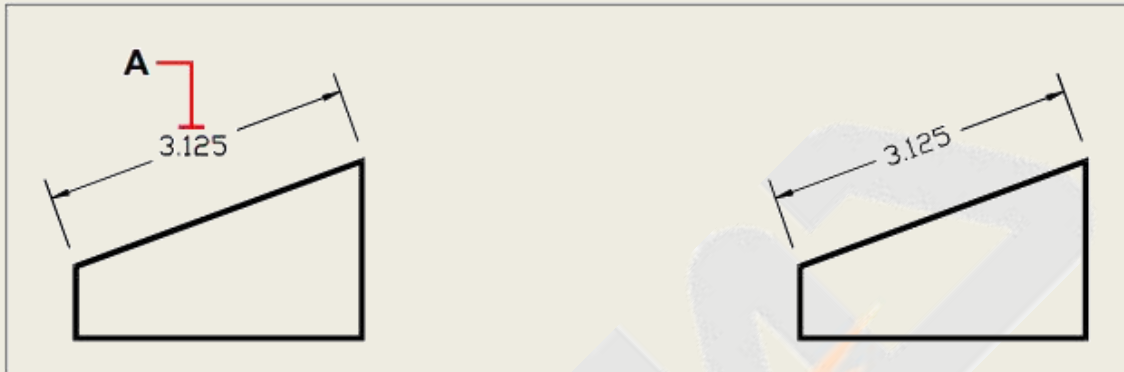
When you rotate or replace selected text, you specify the change first, and then select one or more dimensions to which to apply the change. All the selected dimensions are updated simultaneously.

### Rotate Dimension Text:

Menu: Dimension -> Rotate Dimension Text.

Command: Rotate Dimension Text.

Aliases: Dimedit



Select the dimension to be rotated (**A**), and then type the rotation angle.

Result.

```
Command: _DIMEDIT
Enter type of dimension editing: Home/New/Rotate/Oblique/<Home>: _ROTATE
Angle of dimension text: 30
Select entities to rotate text:
Entities in set: 1
Select entities to rotate text:
```

### Reposition Dimension Text:

To restore the dimension text to the original position, choose Restore. To change the angle of the dimension text, choose Angle. To change the left or right orientation of the dimension text, choose Left or Right.

Menu: Dimension -> Reposition Dimension Text

Command: DIMTEDIT

Aliases: DIMTEDIT

### Restore Text Position:

You can locate dimension text manually and specify its alignment and orientation.

Menu: Dimension -> Restore Dimension Text

Command: DIMTEDIT

Aliases: DIMTEDIT

**SAVE STYLE:** It allows you to save your current dimension style as a new dimension style

Menu: Dimension -> Save Style

Command: DIMSTYLE

Aliases: DIMSTYLE

```
Command: _DIMSTYLE
Enter a dimension style option
[ANnotative/Save/Restore/STatus/Variables/Apply/?] <Restore>: _SAVE
Enter name for new dimension style or [?]: jytra
```

**RESTORE STYLE:** It allows you to restore your dimension style from a group of dimension styles

Menu: Dimension ->Restore Style    Command: DIMSTYLE->Restore    Aliases: DIMSTYLE

```
Command: _DIMSTYLE
Enter a dimension style option
[ANnotative/Save/Restore/STatus/Variables/Apply/?] <Restore>: _RESTORE
? to list styles/ENTER to select dimension/<Dimension style>: jytra
```

**Override:** Controls overrides of system variables used in selected dimensions.

Menu: Dimension ->Override.

```
Command: dimoverride
Enter dimension variable name to override or [Clear overrides]: cen
DIMCEN is now <2.5>. New value: 10
```

## Chapter 7: Format Commands

### LAYERS:-

#### About Layers and its Applications:

You can use layers to control the visibility of objects and to assign properties to objects. Layers can be locked to prevent objects from being modified.

You can reduce the visual complexity of a drawing and improve display performance by controlling how objects are displayed or plotted. For example, you can use layers to control the properties and visibility of similar objects, such as electrical parts or dimensions. Also, you can lock a layer to prevent objects on that layer from being accidentally selected and modified.

#### Setting current layer:

Layers in ActCAD® are like the transparent overlays you use in manual drafting. You use layers to organize different types of drawing information. In ActCAD, each entity in a drawing exists on a layer. When you draw an entity, it is created on the current layer.

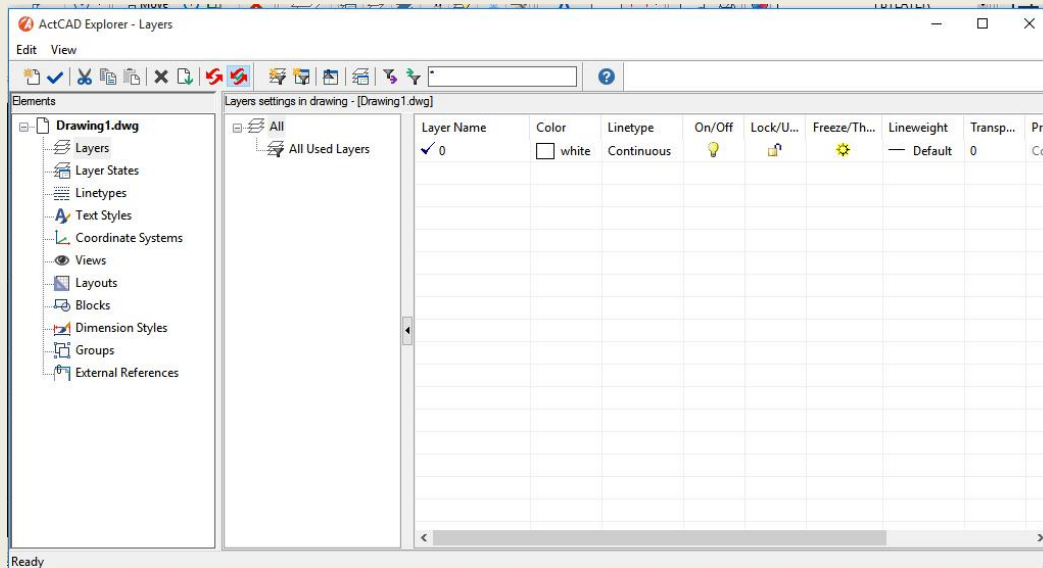
## Layer properties manager:

Layers are used to organize and classify the objects of a drawing. You can create and name a new layer for each homogeneous group of objects (for example walls, dimensions etc.) and then assign common properties to each layer. The organization of objects into layers allows you to control the visibility and properties of a large number of entities separately for each layer and it's useful to make changes quickly.

Menu: Format -> Layer.

Command: Layer.

Aliases: LA



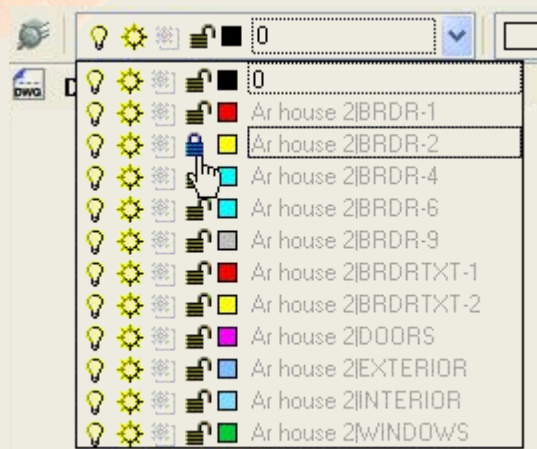
**Manage layers:** ActCAD allows you to manage layers in different ways like lock layer, Unlock layer, Hide layer

Menu: Format -> Layer.

Command: Layer.

Aliases: LA

**To Lock or Unlock Layer:** Select lock or unlock layer option from entity properties tool bar



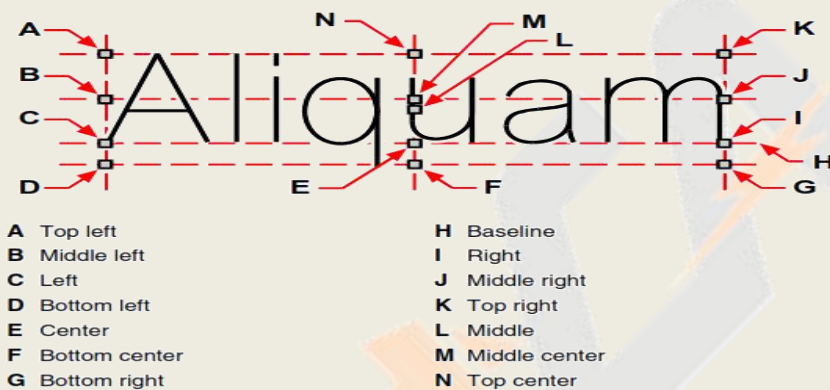
## Text Styles:

When you add text to a drawing, it uses the current text style. Text style determines the font, size, angle, orientation, and other characteristics. Every drawing has a default text style, named Standard, which initially uses the *icad.fnt* font. You cannot delete the Standard style, but you can rename it or modify it. You can change the font, the size of the font, and the oblique angle applied to it. If you change the font or orientation of an existing style, all existing text entities created using that style are automatically updated to reflect the new font or orientation. Changing any other characteristic has no effect on existing text. You also can create and use an unlimited number of additional text styles.

Menu: Format -> Text Styles.

Command: Text Styles.

Aliases: Text Styles.

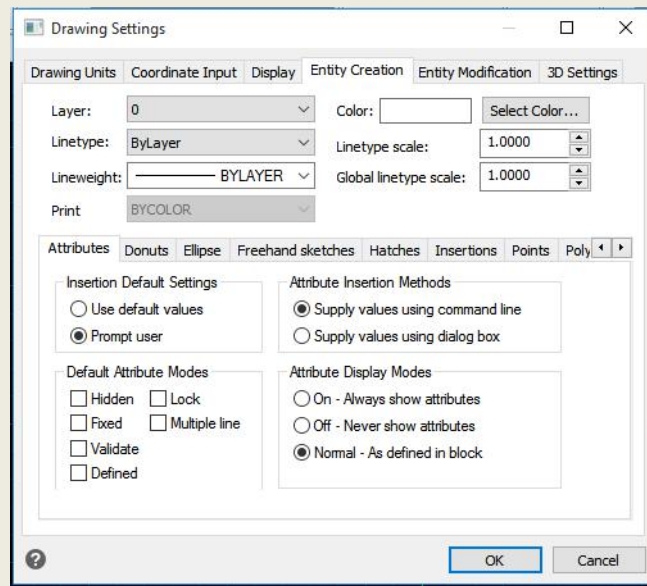


## Point Style:

Menu: Format -> Point Styles.

Command: Point Styles..

Aliases: PL.



```
Command: po
Settings/Multiple/<Location of point>: s
```

**Settings:** Displays the Point Style dialog box, in which you can specify the style and size for a point.

**Multiple:** Creates a series of point figures. The system prompts you repeatedly to specify a location before you enter ENTER:

<Location of point>: Specify a point or enter S

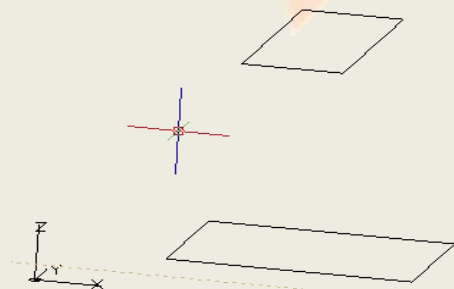
## Elevation:

The elevation of an entity is its z-coordinate position in relation to the xy plane in which the entity is drawn. An elevation of 0 indicates that the entity is drawn on the xy plane of the current UCS. Positive elevations are above this plane; negative elevations are below it.

Menu: Format ->Elevation.

Command: Elevation.

Aliases:ELEV.



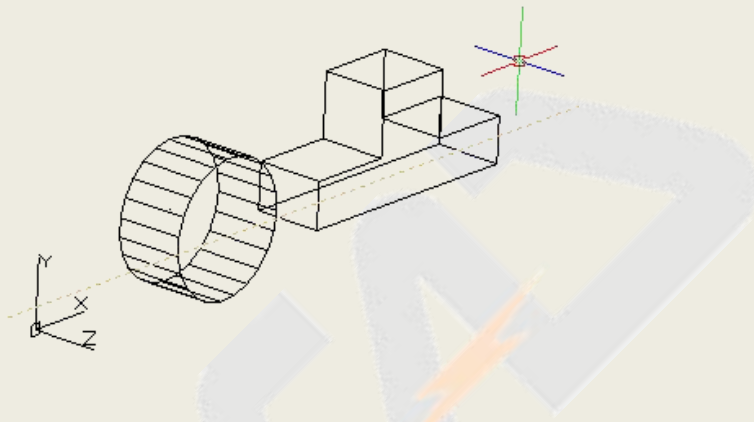
```
Command: elev
New current elevation <0.000000>: 100
New current thickness <0.000000>:
```

**Thickness:**

The thickness of an entity is the distance it is extruded above or below its elevation. A Positive thickness extrudes the entity upward in the positive z direction of the entity; a negative thickness extrudes it downward in the negative z direction. The thickness is applied uniformly to the entire entity. You can extrude any two-dimensional entity into a three-dimensional entity by changing the thickness of the entity to a nonzero Value. For example, a circle becomes a cylinder, a line becomes a three-dimensional plane, and a rectangle becomes a box.

Menu: Format ->Thickness.

Command: Thickness. Aliases:Thickness.

**Rename:**

Named objects are categories of items such as layers, dimension styles, table styles, text styles, and so on. It is strongly recommended that you create standard naming conventions for them and save them in drawing template files.

Menu: Format ->Rename.

Command: Rename.

Aliases:Rename

## Chapter 8.Insert Commands

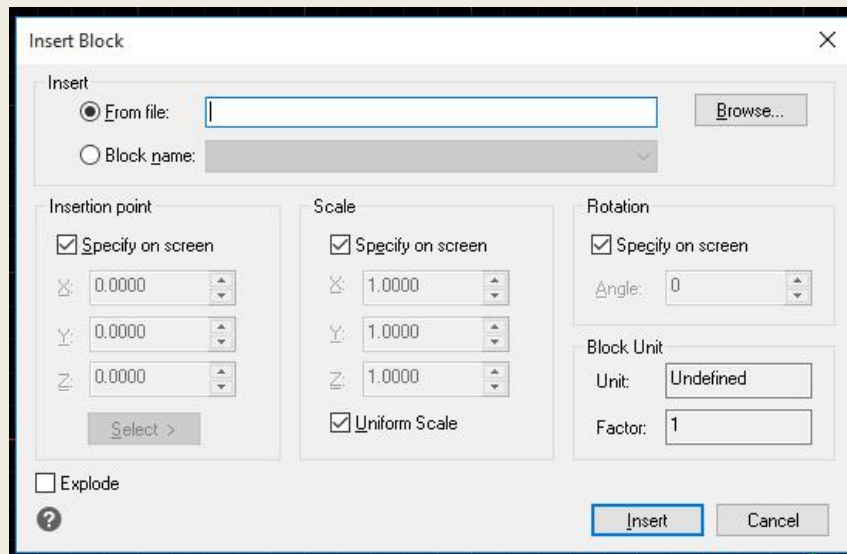
**INSERT BLOCK:**

Inserts blocks into current drawing file. In this you can specify

Menu: Insert->Block

Command: Insert

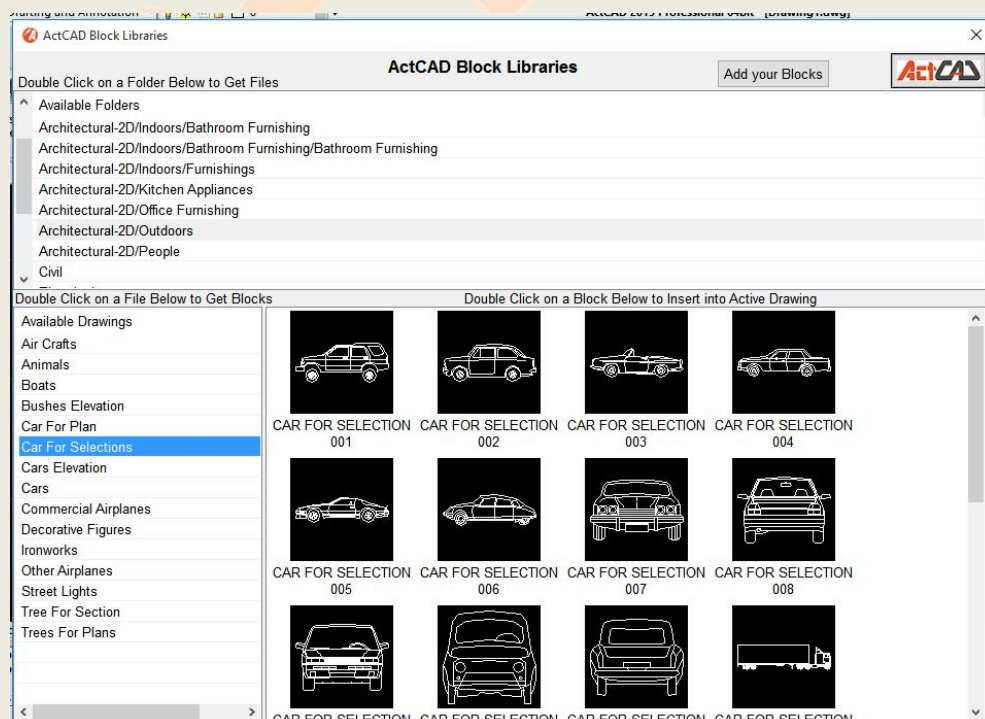
Aliases: I



Insert Block Dialog box

## BLOCK FROM LIBRARY-VARIOUS STANDARED BLOCKS AVIALABLE:

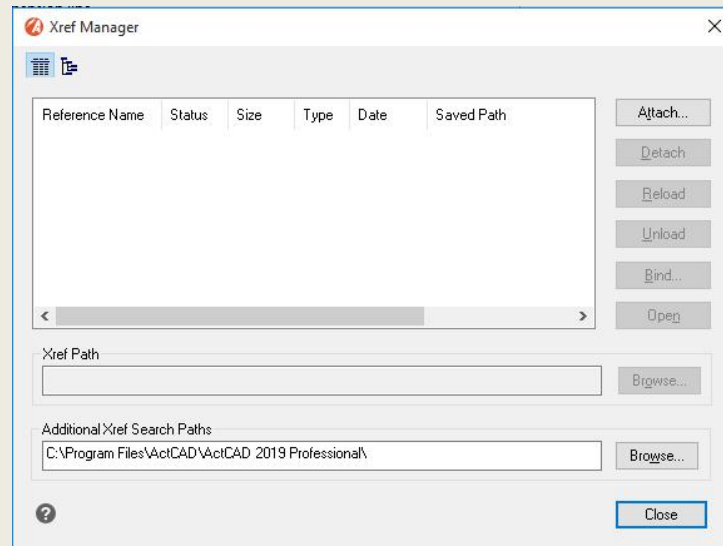
In the ActCAD Libraries section you find the default blocks already present when you install the software. These are organized in libraries (eg: Architectural, Electrical, Mechanical ...), categories (outdoors, indoors, people) and blocks. To insert a block into a drawing you need to select it and press the Insert icon or, simply, you can drag and drop it into the drawing.



## EXTERNAL REFERENCE:

You can attach another drawing to the current drawing as an external reference and insert a copy of the drawing into the current drawing. The name of the external reference drawing is added to the Block Name list. Changes made later to the referenced drawing will appear in this drawing when you reload the external reference.

Menu: Insert -> External Reference    Command: External Reference    Aliases: XA, -XR, XATTACH



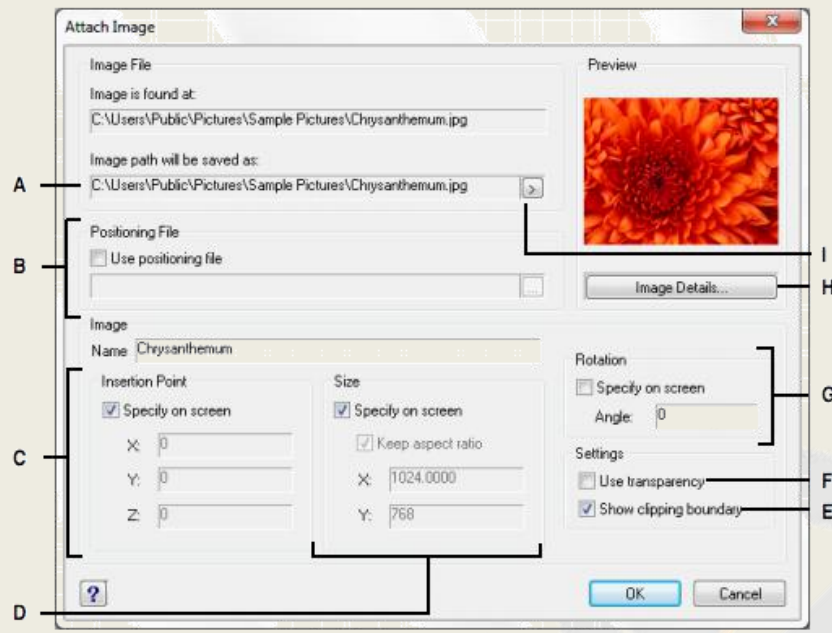
ExternalReference Dialog box

## ATTACH RASTER IMAGE:

When you attach an image to a drawing, the image displays in the drawing but is not saved in the drawing. The image file remains saved in its original location on your computer, network, or other media.

If you send or receive drawings that contain images, it is important to include with the drawing all of the image files attached to it. When you open a drawing that contains images, the source image files must be accessible for the images to display in the drawing.

Menu: Insert -> Attach raster image    Command: Image    Aliases: Image



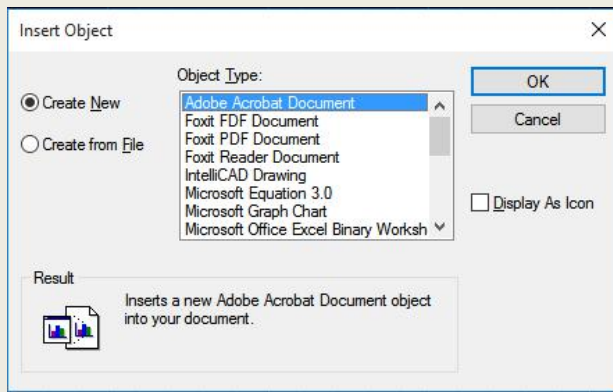
Attach Raster Image Dialog box

- A. Enter a different image file location, if necessary.
- B. Choose to place the image automatically in the drawing using specifications located in a positioning file. Enter the TWF file associated with the image or click [...] to select it.
- C. Choose to specify the insertion point in the drawing upon insertion, or enter coordinates.
- D. Choose to specify the size of the image in the drawing upon insertion or enter the size values.
- E. Choose whether to turn clipping display on or off for the image.
- F. Choose whether entities located under the image are visible (for images that support alpha transparency).
- G. Choose to specify rotation in the drawing upon insertion, or enter how many degrees to rotate the image to the left.
- H. Click to view image information, including color depth and resolution.
- I. Choose to save the image path as its full path, relative path to the current drawing folder, or file name in the current drawing folder.

#### OLE OBJECTS: (Object linking with embedding)

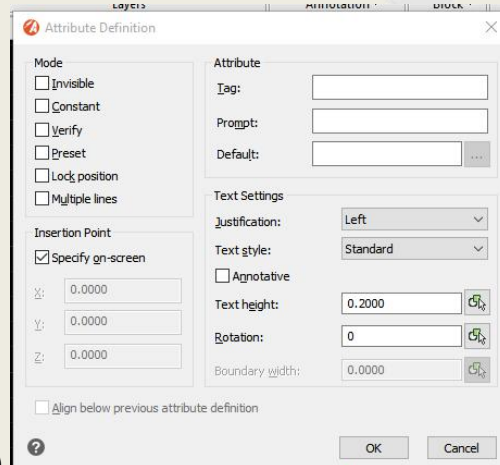
You can update a linked object automatically every time you open the drawing or you can do so only when you specify. Anytime a link is updated, changes made to the object in its original file also appear in the ActCAD drawing, and the changes also appear in the original file if they were made through ActCAD.

Menu: Insert -> OLE object    Command: OLE object    Aliases: IO



## DEFINE ATTRIBUTES:

You add an attribute to a drawing by first defining it and then saving it as part of a block definition. To define an attribute, you specify the characteristics of the attribute, including its name, prompt, and default value; the location and text format-ting; and optional modes(hidden, fixed, validate,



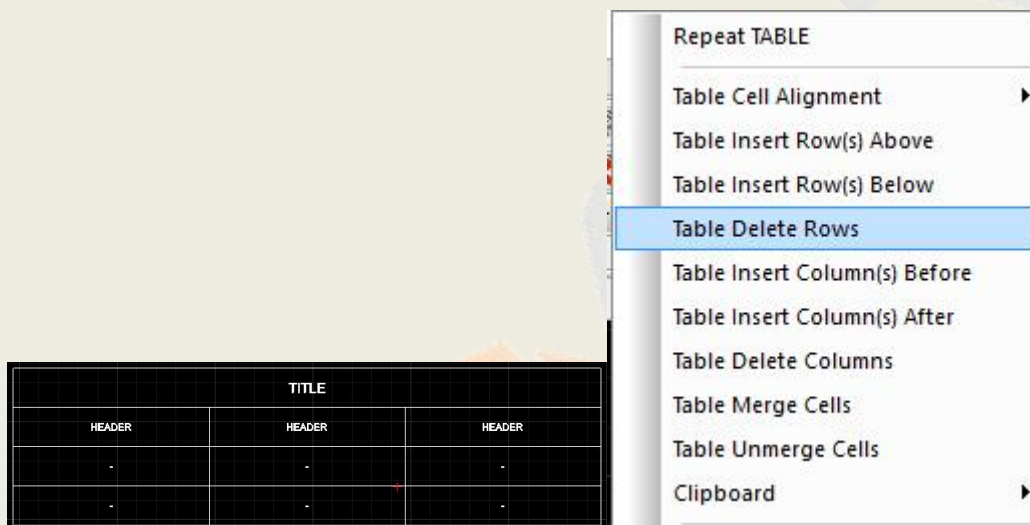
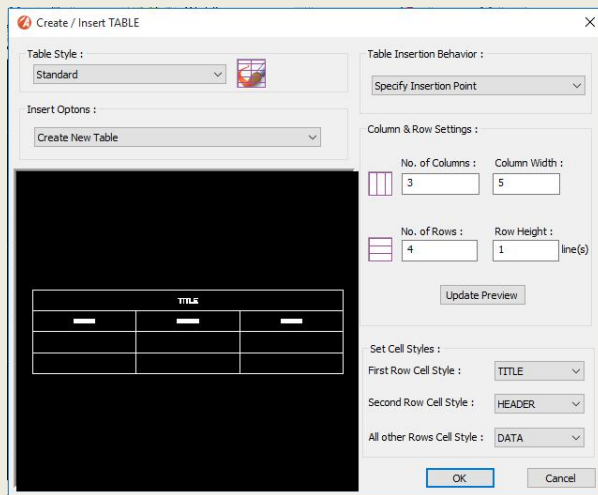
and predefined).

Menu: Insert-> DefineAttributes    command: Define Attributes    Aliases: -ATE

## Table:

In ActCAD you can create and edit tables very easily. There is a provision to import an external CSV to table and to export CAD table data to an external CSV file. This table is fully compatible with tables in other CAD software. You can perform many operations like insert/delete rows, insert/delete columns, merge cells, un-merge cells etc. You can change cell Background and Text Colors. Apart from these, many controls added to each cell like lock, unlock, format etc.

A table is a compound object that contains data in rows and columns. It can be created from an empty table or a table style.

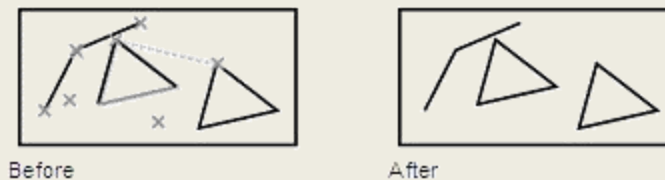


## Chapter 9.View Commands

### Redraw:

As you work on a drawing, visual elements may remain after the completion of a command. You can remove these elements by refreshing, or redrawing, the display.

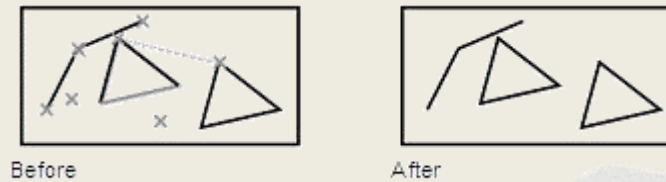
Menu: View -> Redraw Command: Redraw Aliases: R



## Regen:

Regenerates, or reproduces, the current viewports, and recalculates the screen location of all entities.

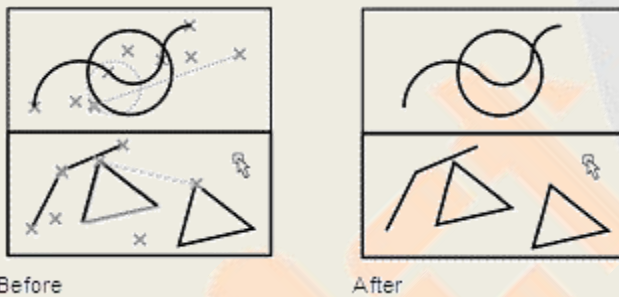
Menu: View -> Regen Command: Regen Aliases: RE



## Regen All:

Regenerates, or reproduces, the current viewports, and recalculates the screen location of all entities.

Menu: View -> Regen All Command: Regen All Aliases: REA



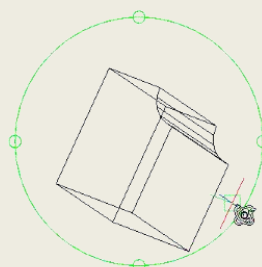
## REALTIME OPERATIONS:

### 3D Orbit:

ActCAD allows you to rotate the visualisation in real time to view your drawing from any angle while in model space. You cannot rotate the view while in paper space, but also within the layout windows.

If desired, you can continue the rotation of a view after you release the mouse button. Choose Tools > Options to enable continuous motion.

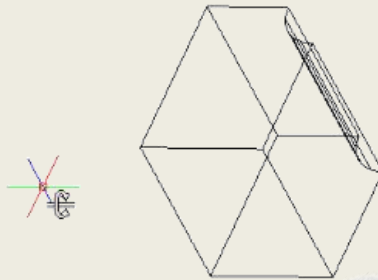
Menu: View -> 3D Orbit Command: 3D Orbit Aliases: RTROT (Ctrl+Left Mouse Button)



**Real-time x:**

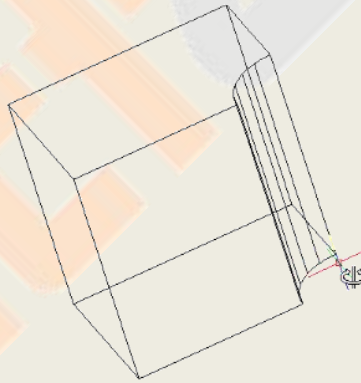
Allows you to rotate the view of entities in real time while maintaining the x-axis. You can rotate the view while in model space, not paper space.

Menu: 3D Orbit -> Real-time x Command: Real-time x Aliases: RTROT X

**Real-time Y:**

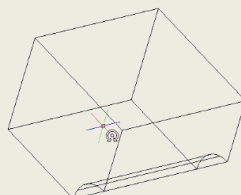
Allows you to rotate the view of entities in real time while maintaining the y-axis. You can rotate the view while in model space, not paper space.

Menu: 3D Orbit -> Real-time y Command: Real-time y Aliases: RTROT Y

**Real-time Z:**

Allows you to rotate the view of entities in real time while maintaining the z-axis. You can rotate the view while in model space, not paper space.

Menu: 3D Orbit -> Real-time z Command: Real-time z Aliases: RTROT Z



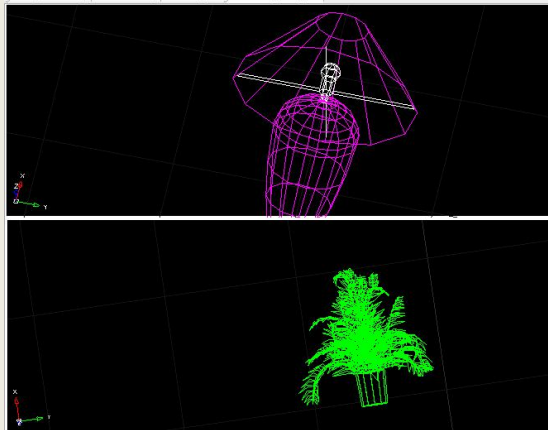
## View ports:

You can divide a single drawing window into multiple tiled windows (called view-ports) on the Model tab. You can control the number of windows created and the arrangement of the windows. You can also save and restore named window configurations and display a list of the current and saved window configurations.

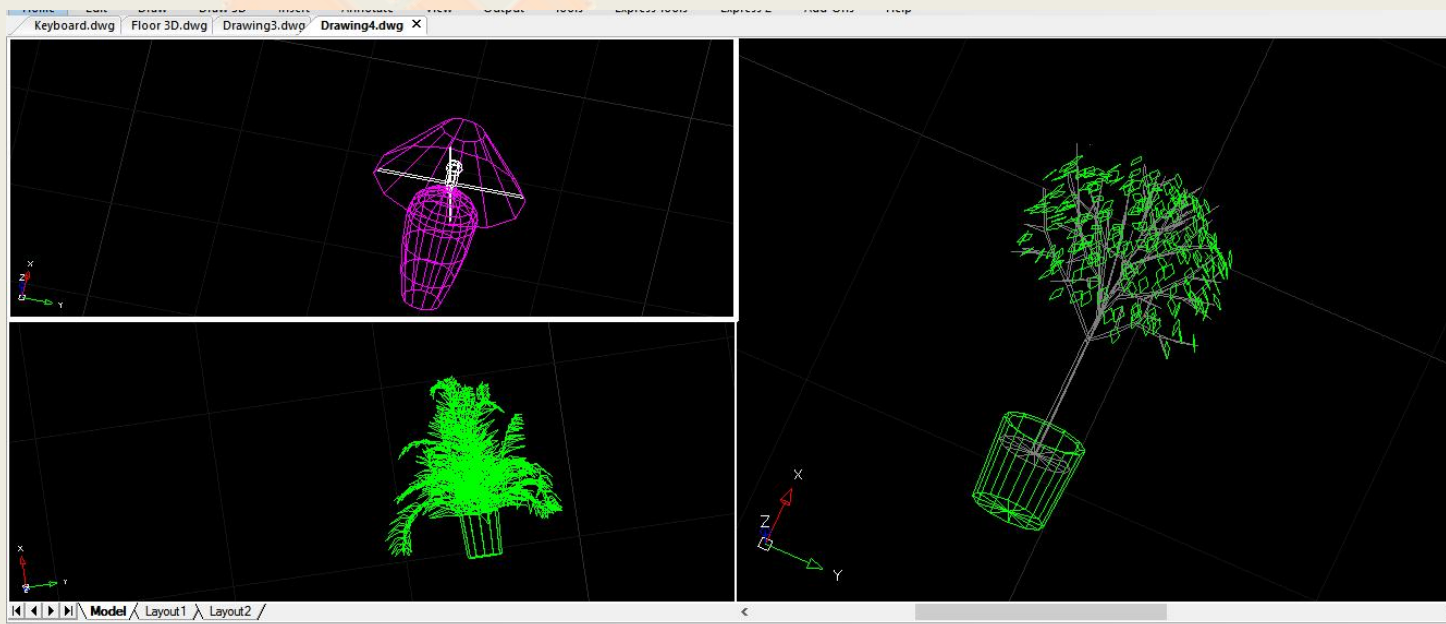
1. Creates one or more viewports in the current view.
2. You can create multiple viewports in paper space or model space using VPORTS.

Menu: View -> View ports Command: View ports Aliases: MV

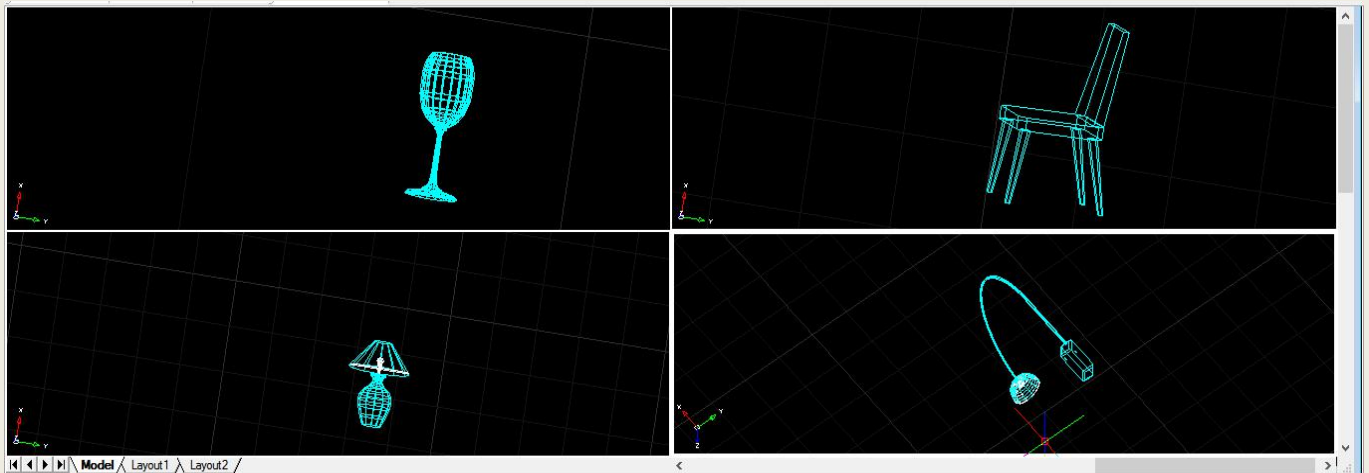
Divides a viewport into two equivalent viewports.



Divides a viewport into three viewports.



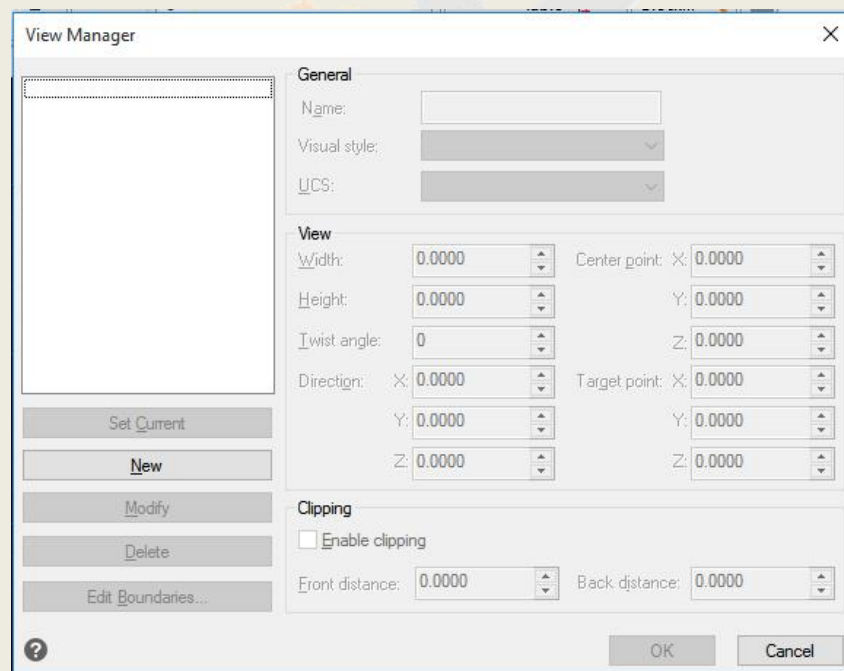
Divides a viewport into the four equivalent viewports.



### Named Views:

After you save one or more named views, you can restore any of those view in the current window using either the view command or the ActCAD Explorer.

Menu: View -> Named views    Command: Named views    Aliases: v



### 3D Views:

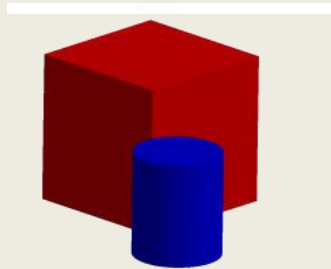
To quickly set a view, you can choose one of the predefined 3D views. You can select predefined standard orthographic views Top, Bottom, Front, Left, Right, and Back or SW (southwest) Isometric, SE (southeast) Isometric, NE (northeast) Isometric, and NW (northwest) Isometric views.

Menu: View -> 3D views Command: 3D view Aliases: Plan

### Orthographic views:

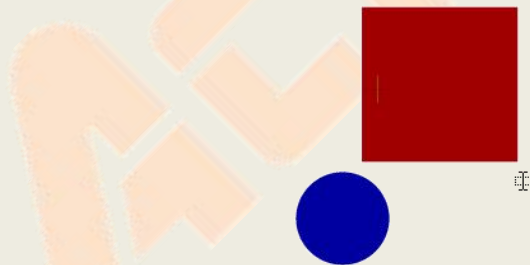
Orthographic views show objects from points of view that are perpendicular to XY, XZ and YZ plans, obtaining Top, Bottom, Front, Left, Right, and Back views.

**Draw a cube and a cylinder as shown in the example:**



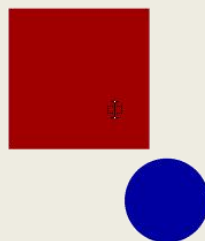
### Top view:

Menu: View -> 3D views Command: Top view



### Bottom view:

Menu: View -> 3D views Command: Bottom view

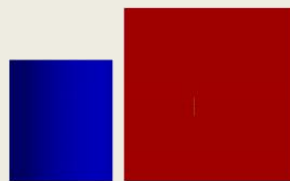


**Left side view:**

Menu: View -> 3D views Command: Left side view

**Right side view:**

Menu: View -> 3D views Command: Right side view

**Front side view:**

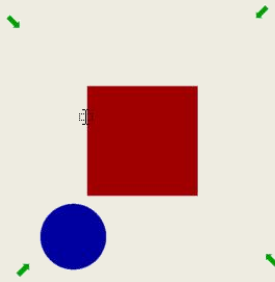
Menu: View -> 3D views Command: Right side view

**Back side view:**

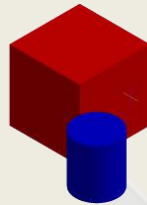
Menu: View -> 3D views Command: Right side view

**Isometric views:**

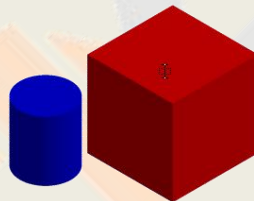
To understand how the isometric views work, look down at the top of a box. If you move toward the lower-left corner, you are viewing the box from the SW Isometric View. If you move toward the upper-right corner of the box, you are reviewing it from NE Isometric View.

**SW Isometric views:**

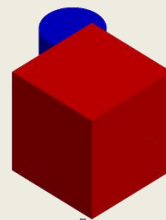
Menu: View -> 3D views Command: SW Isometric view

**SE Isometric views:**

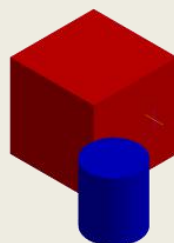
Menu: View -> 3D views Command: SE Isometric view

**NE Isometric views:**

Menu: View -> 3D views Command: NE Isometric view

**NW Isometric views:**

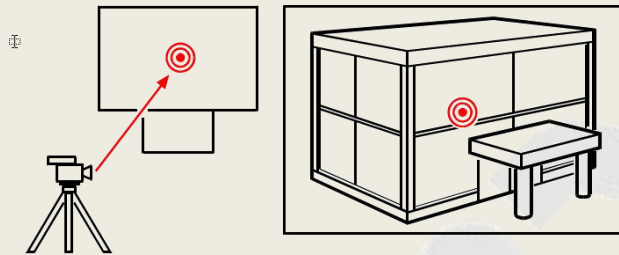
Menu: View -> 3D views Command: NE Isometric view



## Dynamic View:

Dynamically defines parallel or visual perspective views. Cameras and targets are used to dynamically view your three-dimensional drawing. You can specify the view-ing direction between the camera and the target. The centerpoint of the selected entities is considered the target.

Menu: View -> Dynamic View Command: Dynamic View Aliases: Dview

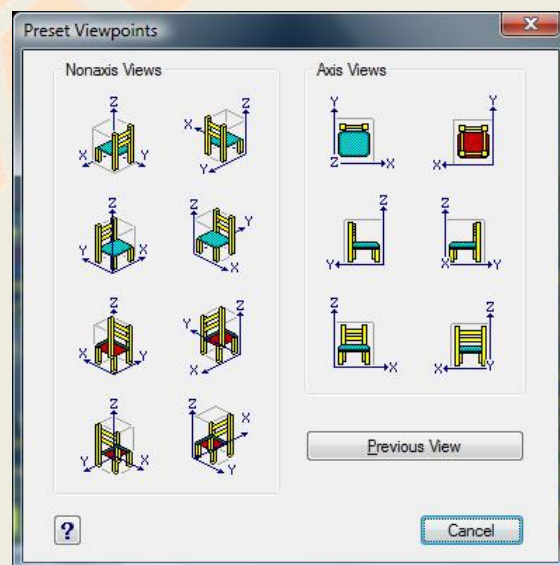


```
Command:
_DVIEW
ENTER for all entities/<Select entities to see in preview>:
Entities in set: 1
ENTER for all entities/<Select entities to see in preview>:
Enter option
[CAmera/TARget/DIstance/POints/PAN/Zoom/TWist/CLip/Hide/Off/Undo]: ca
Enter option
[CAmera/TARget/DIstance/POints/PAN/Zoom/TWist/CLip/Hide/Off/Undo]: ta
Enter option
[CAmera/TARget/DIstance/POints/PAN/Zoom/TWist/CLip/Hide/Off/Undo]: d
New camera-to-target distance <1.7321>: 100
```

## Viewpoint Presents:

To select a preset viewpoint you can also use the PresetViewpoints dialog box that contains preset views buttons formerly explained plus isometric views from under-neath the XY plane.

Menu: View -> Viewpoint presets Command: Viewpoint presets Aliases: Setvpint



- Click to select a non-axis viewpoint.
- Click to select the previous viewpoint.
- Click to select a viewpoint aligned with an axis.

## VISUAL STYLE:

Menu: View -> Visual Style   Command: Visual Style   Aliases:

## SHADE:

You can create a simple shaded picture of the objects displayed in the current viewport. Shading fills certain objects with a solid color and removes the display of lines that are located behind the shaded objects. Each object is shaded using its current color.

In any view, you can shade the following types of objects:

Objects with nonzero thickness

Circles

Solid-filled polygons

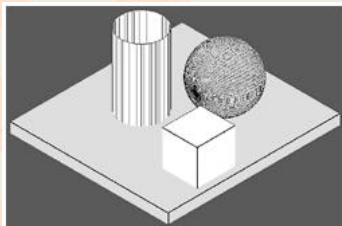
Wide polylines

Surfaces and 3D solids (when viewing models created in ActCAD)

Menu: Visual Style -> Shade

Command: Shade

Aliases: Sha



Shaded image

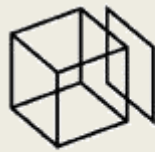
## HIDE:

Creates a hidden-line view of your model by removing all the lines that would be hidden behind other surfaces when seen from your vantage point.

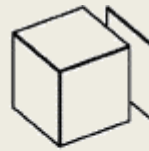
Menu: Visual Style -> Hide

Command: Hide

Aliases: HI



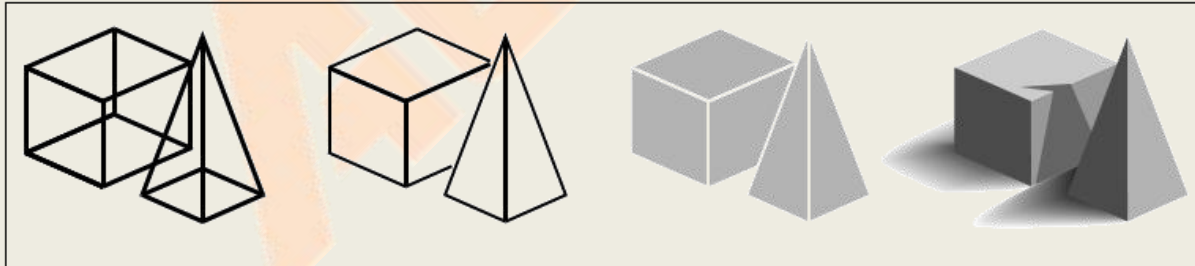
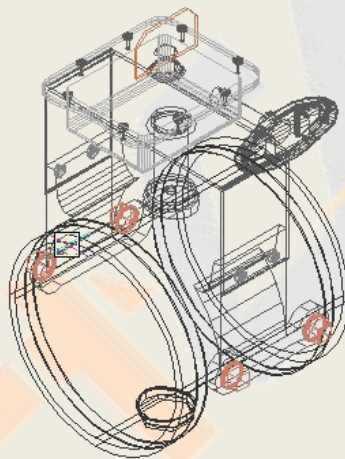
Before and after using the **Hide** command.



Result.

## WIRE FRAMES:

Wire-frame models, which consist of lines and curves that define the edges of a three-dimensional entity. You can create a wire-frame model by drawing lines, arcs, polylines, and other two-dimensional entities anywhere in three-dimensional space. Wire-frame models have no surfaces; they always appear as outlines. Because you must individually draw and position each entity that makes up a wire-frame model, creating one can be exacting and time-consuming.



Wire-frame model.

Hidden-line image.

Shaded image.

Rendered image.

## Chapter 10 Tools

### SPELL CHECK:

Corrects the spelling in text objects of a drawing.

It is possible to do the spell checking for any text of a drawing, including:

- Single line text
- Multiline text
- Text in block attributes
- External references

For attributes, only values are checked.

To do the check spelling in another language, you can choose a different dictionary.

You can also customize and manage several dictionaries and switch between them.

To do the check spelling

Do one of the following:

- Choose Tools > Spell Check.
- Type spell and then press Enter.

Menu: Tools-> Spell Check    Command: Spell Check    Aliases: SP

### MAIN DICTIONARY:

The check spelling function allows to choose a main dictionary among many different languages. Dictionary files have a .dic extension. To do the check spelling in another language, you can switch to a different main dictionary.

Do one of the following:

- From the Main Dictionary box, select another dictionary in a language that you need.
- With the Download button you can download a new dictionary.

The download button is linked to a web page that manages a lot of free downloadable dictionaries in many different languages. Put downloaded dictionaries in the same folder shown in the Custom Dictionary box.

**DRAW ORDER:**

The draw order commands allow you to arrange the images and entities in your drawing. When multiple entities overlap, you can change the order in which they are displayed and printed. You can move entities to the front, back, or on top or below of another entity.

Menu: Tools-> Draw Order Command: Draw Order Aliases: Draw Order  
To reorder entities

1 Do one of the following:

- Choose Image > Draw Order.
- Type draworder and then press Enter.

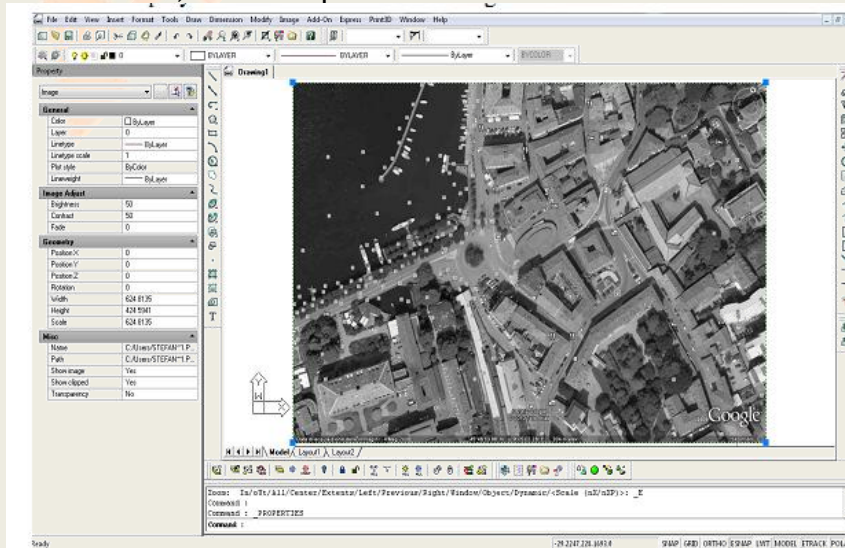
2 Select the entity you want to reorder, and then press Enter.

The following suggestions may be of help:

- Once placed, you can use the ActCAD 'move' and 'rotate' commands to refine your model's position relative to the image. Do not adjust the position of the Google Earth's image. For details, see Chapter 10, "Rearranging entities."
- Use the Draw Order command to send an image below.
- The image is black and white - a current limitation imposed by Google Earth.

**To use Draw Order command**

1. Do one of the following:
  - Choose Tools > Draw Order.
  - On the Tools toolbar, click the Draw Order tool
  - Type draw order and then press Enter.
2. Select the entity you want to reorder, and then press Enter.
3. In the prompt box, specify the new drawing order, and then press Enter.
4. If you are reordering above or under, select the entity you want the first entity to be above or below, and then press Enter.



**INQUIRY:**(Distance, Area, List, Id Point, Mass Property, Drawing Status, Time,Status, Set Variable Calculator)

### Distance:

Menu: Tools-> Distance Command: Distance Aliases: Dist

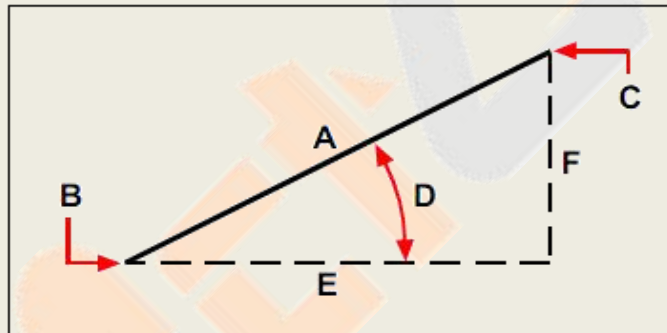
To calculate the distance between two points and their angle

1. Do one of the following:
  - Choose Tools > Inquiry > Distance.
  - On the Inquiry toolbar, click the Distance tool ( ).
  - Type distance and then press Enter.
2. Specify the first point.
3. Specify the second point.

The following type of information is displayed:

Distance = 13.2850, Angle in XY Plane = 31°, Angle from XY Plane = 0°

Delta X = 11.3878, Delta Y = 6.8418, Delta Z = 0.0000



Use the Distance command to calculate the distance (A) between two points (B and C), the angle in the xy plane (D), the angle from the xy plane, and the delta x (E), delta y (F), and delta z distances between the two points.

### LIST:

Menu: Tools-> List Command:list Aliases: li

To display information about an entity

Advanced experience level

1. Do one of the following:
  - Choose Tools > Inquiry > List.
  - On the Inquiry toolbar, click the List tool ( ).
  - Type list and then press Enter.
2. Select one or more entities.

3. Press Enter.

NOTE To return to the drawing window, press F2.

The following type of information is displayed:

```

Command: LIST
SCroll/Sort/Tracking/<Select entities to list>:
Entities in set: 1
SCroll/Sort/Tracking/<Select entities to list>:
----- 3DSOLID -----
Handle: 1423
Current space: Model
Layer: KEYBOARD
Color: BYLAYER
Linetype: BYLAYER
Line weight: BYLAYER
History: Record
Show History: No
Bounding Box:
Lower Bound: X = 2'-5 5/8" Y = 0'-1 3/16" Z = 1/2"
Upper Bound: X = 3'-11 1/4" Y = 0'-8 1/16" Z = 0'-1 7/8"

```

## DRAWING STATUS:

Menu: Tools-> Drwing status Command: Drawing status Aliases: Drwing status

You can display information about the current status of a drawing, including:

- Drawing name.
- Limits.
- Insertion base point.
- Snap and grid settings.
- Current layer, color, and linetype.
- Current settings for various modes (fill, grid, orthogonal, snap, blips, and so on).

To display the drawing status

Advanced experience level

1. Do one of the following:
  - Choose Tools > Inquiry > Drawing Status.
  - Type status and then press Enter.

The following type of information is displayed:

```

Command: STATUS
Status:
  Current drawing name:
  Drawing limits are: X= 0.000000 Y= 0.000000 Z= 0.000000
                    X= 12.000000 Y= 9.000000 Z= 0.000000
  Paper space limits are: X= 0.000000 Y= 0.000000 Z= 0.000000
                       X= 12.000000 Y= 9.000000 Z= 0.000000
  Screen width (pixels): 1337
  Screen height (pixels): 425
  Insertion base is: X= 0.000000 Y= 0.000000 Z= 0.000000
  Snap resolution is: X= 0.500000 Y= 0.500000 Z= 0.000000
  Grid spacing is: X= 0.500000 Y= 0.500000 Z= 0.000000
  Current layer: 0
  Current color: BYLAYER
  Current linetype: ByLayer
  Current elevation: 0.000000
  Current thickness: 0.000000
  Current Lineweight: BYLayer
  Fill: on
  Grid: on
  Ortho: off
  Snap: off
  Blips: off
  Drag: on
  LWeight: off
  Command echo: on
  Positive angle direction: Counterclockwise
  Angular units: Decimal degrees
  Dimension units: Decimal
  Pickbox height: 3
4 found
Entities in drawing: 4

```

## TIME:

Menu: Tools-> TIME Command:TIME Aliases: TI

To display the timer information

Advanced experience level

1. Do one of the following:
    - Choose Tools > Inquiry > Time Variables.
    - Type time and then press Enter.
  2. Select any of the prompt box options:
  3. Choose Timer On to turn the elapsed timer on.
- Choose Timer Off to turn the elapsed timer off.

In the prompt box, choose Cancel to exit the command.

Each time you display the timer information, the following type of information is displayed:

```

Command: ti
The current time is: Thu May 31 2012 11:16:35AM
Times for this drawing:
  Drawing was created on: Thu May 31 2012 01:54:24PM
  Drawing was last updated on: Thu May 31 2012 11:09:18AM
  Total editing time: 0 days 02:52:14:187
  Elapsed timer (on): 30 days 11:16:35:811
ON/OFF/Display/Reset:

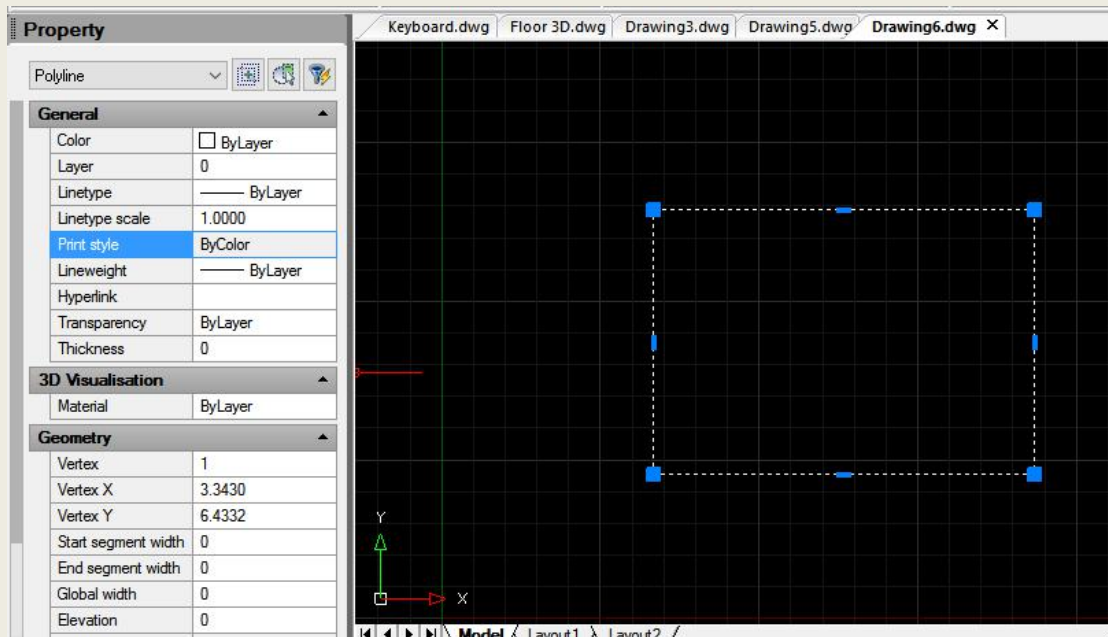
```

## PROPERTIES:

The Properties palette can be docked in the drawing area. You can Hide the palette

And give it a transparency effects.

Menu: Tools-> Properties Command: Properties Aliases: Properties(Ctrl+1)



- Enter a new value.

Height	17.400451
Text	This is a test
Rotation	0.0000

- Select a value from a list.


horizontal alignment	Center
Alignment point	Aligned
vertical alignment	Center
<b>Geometry</b>	Fit
Insertion point	Left
	Middle

- Change the property value in a dialog box.

Height	17.400451
Text	This is a test
Rotation	217.5368

- Use the Pick Point button to change a coordinate value.

<b>Geometry</b>	
Start point	248.68,129.67,0.00
End point	330.53,220.21,0.00



The properties common to the selected objects are displayed in the Properties palette. You can then modify the properties of the selected objects in the Properties palette or you can make other changes to the selected objects by entering an editing command.

To modify the properties from Properties Palette

1 Do one of the following:

- Choose Tools > Properties.
- On the Standard toolbar, click the Properties tool ( ).
- Type properties and then press Enter.

2 Select the entities, and then press Enter.

3 Make changes to the properties you want to change, and then click OK

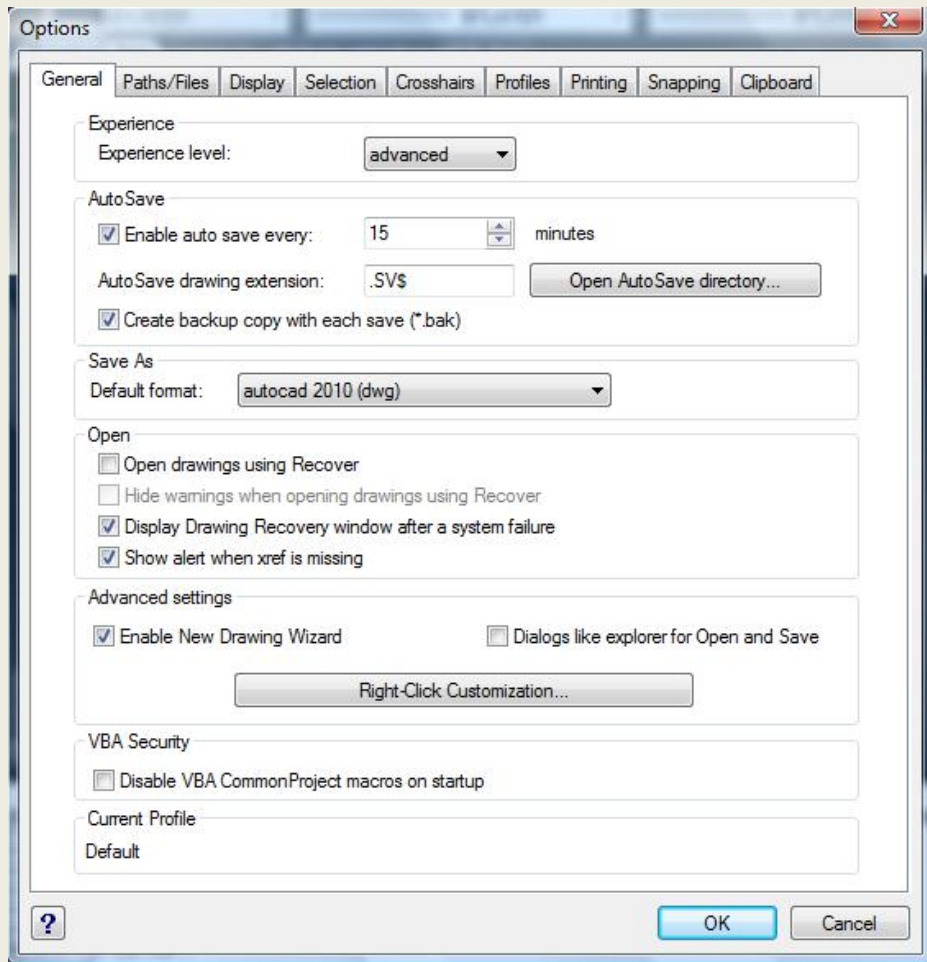
### MACROS :

Each time you start ActCAD, macros are automatically loaded for the Visual Basic Application (VBA) CommonProject. If you do not plan to use VBA, disabling the macros may improve performance. In addition disabling the macros can enhance security if you are running ActCAD at a low security level.

Menu: Tools-> Options> macros      Command: macros      Aliases: macros

### To change the options on the General tab

1. Do one of the following:
  - Choose Tools > Options.
  - Type config and then press Enter.
2. Click the General tab.
3. Under Experience level, select Beginner, Intermediate, or Advanced.



## Loading and running LISP Program:

Menu: Tools->Load LISP Command: Load LISP Aliases: **appload**

Another way you can customize ActCAD is to add custom programs written in any of several programming languages that run within ActCAD, including the following:

- LISP
- Visual Basic for Applications (VBA)
- ADS
- DIESEL

In ActCAD, you can run many programs originally created for use with AutoCAD.

Specifically, you can use programs written entirely in AutoLISP with no modification. In addition, you can run many ADS programs originally written for use with

AutoCAD after first recompiling them using the ActCAD run-time libraries. Many

AutoCAD third-party programs are compatible with ActCAD

Using LISP routines: ActCAD supports the LISP programming language and is compatible with AutoLISP, the implementation of the LISP language in AutoCAD. This means that you can load and run any AutoLISP program written for use with AutoCAD.

## To load a LISP routine

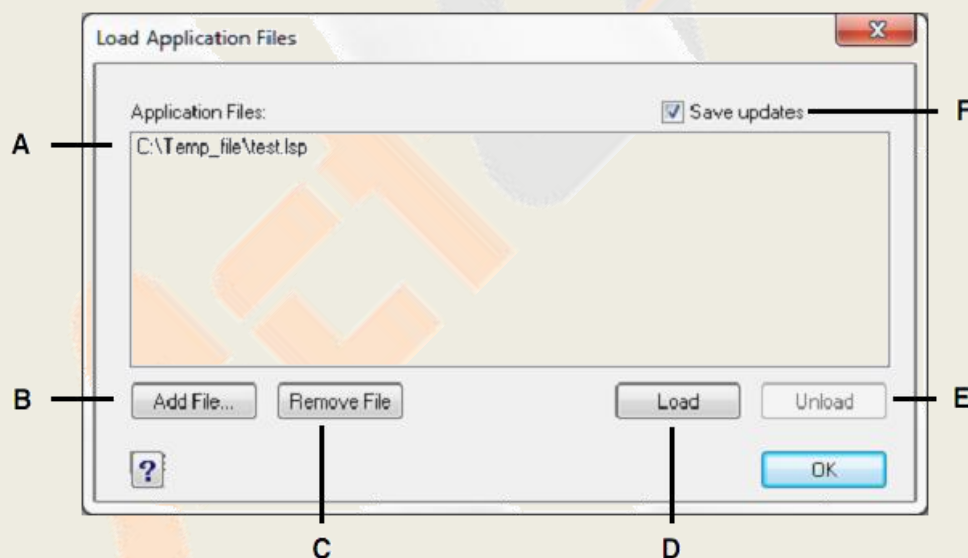
1 Do one of the following:

- Choose Tools > Load LISP or SDS Application.
- Type `appload` and then press Enter.
- Drag and drop the LISP file into ActCAD.

2 In the Load Application Files dialog box, click Add File.

3 Select the LISP file that you want to load, and then click Open.

4 Click OK.



A Lists the names of LISP and SDS files already loaded.

B Click to add a LISP or SDS file.

C Click to remove the highlighted file.

D Click to load the highlighted file.

E Click to unload the highlighted file.

F Select to save the current list to the `icadload.dfs`

file when you click Load, Unload, or OK.

## To run a LISP routine

1 Do one of the following:

- Choose Tools > Load LISP or SDS Application.
- Type `appload` and then press Enter.

2 In the Load Application Files dialog box, choose the routine you want to run (make sure that it is the only one selected), and then click Load. Some LISP routines are created in such a way that you can run them by simply typing the name of the routine, or by typing a keyword, directly in the command bar. If nothing happens when you attempt to run the LISP routine from within the Load Application Files dialog box, turn on the display of the command bar or Prompt History window by choosing View > Command Bar or View > Prompt History Window, and look for an entry that is similar to the following: Loading D:\path\routine.lspC:KEYWORD

## Drawing Settings

With ActCAD, you typically draw at full-size (1:1 scale), and then set a scale factor when you print or plot your drawing. Before you begin drawing, however, you need to determine the relationship between drawing units and real-world units.

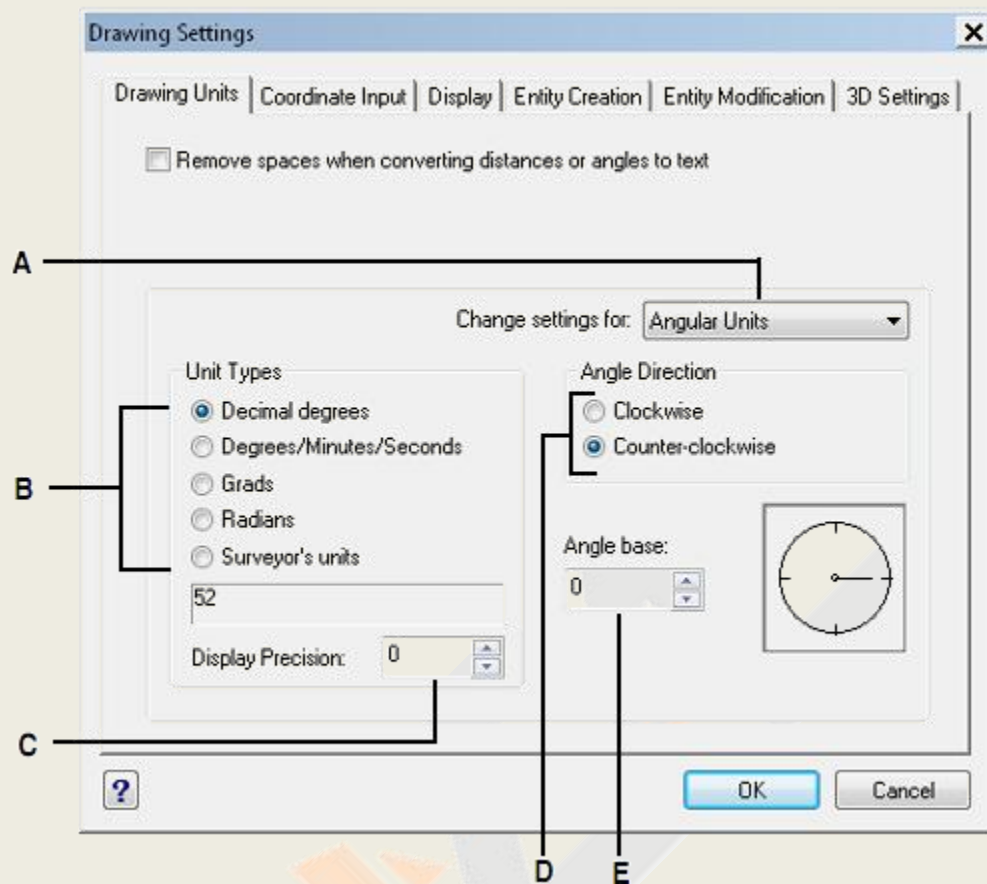
Menu: Tools > Drawing Settings Command: Drawing Settings Aliases: SE

To set the angular drawing units

1 Do one of the following:

- Choose Tools > Drawing Settings.
- Type `settings` and then press Enter.

2 Click the Drawing Units tab.



- A Determines the type of units you are controlling.
- B Select the type of angular units.
- C Choose the display precision for angular units.
- D Select the direction in which angles increase.
- E Select the angle base, the direction of the zero angle.

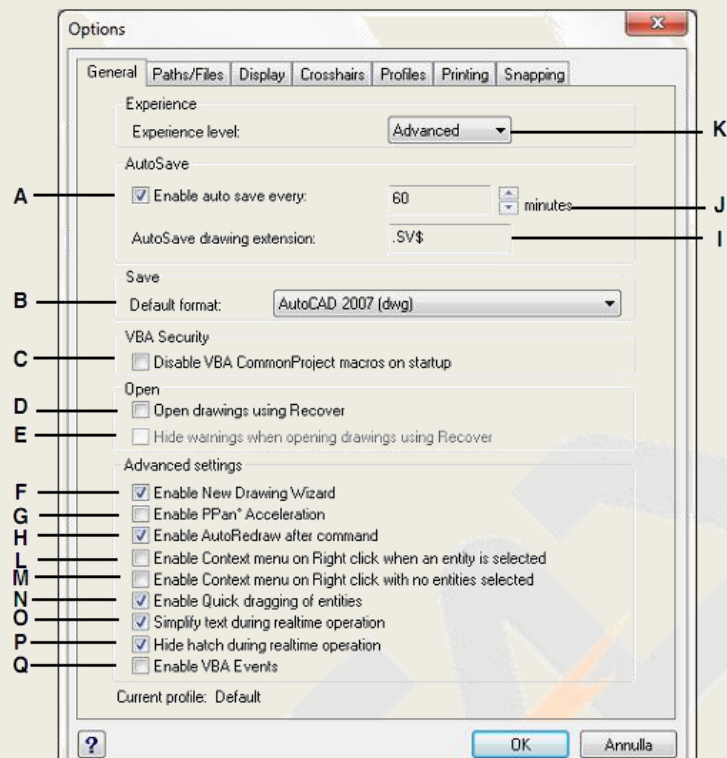
#### ActCAD OPTIONS:

You can specify locations for various file types such as those used for drawings, fonts, and menus in the Options dialog box on the Paths/Files tab. You can even specify multiple paths for the same file type. In addition, you can change the names of the default system files that are used for functions such as font mapping and error logging.

Menu: Tools >Options

Command: Options

Aliases: pr



A Click to enable AutoSave feature.

B Select the default file format that displays when saving drawings using the Save As dialog box.

C Click to disable VBA CommonProject macros on startup. (Available if supported by your version of ActCAD.)

D Select to check all drawings for errors when using the Open command, and attempt recovery, as needed.

E Select to hide warning messages when opening drawings, if the check box Open Drawings using Recover is marked.

F Select to check New Drawing Wizard.

G Select to check PPan Acceleration.

H Select to check AutoRedraw after commands.

I Type the file extension for AutoSaved files.

J Enter frequency of AutoSave in minutes.

K Select the experience level.

L Select to enable Content menu on right click.

**M** Select to enable the context menu if no entities are selected.

**N** Select to enable the quick dragging.

**O** Select to enable the simplification of the text during the realtime commands.

**P** Select to hide the hatching during the realtime commands.

**Q** Select to enable VBA Events.

**IN-PLACE BLOCK EDIT** In the previous ActCAD versions to modify and therefore replace all the occurrences inserted in a block drawing, it was needed to explode it, make the changes, and make it up again with the same name to redefine it.

Menu: Tools > In-place block edit Command: In-place block edit Aliases: REFedit

### To remove the object from a block

1 Do one of the following:

- Choose Tools > In-place Block Edit > Edit Reference and then select the block.
- Double click on the block to modify.
- Type refedit and press Enter key, then select the block.



### DATA EXTRACTION:

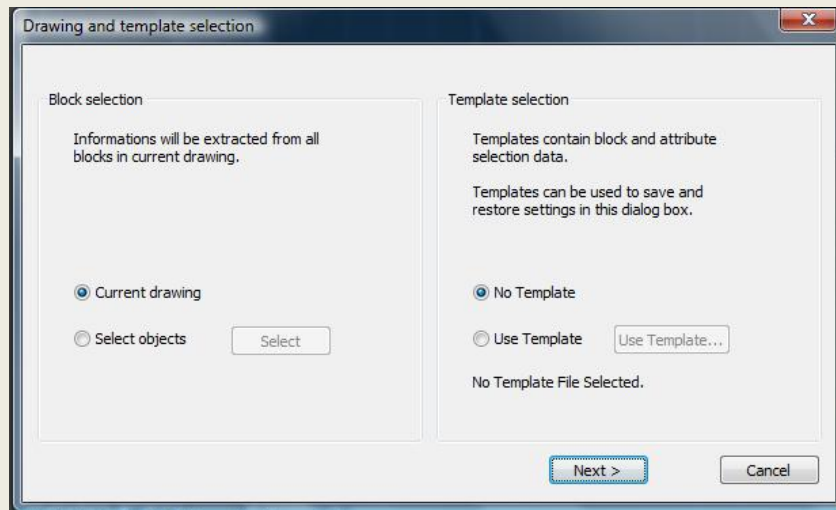
You can extract information from the blocks with attributes in the current drawing.

The information is used to create a data extraction table saved in an external file

Menu: Tools > Data extraction Command: Data extraction Aliases: Data extraction

1 Do one of the following:

- Select Tools > Data Extraction...
- Type EATTEXT and then press Enter.



2 The data extraction process starts with the Drawing and template selection dialog box.

No Template: creates a new data extraction.

Use Template: uses the settings previously saved in an attribute extraction

## UCS(USER COORDINATE SYSTEMS)

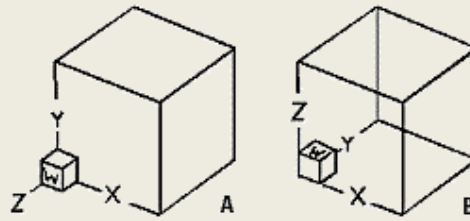
When working in three-dimensional space, you can define a UCS with its own 0,0,0 origin and orientation separate from the WCS. You can create as many user coordinate systems as you want, and then save and recall them as you need them to simplify construction of three-dimensional entities.

Menu: Tools > ucs Command: ucs Aliases: ucs

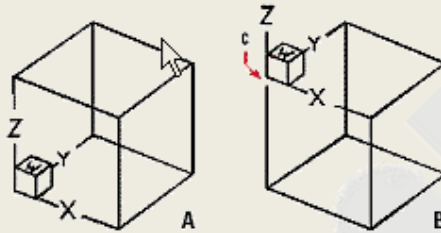


Usc:1.World UCS2. 2.UCS previous3.Face UCS 4.Object UCS 5.Origin UCS

6.View UCS 7.Rotate UCS about X-axis, Y-axis, Z-axis 8.UCS icon settings

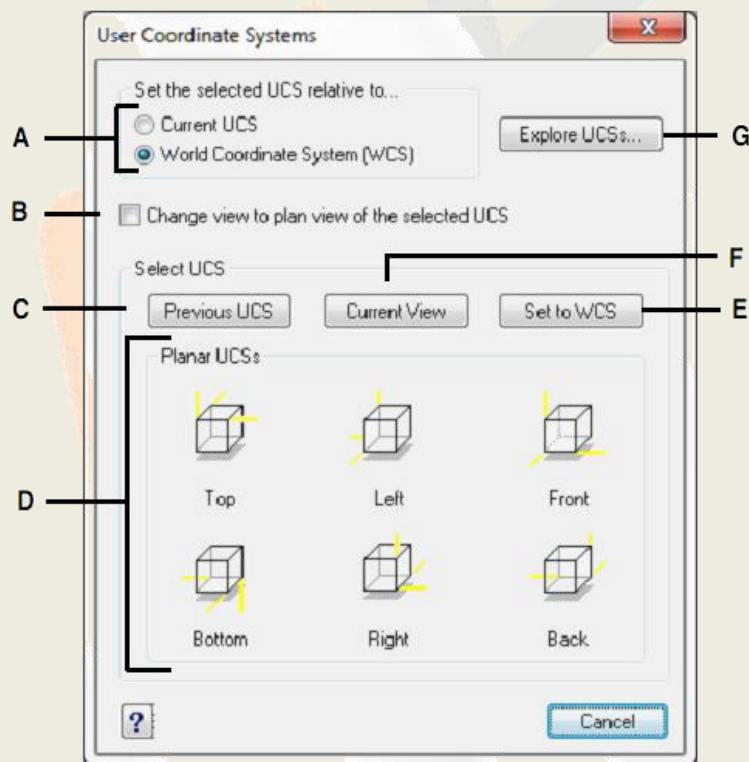


- A. Before changing the UCS to the World Coordinate System.  
B. After changing the UCS to the World Coordinate System.



- A. Before changing the origin of the current UCS.  
B. After changing the origin of the current UCS.  
C. The new origin (0,0,0).

Under Select UCS, click the button corresponding to the UCS you want as the new



current UCS..

- A Specify whether to define the new UCS relative to the current UCS or to the World Coordinate System (WCS).

- B Select to change the display to the plan view of the new UCS.
- C Click to select the previous UCS.
- D Click one of these buttons to select the view you want of a preset UCS.
- E Click to select the WCS.
- F Click to align the UCS with the current view.
- G Click to display the ActCAD Explorer.

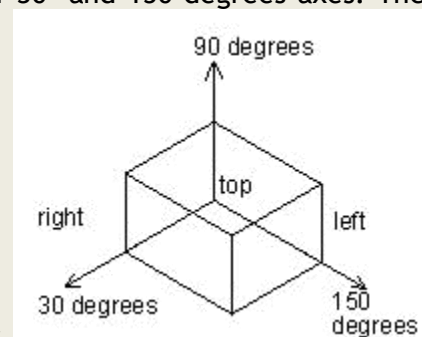
## Chapter 11 : Isometric Drawings

### Isometric views

To understand how the isometric views work, look down at the top of a box. If you move toward the lower-left corner, you are viewing the box from the SW Isometric View. If you move toward the upper-right corner of the box, you are viewing it from NE Isometric View.

By aligning along three major axes, isometric drawing simulates a 3D drawing from specified viewpoint. When the snap angle is set to 0, the axes of the isometric plane are 30 degrees, 90 degrees and 150 degrees. With Isometric Snap on, you can work on any of three isometric planes, each with a pair of associated axes.

- Left. The left isometric plane defined by a pair of 90- and 150-degrees axes. The snap and grips align along the 90- and 150-degree axes.
- Top. The top isometric plane defined by a pair of 30- and 150-degrees axes. The snap



and grips align along the 30- and 150-degrees axes.

- Right. The right isometric plane defined by a pair of 90- and 30-degrees axes. The snap and grips align along the 90- and 30-degrees axes.

### Set isometric snap and Grid:

You can use the Isometric Snap And Grid option to create two-dimensional isometric drawings. With the isometric option, you are simply drawing a simulated three-dimensional view on a two-dimensional plane, much the same as you might draw on a piece of paper. Do not confuse

isometric drawings with three-dimensional drawings. You create three-dimensional drawings in three-dimensional space.

**settings:**

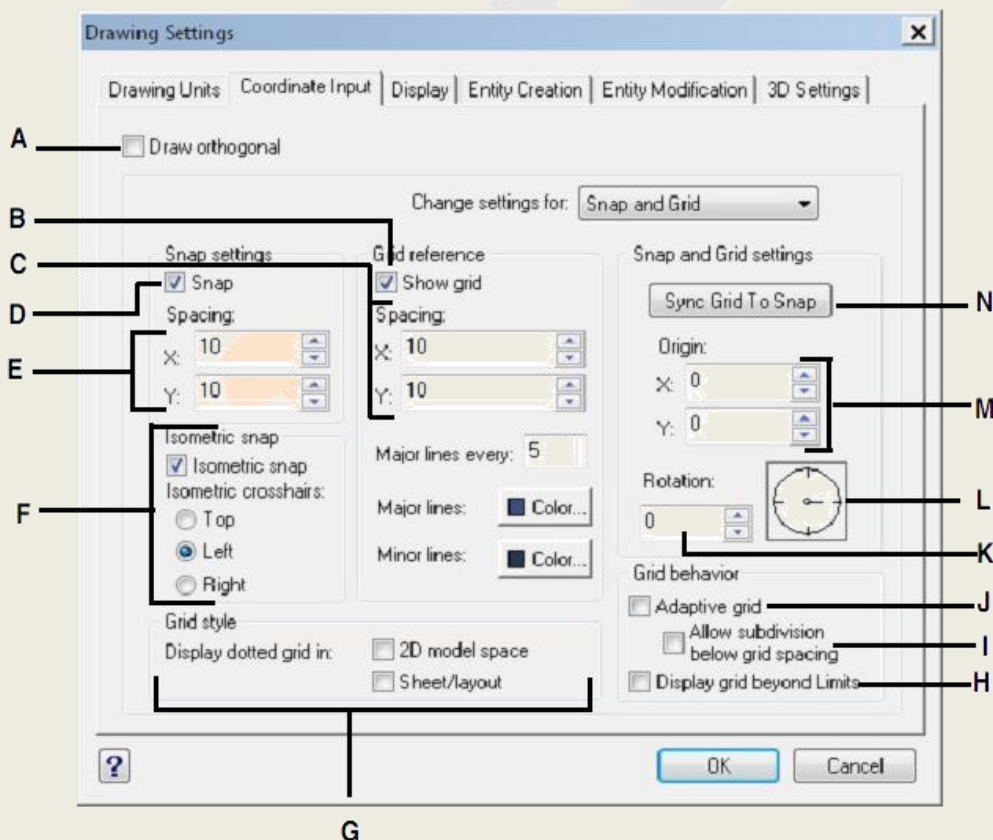
### Setting a reference grid

A reference grid displays as a pattern of regularly spaced lines. You can turn the display on and off, and you can specify how far apart the lines are spaced and their color.

The reference grid extends even over the limits of the drawing, helping you to visualize the boundary of your drawing and to align entities and visualize distances between entities. You can turn the grid on and off as needed. You can also change the spacing of the grid at any time.

### Setting snap spacing:

Another way to ensure drawing accuracy is to turn on and set snap spacing. When snap is turned on, the program restricts the selection points to predetermined snap intervals. Although it is often helpful to match the grid spacing to the snap spacing or another related setting, the settings do not have to match.



- A: Click the check box to enable orthogonal mode.
- B: Click the check box to display the reference grid.
- C: Specify the x and y grid spacing.

- D: Click the check box to enable snap mode.
- E: Specify the x and y snap spacing.
- F: Click the check box to use an isometric snap and grid and the isometric plane
- G: Specify where display the grid
- H: Click the check box to display the grid over the limits
- I: Click the check box to display the grid according to the zoom in
- J: Click the check box to display the grid according to the zoom out
- K: Specify the grid rotation angle.
- L: Indicates the current grid rotation angle. M Specify the x- and y-coordinates of origin.
- N: Click to match the grid spacing to the snap spacing.

### Change the crosshair:

In the Options dialog box, on the Crosshairs tab, you can control how the crosshairs display. To help you differentiate the x-, y-, and z-axes, a different color is assigned to each. You can change the default axes colors to any color you want. In addition, you can specify the size of the crosshairs display, enable grips attraction for the cursor, and you can elect to use crosshairs as the default pointer shape.

To turn the Isometric Snap And Grid option on

1 Do one of the following:

- Choose Tools > Drawing Settings.
- Type settings and then press Enter.

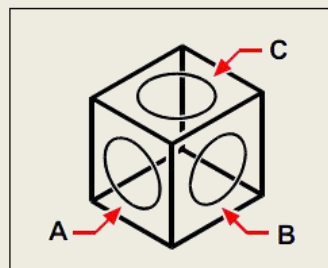
2 Click the Coordinate Input tab.

3 Under Change Settings For, select Snap And Grid.

4 Click the Isometric Snap And Grid check box.

5 Under Isometric Crosshairs, click the option for the isometric plane you want (Top, Left, or Right).

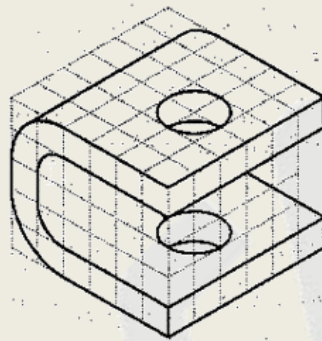
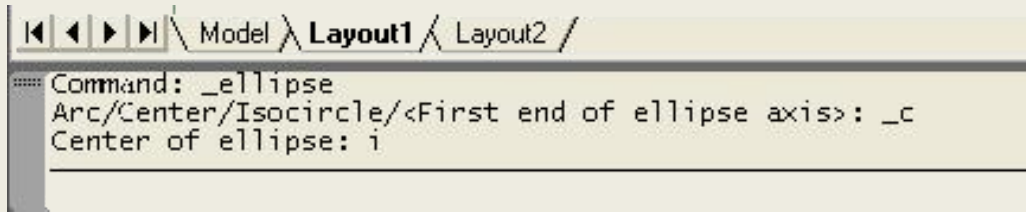
6 Click OK.



Isometric planes left (A), right (B), top (C).

### Draw Isometric Circles:

You can create an isometric circle on the current isometric plane using Ellipse Isocircle. The Isocircle option is available only when the Isometric Snap is on



### OBLIQUE DIMENSIONS :

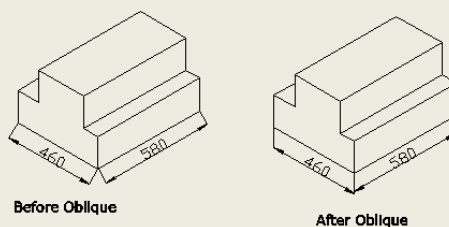
Extension lines are normally created at a perpendicular angle to the dimension line. You can change the angle of the extension lines, however, so that they tilt relative to the dimension line.

1 Do one of the following:

- Choose Dimension > Make Oblique.
- On the Dimension toolbar, click the Dimension Edit tool ( ) and then in the prompt box.
- Type dimedit, press Enter, and then in the prompt box, choose Oblique Lines.

2 Select the linear dimension, and then press Enter.

3 Type the obliquing angle, and then press Enter.



## Chapter 12 Layout, Viewports and Plotting

### UNDERSTANDING LAYOUTS

When you create a drawing, you do most of your work on the Model tab. Each drawing that you create can contain numerous layouts that simulate the paper on which you will print a copy of the drawing. Each of these layouts is created on a Layout tab.

### LAYOUT:

You can prepare a separate layout for each way you want to print your drawing. The layout allows you to organize different views to control which portion of your drawing prints and at what scale.

Menu: Insert -> Layout Command: LAYOUT

```
Command: LAYOUT
Enter layout option [Copy/Delete/New/Rename/Set/SAve/Template/? to list] <Set>: N
Enter a layout name <Layout1>: LAYOUT1
```

To Rename ->

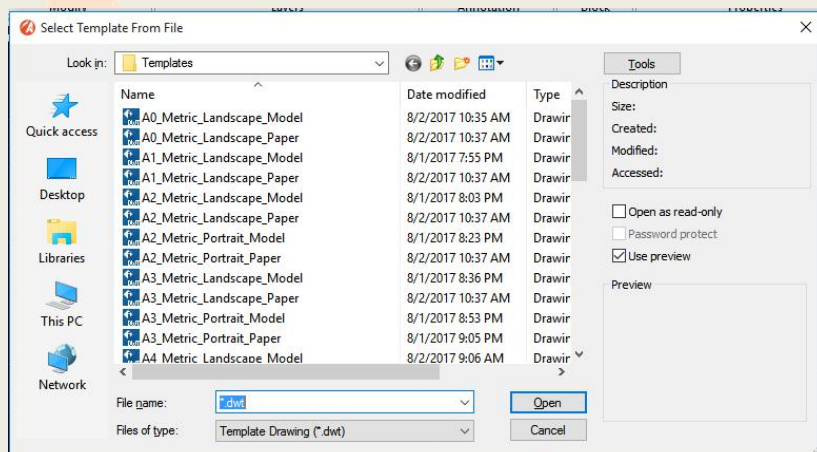
```
Command: LAYOUT
Enter layout option [Copy/Delete/New/Rename/Set/SAve/Template/? to list] <Set>: R
Enter a layout name <LAYOUT>: LAYOUT1
Enter new layout name: : LAYOUT2
```

To Delete ->

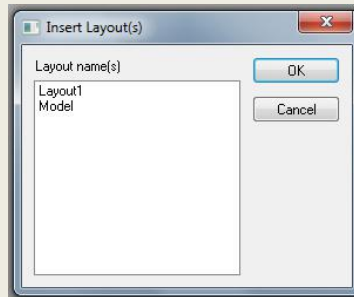
```
Command: layout
Enter layout option [Copy/Delete/New/Rename/Set/SAve/Template/? to list] <Set>: d
Enter a layout name <Layout1>: layout1
```

To Template ->

```
Command: LAYOUT
Enter layout option [Copy/Delete/New/Rename/Set/SAve/Template/? to list] <Set>: T
```



Select Existing Layout/Press ok



### Selecting a printer or plotter:

You can specify a printer or plotter to be used when printing any drawing. You can print your drawing on any printer or plotter that is compatible with Windows, including raster printers.

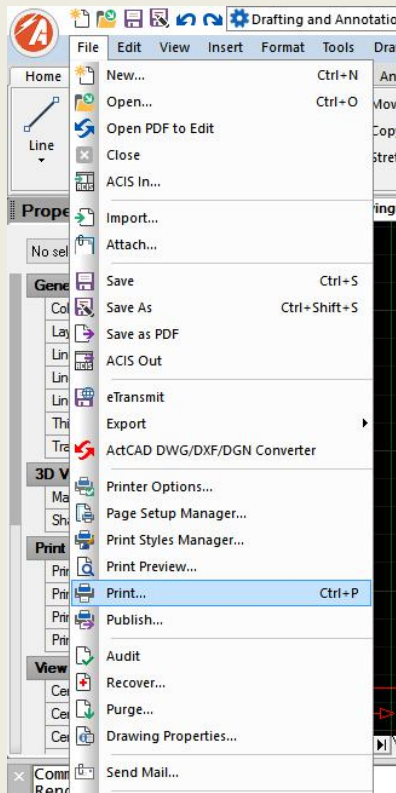
Menu: File -> Print Setup

Command: PRINT

1) From the Printer list, select a printer or plotter.

2) Do one of the following:

Choose File > Plot.



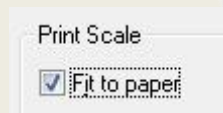
- On the Standard toolbar, click the Plot tool ( ).
- Type print and then press Enter.

Command: PRINT

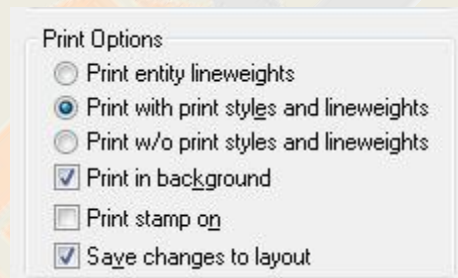
- 3) Click the Scale/View tab.



- 4) To scale the drawing to fit on one printed page, in Print Scale, select Scale to Fit

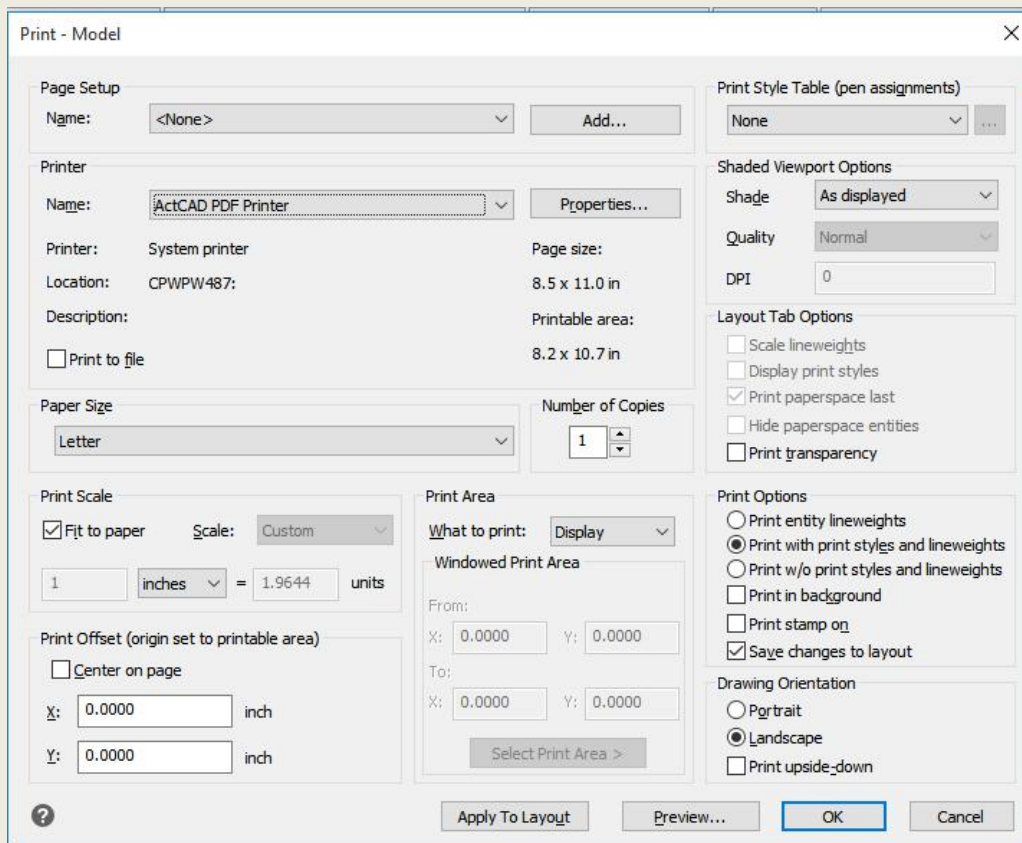


- 5) Select Save Changes to Layout, and then click Apply to save your changes.



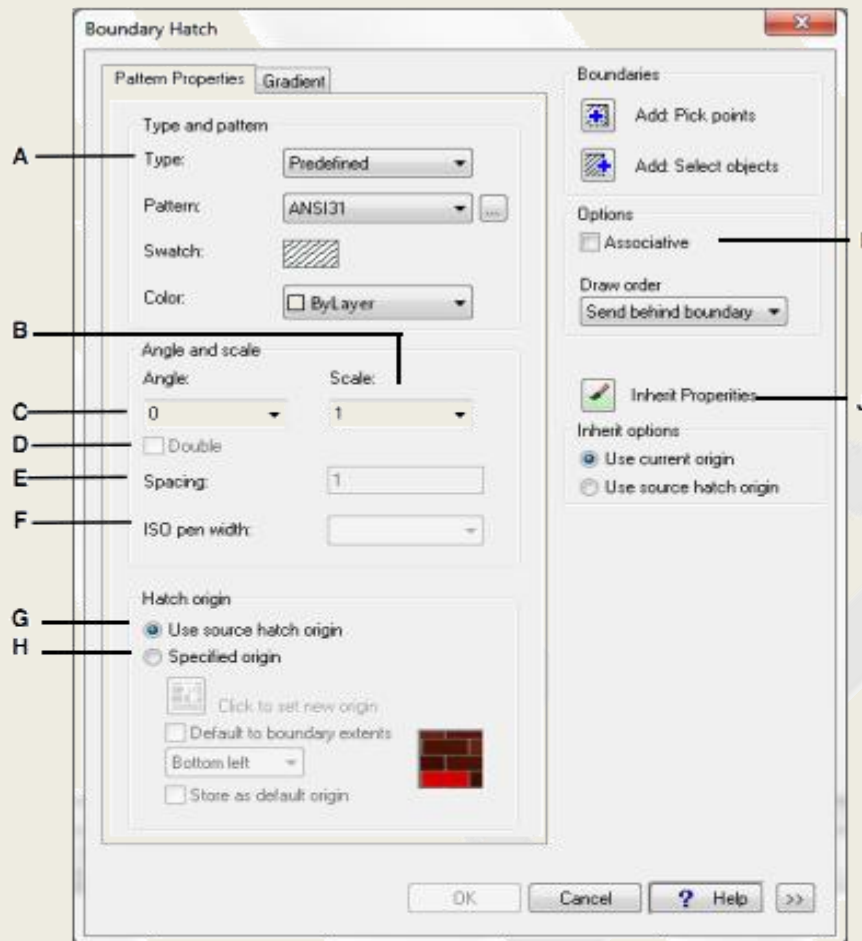
## SETTING THE SCALE AND VIEW

You can print or plot the entire drawing or a selected portion, depending on which options you select in the Print dialog box. You can choose to print what is visible on the screen, or you can specify to print an area of the drawing.



## Edit Hatch:

- 1) To edit the hatch, select the hatch content by double clicking on it & then Boundary objects window opens for edit.
- 2) Select the content to edit as shown below and press ok.

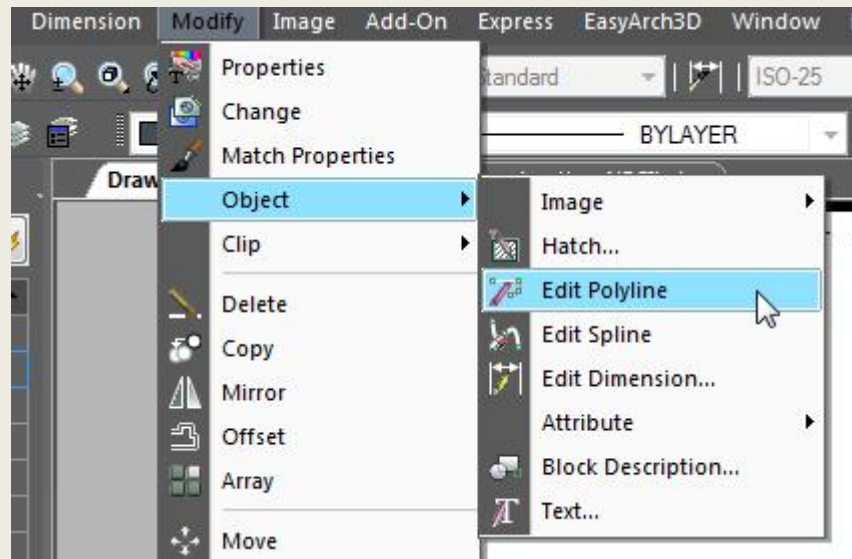


- A Determines how the hatch will be defined.
- B Determines density of hatching for Predefined hatch patterns.
- C Sets angle of hatch in relation to entity (User Defined only).
- D Imposes another copy of the specified pattern at a 90-degree angle over the first.
- E Determines density of hatching for User Defined hatch patterns.
- F Sets pen width for ISO-standard pattern.
- G Sets the current ucs origin.
- H allows you to choose a new origin.
- I Select to update the hatch automatically if you move any of its boundaries.
- J Select to copy an existing hatch pattern properties.

## Edit polyline:

You can edit entire polylines and individual segments using the Edit Polyline tool in the Modify > Object menu. You can convert polylines into arc and line entities using the Explode tool ( ) on the Modify toolbar. You can control whether wide polylines are shown filled or as outlines using the Fill tool in the Format menu

Menu: Modify ->Object -> Edit polyline      Command: pedit



```

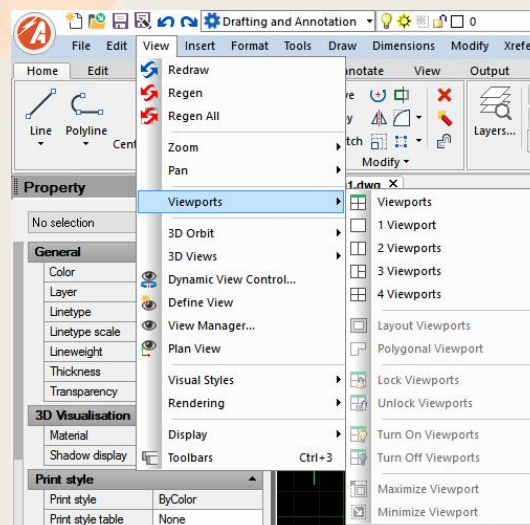
Command: pedit
Multiple/Select polyline to edit:
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: c
Edit polyline: Edit vertices/Open/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: o
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: d
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: f
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: j
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: R
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: S
Edit polyline: Edit vertices/Close/Decurve/Fit/Join/Linetype-mode/Reverse/Spline/Taper/Width/Undo/<eXit>: T
Enter new starting polyline width <0> :
Second point:
Enter new ending polyline width <0> : 10|
    
```

## VIEW PORTS IN LAYOUT:

Create at least one layout viewport on the Layout tab. Use each viewport to help control which portion of the drawing prints and at what scale.

Menu-> view-> viewports

Command: viewports

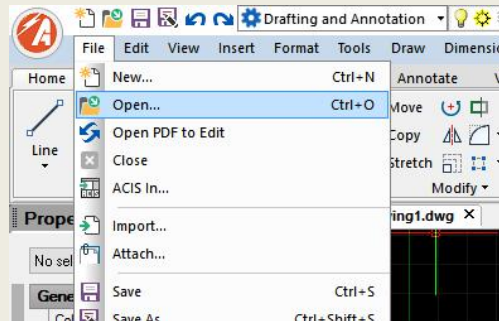


```
Command: viewports
Viewports: ON/OFF/Fit/Lock/Entity/Polygonal/2/3/4/<First corner>: 2
Two viewports: Horizontal/<Vertical>: v
Fit to screen/<First corner of bounding rectangle>: s
```

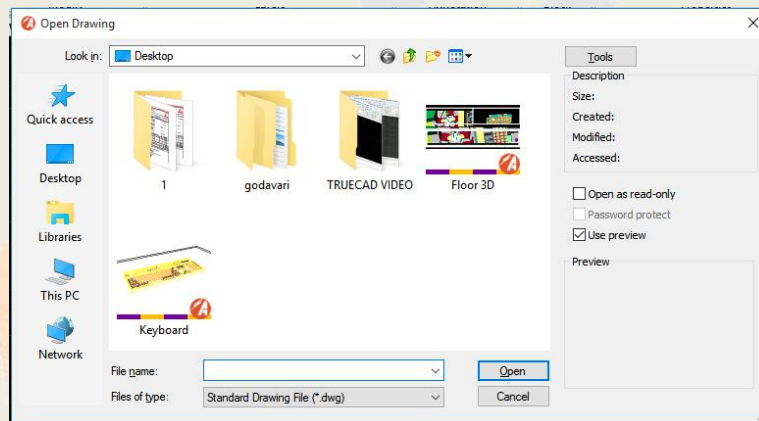
## FILE IMPORTS:

Importing DXF, DWF, and DWT files is similar to opening a standard drawing file.

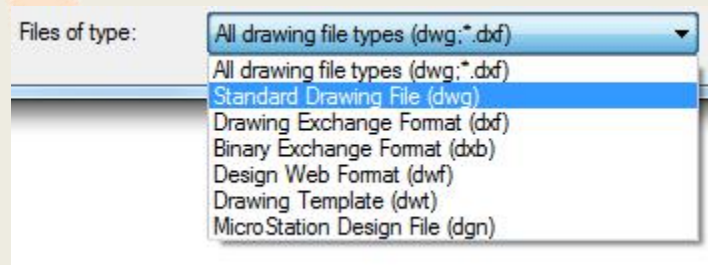
Menu-> File-> open    Command-> OPEN



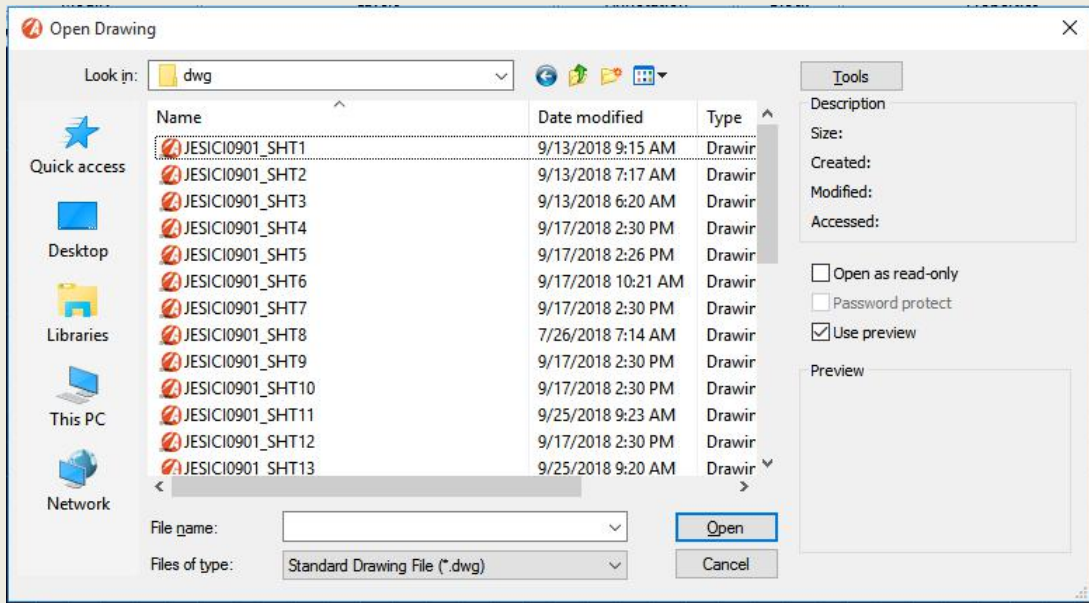
2) Choose the folder that contains the drawing.



3) In Files of Type, choose the type of drawing you want to import.



4) Choose the file you want to open.

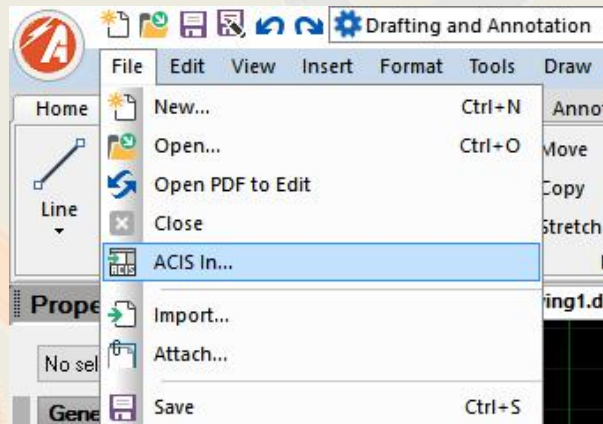


## IMPORTING AN ACIS FILE:

An ACIS file contains three-dimensional solids, regions, or bodies that are saved as an ASCII .sat file.

Menu-> File-> Acis In

Command-> ACISIN



2 Choose the directory containing the SAT file.

3 Choose the SAT file you want to open.

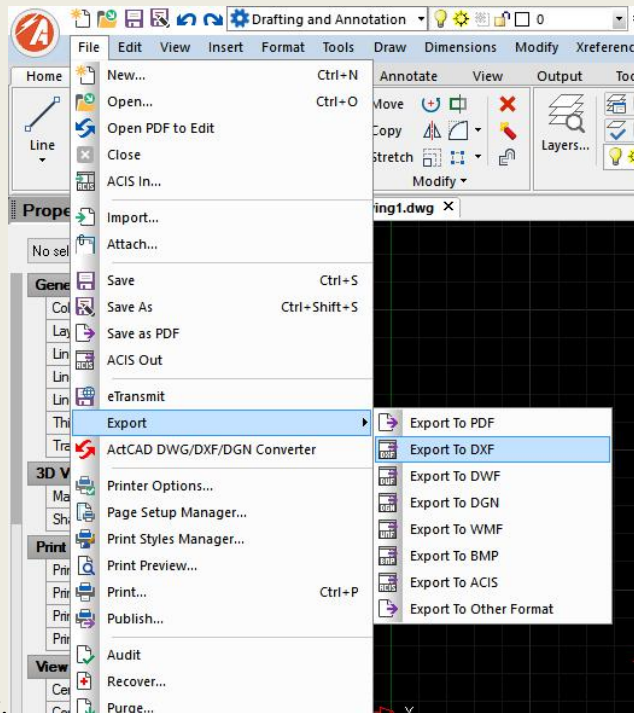
4 Click Open.

## EXPORT:

You can save or export ActCAD drawings in a number of different formats for use with other programs. When you save a drawing in a different format, the program saves all the entities in the drawing to the new file. Or, you can choose which entities are included in the new file.

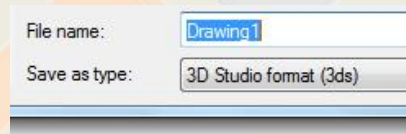
Menu-> File-> Export

Command-> EXPORT



2) Choose the file format.

3) Specify the name of the file you want to create.



4) Click Save.

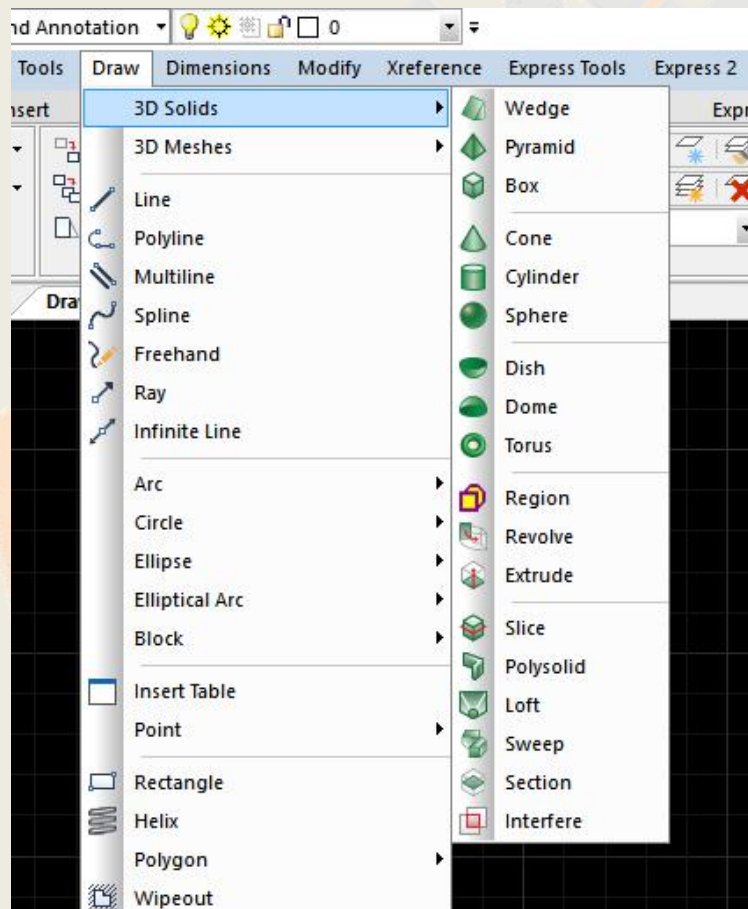
## Chapter 13 Draw Commands - 3D

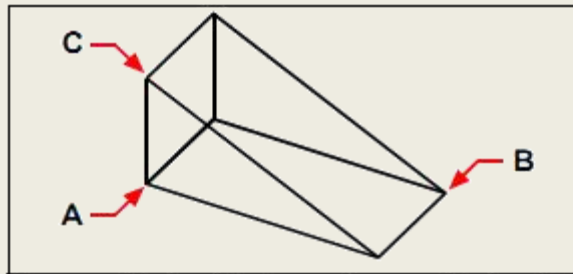
### Wedge:

You can create three-dimensional wedges consisting of five surface planes. The base of the wedge is always parallel with the xy plane of the current UCS with the sloped face opposite the first corner.

Menu-> Draw-> 3d Entities->Wedge

Command-> AI\_WEDGE





First corner of the base (A), the opposite corner of the base (B), and the height (C).

```
Command: ai_wedge
Specify corner point of wedge: 0,0,0
Specify length of wedge: 100
Specify width of wedge or [Cube]: 100
Specify height of wedge: 250
```

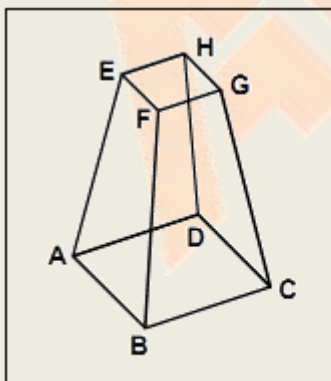
## PYRAMID:

You can create tetrahedrons (three-sided pyramids) or four-sided pyramids. The sides of the resulting pyramid can meet at a point (the apex) or can form a three- or four-edged top. The sides of a four-sided pyramid can also meet along a ridge defined by two points. The base of the pyramid is always parallel with the xy plane of the current UCS. You position the pyramid by specifying a corner of the base.

Menu-> Draw-> 3d Entities->Pyramid

Command-> AI\_PYRAMID

```
Command: ai_pyramid
First point for base of pyramid: 0,0,0
Second point: 100,0,0
Third point: 100,100,0
Tetrahedron/<Last point for base>: 0,100,0
Ridge/Top surface/<Apex of pyramid>: t
First point on top surface of pyramid: 0,0,150
Second point: 75,0,150
Third point: 75,75,150
Last point of top surface: 25,75,150
```



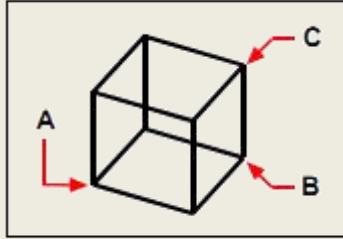
The first point (A), second point (B), third point (C), and fourth point (D) of the base, and the first point (E), second point (F), third point (G), and fourth point (H) of the top surface.

## BOX:

You can create rectangular boxes, or cubes. A box consists of six rectangular surface planes. The base of the box is always parallel with the xy plane of the current UCS.

Menu-&gt; Draw-&gt; 3d Entities-&gt;Box

Command-&gt; AI\_BOX



First corner of the base (A), the opposite corner of the base (B), and the height (C).

```
Specify corner point of box: 0,0,0
Specify length of box: 150
Specify width of box or [Cube]: 100
Specify height of box: 100
Specify rotation angle of box about the Z axis: 90
```

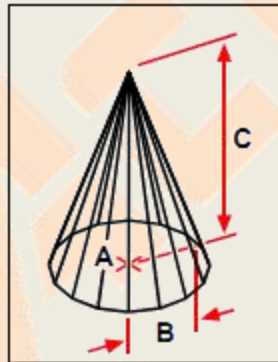
**CONE:**

You can create three-dimensional cones defined by a circular base and tapering to a point perpendicular to the base. The base of the cone is always parallel with the xy plane of the current UCS.

Menu-&gt; Draw-&gt; 3d Entities-&gt;Box

Command-&gt; AI\_BOX

```
Command: AI_CONE
Center of base of cone: 0,0,0
Diameter/Radius of base: 100
Diameter/Radius of top surface <0>: 0
Height of cone: 100
Number of divisions around cone <16>: 20
```



Center of the base (A), the radius of the base (B), and the height (C).

**CYLINDER:**

You can create cylinders defined by a circular base. The base of a cylinder is always parallel with the xy plane of the current UCS; the height of a cylinder is always parallel with the z-axis.

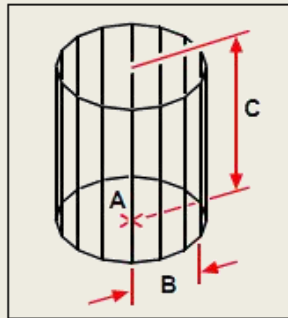
Menu-&gt; Draw-&gt; 3d Entities-&gt;Cylinder

Command-&gt; AI\_CYLINDER

```

Command: AI_CYLINDER
Center of base of cylinder: 0,0,0
Diameter/<Radius of cylinder>: 100
Height of cylinder: 150
Number of divisions around cylinder <16>: 20

```



Center of the base (A), radius of the base (B), and the height (C).

## SPHERE:

You can create spheres. The latitude lines of a sphere are always parallel with the xy plane of the current UCS; the central axis is always parallel with the z-axis. You position a sphere by specifying its center point.

Menu-> Draw-> 3d Entities->Sphere      Command-> AI\_SPHERE

```

Command: AI_SPHERE
Center of sphere:0,0,0
Diameter/<Radius of sphere>: 100
Number of longitudinal sections <16>: 20
Number of latitudinal sections <16>: 10

```



Center (A) and radius (B) of the sphere.

The resulting sphere.

## DISH:

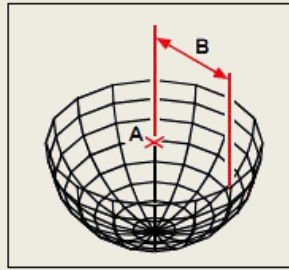
You can create a three-dimensional dish. The latitude lines of a dish are always parallel with the xy plane of the current UCS; the central axis is always parallel with the z-axis.

Menu-> Draw-> 3d Entities->Sphere      Command-> AI\_DISH

```

Command: AI_DISH
Center of dish: 0,0,0
Diameter/<Radius of dish>: 250
Number of divisions around dish <16>: 20
Number of top-to-bottom divisions <8>: 10

```



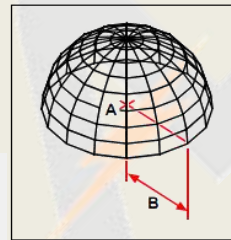
Center (A) and radius (B) of the dish.

**DOME:**

You can create a three-dimensional dome. The latitude lines of a dome are always parallel with the xy plane of the current UCS; the central axis is always parallel with the z-axis.

Menu-> Draw-> 3d Entities->Dome      Command->

```
Command: AI_DOME
Center of dome: 0,0,0
Diameter/<Radius of dome>: 100
Number of divisions around dome <16>: 20
AI_DOME Number of top-to-bottom divisions <8>: 10
```



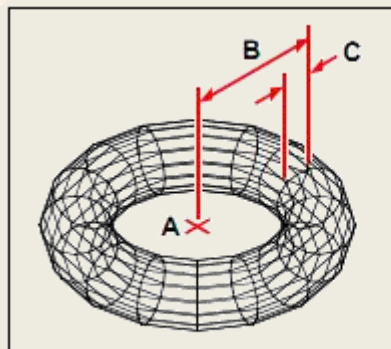
Center (A) and radius (B) of the dome.

**TORUS:**

You can create a three-dimensional donut or ring-shaped entity known as a torus. The diameter of a ring is always parallel with the xy plane of the current UCS. A torus is constructed by revolving a circle about a line drawn in the plane of the circle and parallel with the z-axis of the current UCS.

Menu-> Draw-> 3d Entities->TORUS      Command-> AI\_TORUS

```
Command: AI_TORUS
Center of torus: 0,0,0
Diameter/<Radius of whole torus>: 100
Diameter/<Radius of body of torus>: 20
Divisions around body of torus <16>: 10
Divisions around whole torus <16>: 20
```



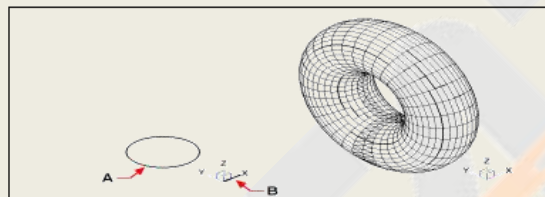
Center (A) and radius of the whole torus (B), and the radius of the body (C).

## REVOLVE:

You can create three-dimensional solids by revolving closed objects, such as polylines, polygons, circles, ellipses, and regions. You can revolve the object about a defined axis, line, polyline, or two points

Menu-> Draw-> 3d Entities->Revolve    Command-> REVOLVE    Aliases: REV

```
Command: REV
Select entities:
Opposite corner:
Entities in set: 1
Select entities:
Specify start point for axis of revolution or define axis by Object/Xaxis/Yaxis <Object>:
Specify endpoint of axis:
Specify angle of revolution or [Start angle] <360>: 360
```



Select the entity to revolve (A) and the axis (B) about which to revolve it., followed by the angle of revolution.

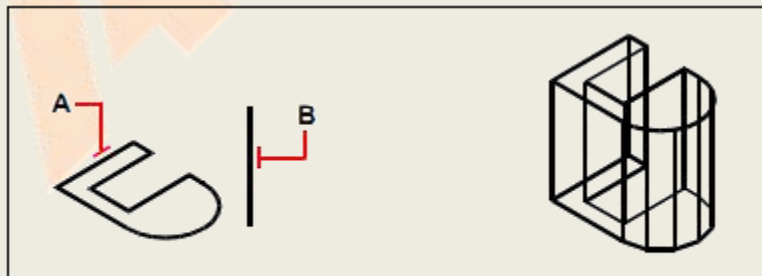
The resulting revolved solid.

## EXTRUDE:

You can create three-dimensional solids by extruding closed objects, such as polylines, polygons, circles, ellipses, closed splines, donuts, and regions. You can extrude the object along a selected path, or you can specify its height and taper angle

Menu-> Draw-> 3d Entities->EXTRUDE    Command-> EXTRUDE    Aliases: EXT

```
Command: EXTRUDE
Select entities:
Entities in set: 1
Select entities:
Specify height of extrusion or [Direction/Path/Taper angle] <0>: 150
```



Select the entity to extrude (A) and the extrusion path (B).

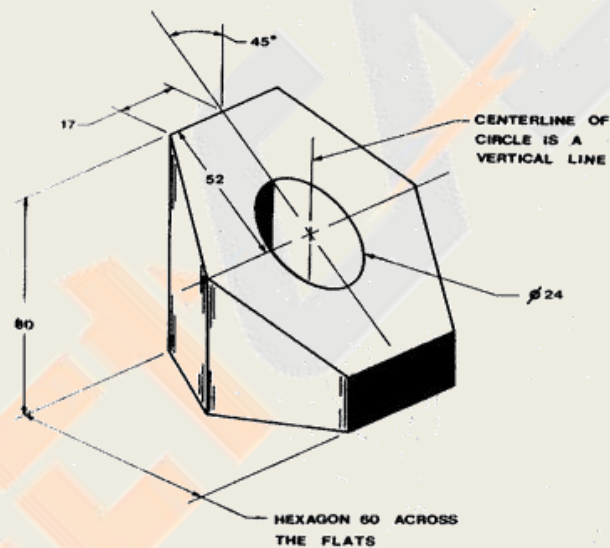
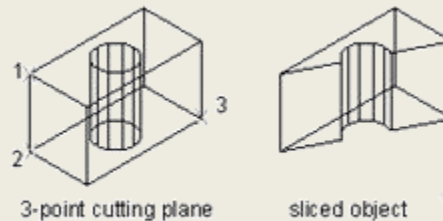
The resulting extruded solid.

## Slice:

You can section or slice a three-dimensional solid, region, or body (typically a sheet). When you section a solid, you obtain an “inside view” by creating a cross-section through the solid as a region or block. When you section a region or body, the resulting intersections are curves.

Menu: Draw » Solids » Slice    Command:SLICE    Aliases:SL

```
Command: SLICE
Select objects to slice:
Entities in set: 1
Select objects to slice:
Specify start point of slicing plane or [planar Object/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>:
Specify second point on plane:
Specify third point on plane:
Specify a point on desired side or [keep Both sides] <Both>:
```

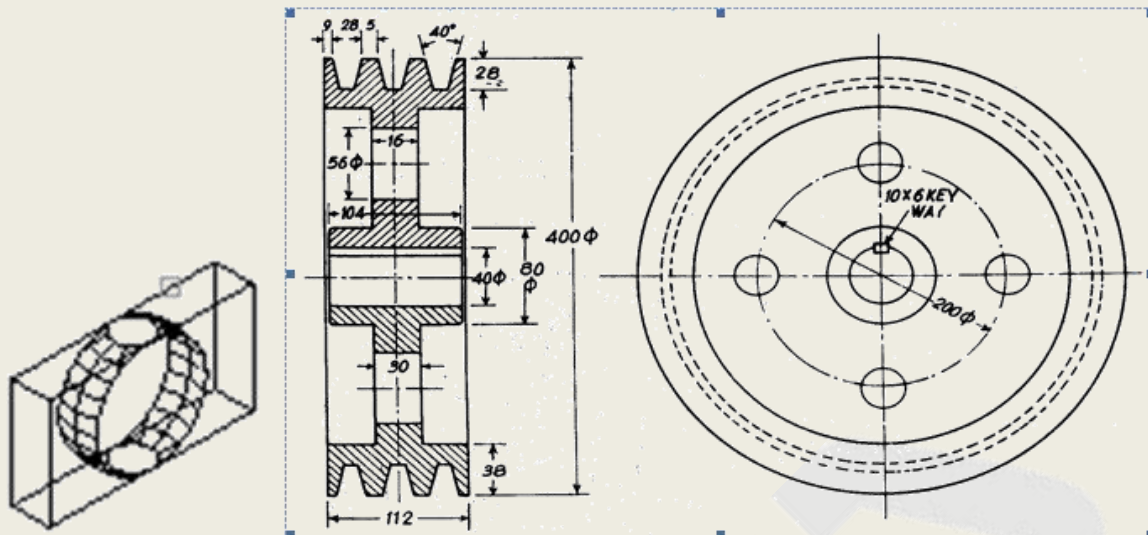


## Section:

When you section a solid, you obtain an “inside view” by creating a cross-section through the solid as a region or block. When you section a region or body, the resulting intersections are curves

Menu: Draw » Solids » Section    Command:SECTION    Aliases:SEC

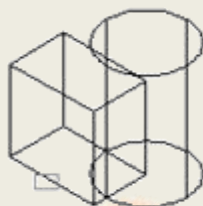
```
Command: SECTION
Select entities:
Entities in set: 1
Select entities:
Specify first point on Section plane by [Object/Zaxis/View/XY/YZ/ZX] <3points>:
Specify second point on plane:
Specify third point on plane:
```



## INTERFERE:

Creates a composite 3D solid from the common volume of two or more solids.

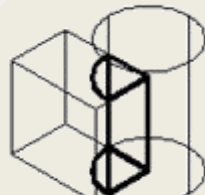
Menu: Draw » Solids » Interference Command: interfere Aliases:INF



first solid selected



second solid selected



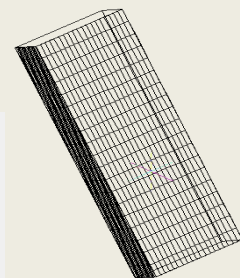
interference solid created

## MESH:

You can create a surface called a Coons surface patch, a mesh connecting four edges. You select the entities that define the edges. Edge entities can be arcs, lines, or polylines. The four edge entities must form a closed loop and share endpoints. A patch is a bicubic surface (one curve extends in the M-direction and the other in the N-direction) interpolated between the four adjoining edges. You can select the edges in any order. The first edge you select determines the M-direction of the mesh

Menu: Draw » Surface » Mesh Command: Aliases:

```
Command: _3D
Wedge/Pyramid/Box/Cone/cylinder/Sphere/Dish/Dome/Torus/Mesh: MESH
First of four corners for polygon mesh:
Second corner point of mesh:
Third corner point of mesh:
Last corner for mesh:
Number of divisions along primary axis (between 2 and 256): 10
Number of divisions along secondary axis (between 2 and 256): 10
```



**2D SOLIDS:**

The SOLID command creates 2D filled polygons. The first two points define one edge of the polygon. The third point is diagonally opposite the second point and the fourth point is diagonally opposite the first point.

Menu: Draw » Surface » 2d solid

Command: Solid

Aliases:

```
Command: _plane
Rectangle/Square/Triangle/<First point of plane>:
Specify second point:
Specify third point:
Specify fourth point or <exit>:
```

**3D SURFACE:**

You can create rectangular boxes, or cubes. A box consists of six rectangular surface planes. The base of the box is always parallel with the xy plane of the current UCS. You position the box by specifying either a corner or the center of the box. You determine the size of the box by either specifying a second corner and the height; defining the box to be a cube and then providing its length; or specifying the length, width, and height.

Menu: Draw » Surface » 3D Surface

Command:

Aliases:

**Revolved Surface:**

You can create a surface of revolution, which is a three-dimensional body that approximates the surface generated by rotating a two-dimensional profile around an axis. You select the two entities that define the profile and the axis. You also specify the starting angle and the number of degrees to revolve the profile.

Menu: Draw » Surface » Revolved Surface

Command: revsurf

Aliases: revsurf



Select the entity to be revolved (A) and the axis of revolution (B).



The resulting revolved surface mesh.

```
Command: revsurf
Select a entity to revolve:
Select the axis of revolution:
Opposite corner:
Angle to begin surface of revolution <0>:
Degrees to revolve entity (+ for ccw, - for cw) <360>:
Command:
```

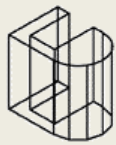
## Tabulated Surface:

You can create an extruded surface, which is a three-dimensional body that approximates the surface generated by extruding a path curve along a direction vector. You select the two entities that define the path curve and direction vector. The length of the direction vector determines the distance the path curve is moved along the direction vector. The extruded entity can be an arc, circle, line, or polyline. You can choose a line or open polyline as the direction vector. The resulting body consists of a series of parallel polygonal planes running along the specified path.

Menu: Draw » Surface » Tabulated Surface    Command: tabsurf    Aliases: tabsurf



Select the entity to extrude (A) and the extrusion path (B).



The resulting extruded surface mesh.

```
Command: tabsurf
Select entity to extrude:
Opposite corner:
Select line or open polyline for extrusion path:
Command:
```

## Ruled Surface:

You can create a ruled surface, which is a three-dimensional body that approximates the surface between two existing entities. You select the two entities that define the ruled surface. These entities can be arcs, circles, lines, points, or polylines.

Menu: Draw » Surface » Ruled Surface    Command: rulesurf    Aliases: rulesurf



Select the first (A) and second (B) defining entities.



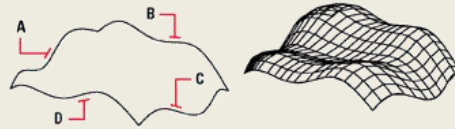
The resulting ruled surface mesh.

```
Command: rulesurf
Select entity to define first end of ruled surface:
Select entity to define other end:
```

## Edge Surface:

Creates a three-dimensional mesh with vertices averaged between four edges. You can apply this command to any four adjoining edges. The edges can be lines, arcs, or open two-dimensional or three-dimensional polylines, but all edges must be touching to form a closed, rectangular entity. You can use the SURFTAB2 system variable to define the N-direction mesh density of the coons surface.

Menu: Draw » Surface » Edge Surface    Command: edgesurf    Aliases: edgesurf



- A. Select the first edge for the mesh.
- B. Select the second edge.
- C. Select the third edge.
- D. Select the fourth edge.

```
Command: edgesurf
Select the first of four connected linear entities for edge surface:
Select second edge:
Select third edge:
Opposite corner:
Select last edge:
```

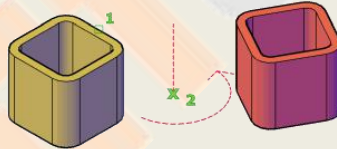
## Chapter 14 - Modify Commands - 3D

### 3D Operations :

#### Rotate 3D :

You can rotate selected entities about a specified axis in three-dimensional space.

Menu: Modify → 3D Operation → Rotate 3D    Command: Rotate3D    Aliases: Rotate3D

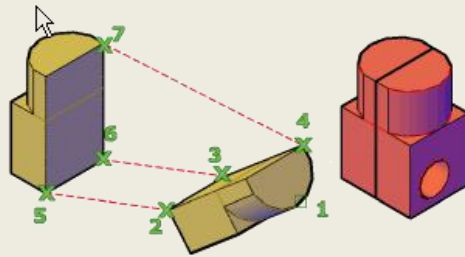


```
Command: rotate3d
FILTER/<Select entities to rotate>:
Entities in set: 1
FILTER/<Select entities to rotate>:
Select axis by: Entity/Last/View/Xaxis/Yaxis/Zaxis/<2 Points>: z
Specify point on the Z axis <0,0,0>: 0,0,0
Reference/<Rotation Angle>: 30
```

#### Align :

Aligning one object with another object by specifying 3 source points and 3 destination points.

Menu: Modify → 3D Operation → Align    Command: align    Aliases: align

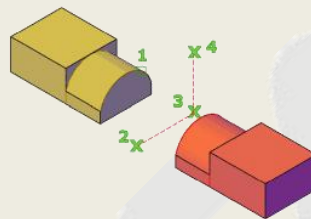


```
Command : align
Select entities:
Entities in set: 1
Select entities:
Specify first source point:
Specify first destination point:
Specify second source point:
Specify second destination point:
Specify third source point:
Specify third destination point:
```

## Mirror 3D :

Creates a mirrored copy of selected objects across a mirroring plane.

Menu: Modify → 3D Operation → Mirror3d    Command: Mirror3d    Aliases: Mirror3d



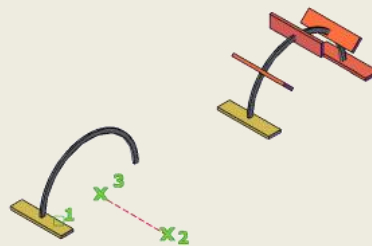
```
Command: mirror3d
Filter/<Select entities to mirror>:
Entities in set: 1
Filter/<Select entities to mirror>:
Define mirror plane by: Entity/Last/View/Zaxis/XY/YZ/ZX/<3points>: 3
First point on plane: 120,0,0
Second point on plane: @0,0,15
Third point on plane: @0,30,0
Delete the original entities? <N> n
```

## 3D Array :

Creates a 3D matrix of objects in a rectangular or polar arrangement.

Menu: Modify → 3D Operation → 3D Array    Command: 3darray    Aliases: 3darray

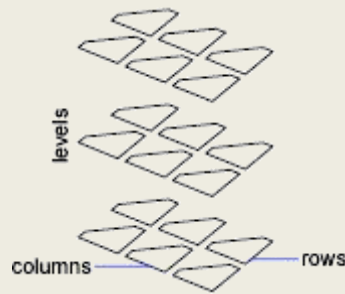
**Polar Array:** Copies objects about an axis of rotation.



```
Command: 3darray
Filter/<Select entities to array>:
Entities in set: 1
Filter/<Select entities to array>:
Type of array: Rectangular/<Polar>: p
ENTER to specify angle between items/<Number of items to array>: 4
Angle to array (+ for ccw, - for cw) <360>: -45
Rotate entities around the array? No/<Yes>: y
Center point of array:
Specify second point along central axis of array:
```

**Rectangular Array :** Copies objects in a matrix of rows (X axis), columns (Y axis), and levels (Z axis).

An array must have at least two rows or two columns or two levels.

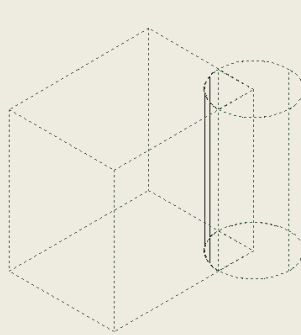


```
Command: 3darray
FILTER/<Select entities to array>:
Entities in set: 1
FILTER/<Select entities to array>:
Type of array: Rectangular/<Polar>: r
Number of rows in the array <1>: 2
Number of columns <1>: 3
Number of levels <1>: 3
Vertical distance between rows: 150
Horizontal distance between columns: 100
Depth between levels: 150
```

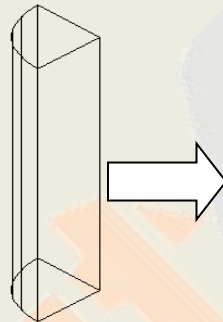
## Interfere :

Creates a temporary 3D solid from the interferences between two sets of selected 3D solids.

Menu: Modify → 3D Operation → Interfere    Command: Interfere    Aliases: Inf



2<sup>nd</sup> Set



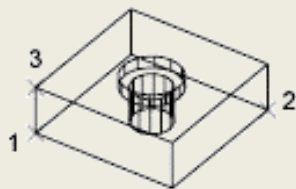
1<sup>st</sup> Set

```
Command: interfere
Select first set of ACIS objects:
Entities in set: 1
Select first set of ACIS objects:
Select second set of ACIS objects:
Entities in set: 1
Select second set of ACIS objects:
Comparing 1 solids against 1 solids.
Interfering pairs: 1
Create interference solids? Yes/<No>: y
Highlight pairs of interfering solids? Yes/<No>: y
Enter an option: <Next pair>/eXit: x
```

## Slice :

Slices the selected solid and retains either halves of the sliced solid or the portion you specified. The sliced solid retains the layer and color properties of the original solids.

Menu: Modify → 3D Operation → Slice    Command: Slice    Aliases: sl



3-point cutting plane



sliced object

```

Command: sl
Select objects to slice:
Entities in set: 1
Select objects to slice:
Specify start point of slicing plane or [planar Object/Surface/Zaxis/View/XY/YZ/ZX/3points] <3points>: 3
Specify first point on plane:
Specify second point on plane:
Specify third point on plane:
Specify a point on desired side or [keep Both sides] <Both>:
  
```

## Solids Editing :

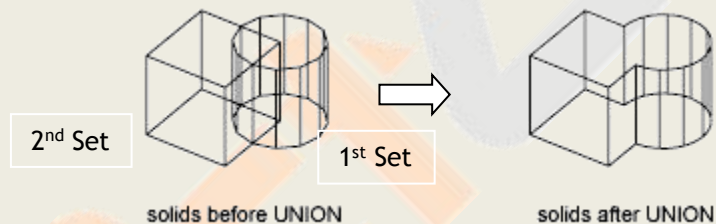
### Union :

Combines selected 3D solids, surfaces, or 2D regions by addition.

Menu: Modify → Solid Editing → Union

Command: union

Aliases: uni



```

Command : uni
Select ACIS objects to union:
Entities in set: 1
Select ACIS objects to union:
Entities in set: 2
Select ACIS objects to union:
  
```

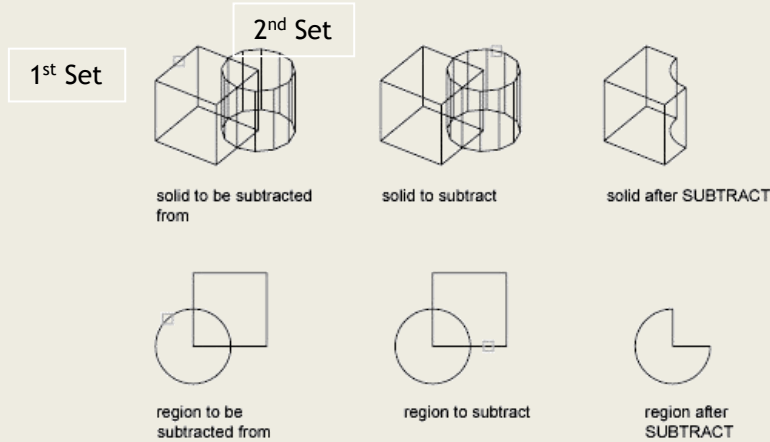
### Subtract :

Combines selected 3D solids, surfaces, or 2D regions by subtraction. With SUBTRACT, you can create a 3D solid or surface by subtracting one set of existing 3D solids from another, overlapping set.

Menu: Modify → Solid Editing → Subtract

Command: subtract

Aliases: su



```
Command: su
Select solids and regions to subtract from: 1st Set , Press Enter
Entities in set: 1
Select solids and regions to subtract from:
Select solids and regions to subtract:
Entities in set: 1
Select solids and regions to subtract: 2nd Set, Press Enter
```

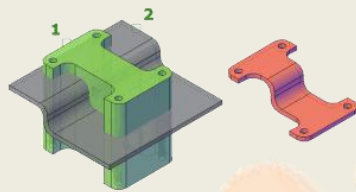
## Intersect :

Creates a 3D solid, surface, or 2D region from overlapping solids, surfaces, or regions.

Menu: Modify → Solid Editing → Intersect

Command: intersect

Aliases: in



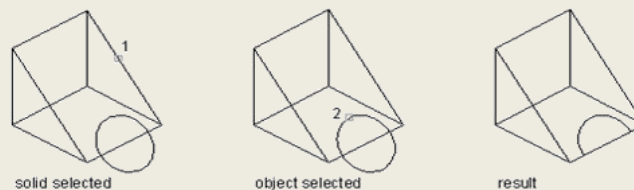
```
Command: in
Select ACIS objects to intersect:
Entities in set: 1
Select ACIS objects to intersect:
Entities in set: 2
Select ACIS objects to intersect:
```

## Imprint :

Imprints 2D geometry on a 3D solid or surface, creating additional edges on planar faces. Imprinting is limited to the following objects: arcs, circles, lines, 2D and 3D polylines, ellipses, splines, regions, bodies, and 3D solids.

Menu: Modify → Solid Editing → Imprint

Command: solidedit



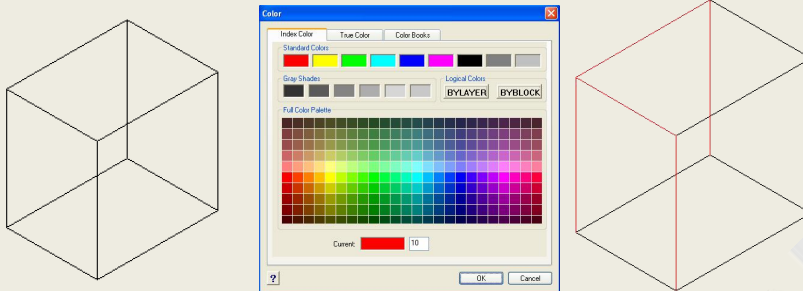
```
Command: _solidedit
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: _body
Enter a body editing option [Imprint/seParate solids/Shell/cLean/Check/Undo/eXit] <eXit>: _imprint
Select a 3D solid: → 1
Select an object to imprint: → 2
Delete the source object <N>: n Enter y (Yes) to remove it. Enter n (No) to retain it.
```

## Colour Edges :

Colour a set of edges for a 3d-solid or surfaces.

Menu: Modify → Solid Editing → colourededges

Command: solidedit



```
Command :_solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: e
Enter an edge editing option: Copy/coLor/Undo/<eXit>: l
Select edges: Remove/Undo:
Select edges: Remove/Undo/ALL:
Select edges: Remove/Undo/ALL:
Select edges: Remove/Undo/ALL:
Select edges: Remove/Undo/ALL:
Select edges: Remove/Undo/ALL:
Select edges: Remove/Undo/ALL:
```

f edges from a 3d solid or from surfaces.

## Extrude Faces :

Extends the dimensions of a 2D object or 3D face into 3D space. In most cases, if you extrude a closed object, the result is a new 3D solid. If you extrude an open object, the result is a surface.

Menu: Modify → Solid Editing → extrude faces

Command: extrude

Aliases : ext



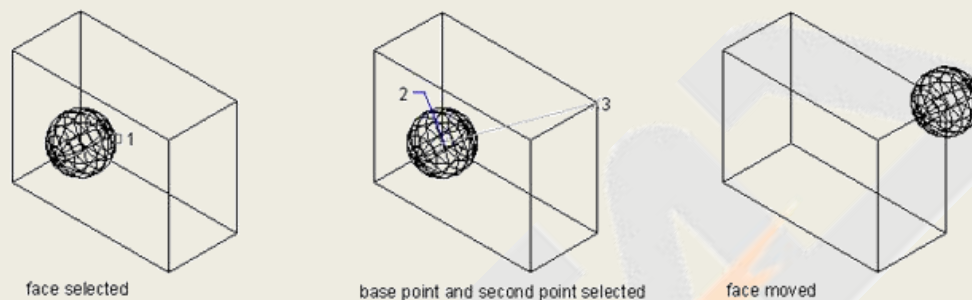
```
Command: ext
Select entities:
Entities in set: 1
Select entities:
Specify height of extrusion or [Direction/Path/Taper angle] <100>: t
Specify angle of taper for extrusion <3>: 4
Specify height of extrusion or [Direction/Path/Taper angle] <100>: -50
```

**Move Faces :**

Move the face of 3D solid to a specified distance.

Menu: Modify → Solid Editing → move face

Command: solidedit



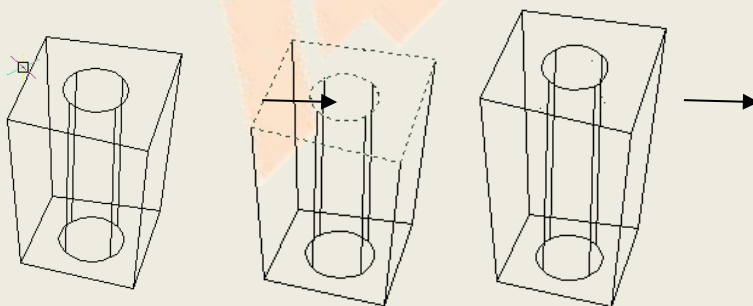
```
Command: _solidedit
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: _face
Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>: _move
Select faces or [Undo/Remove]:
Select faces or [Undo/Remove/ALL]:
Specify start point of direction:
Specify end point of direction:
```

**Offset Face :**

**Offsets faces by a specified distance.**

Menu: Modify → Solid Editing → offset face

Command: solidedit



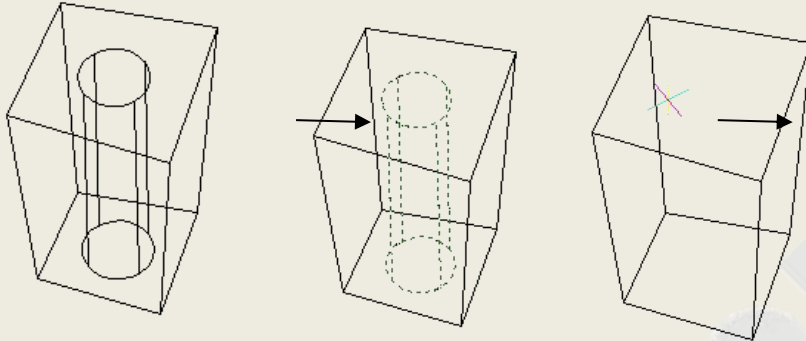
```
Command: solidedit
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: f
Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>: o
Select faces or [Undo/Remove]:
Select faces or [Undo/Remove/ALL]:
Specify the offset distance: 100
Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>: x
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: x
```

## Delete Face :

### Deletes the selected faces.

Menu: Modify → Solid Editing → delete face

Command: solidedit



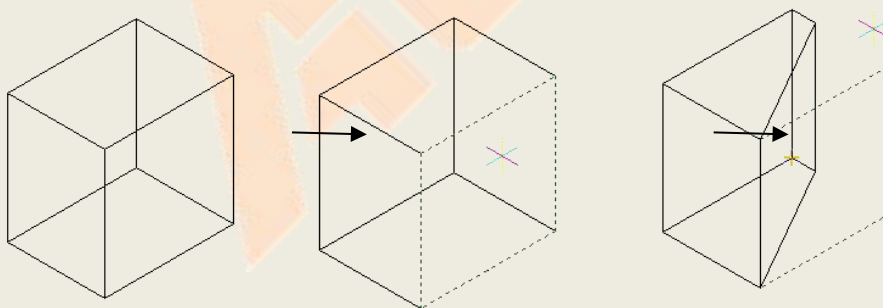
```
Command: solidedit
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: f
Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>: d
Select faces or [Undo/Remove]: 2 face(s) found.
Select faces or [Undo/Remove/ALL]:
Enter a face editing option
[Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/mAterial/Undo/eXit] <eXit>: x
Enter a solids editing option [Face/Edge/Body/Undo/eXit] <eXit>: x
```

## Rotate Faces :

### Rotate the selected faces about the specified axis.

Menu: Modify → Solid Editing → rotate face

Command: solidedit



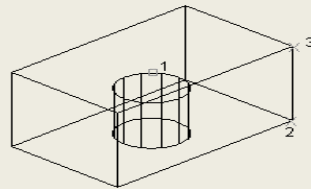
```
Command : Command : _solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: f
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: r
Select faces: Remove/Undo:
Select faces: Remove/Undo/ALL:
Specify the base point:
Specify another point along the axis:
Specify the rotation angle: 30
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: x
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: x
```

## Taper faces :

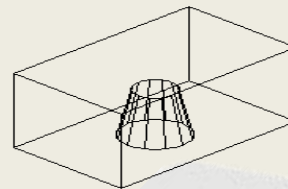
Tapers faces with an angle. The rotation of the taper angle is determined by the selection sequence of the base point and second point along the selected vector.

Menu: Modify → Solid Editing → taper face

Command: solidedit



face and two points selected



face tapered 10 deg

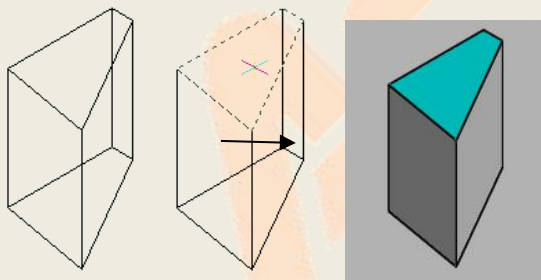
```
Command : Command : _solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: f
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: t
Select faces: Remove/Undo:
Select faces: Remove/Undo/ALL:
Specify the base point:
Specify another point along the axis:
Specify the taper angle: 10
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: x
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: x
```

## Colour Faces :

Colour the set of faces for a 3D solid.

Menu: Modify → Solid Editing → colour face

Command: solidedit



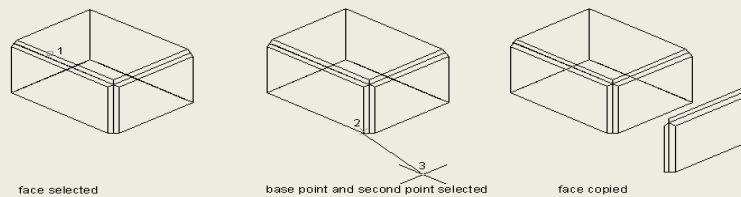
```
Command : Command : _solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: f
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: 1
Select faces: Remove/Undo:
Select faces: Remove/Undo/ALL:
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: x
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: x
```

## Copy Faces :

Copies the selected faces to a specified placement. Copy faces as a region or a body. The direction that you move the region is from the base point to the end point.

Menu: Modify → Solid Editing → copy face

Command: solidedit



```

Command : _solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: f
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: c
Select faces: Remove/Undo:
Select faces: Remove/Undo/ALL:
Specify <Base point>/Vector:
Specify end point:
Enter a face editing option: Extrude/Move/Rotate/Offset/Taper/Delete/Copy/coLor/Undo/<eXit>: x
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: x

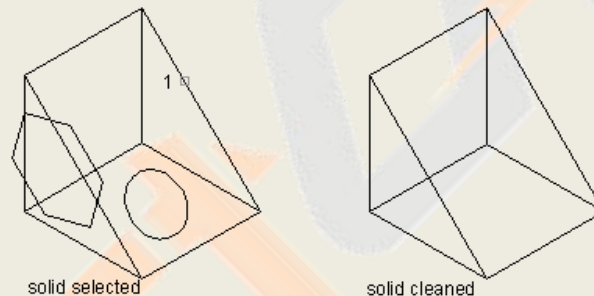
```

### Clean :

Removes shared edges or vertices that have the same surface or curve definition on either side of the edge or vertex.

Menu: Modify → Solid Editing → Clean

Command: solidedit



```

Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: _body
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>: _clean
Select 3d solid:
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>:
Cancel

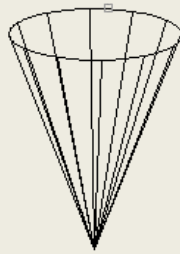
```

### Shell:

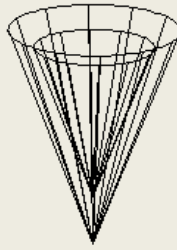
Shelling creates a hollow, thin wall with a specified thickness. You can specify a constant wall thickness for all the faces. You can also exclude faces from the shell by selecting them. A 3D solid can have only one shell. The system creates new faces by offsetting existing ones outside their original positions.

Menu: Modify → Solid Editing → Shell

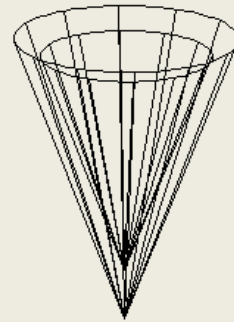
Command: solidedit



face selected



shell offset = 10



shell offset = -10

```

Command : _solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: _body
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>: _shell
Select 3d solid:
Remove faces: Add/Undo/ALL:
Remove faces: Add/Undo/ALL:
Enter the shell offset distance: .2
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>:
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>:

```

### Separate:

Separates 3D solid entities with disjointed volumes into independent 3D solid entities

Menu: Modify → Solid Editing → Separate

Command: solidedit

```

Command : _solidedit
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>: _body
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>: _separate
Select 3d solid:
Enter a body editing option Imprint/seParate/Shell/cLean/Check/Undo/<eXit>:
Enter a solids editing option: Face/Edge/Body/Undo/<eXit>:

```

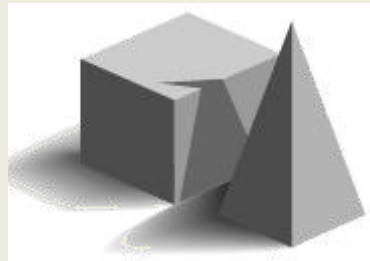
### Check:

Validates the 3D solid object as a valid Shape Manager solid.

## Chapter 15: Rendering

### RENDER:

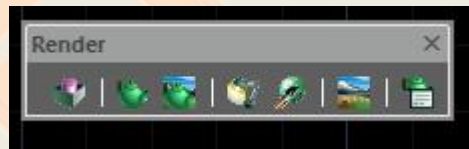
A visualization technique in which all surfaces of a model are shaded as though they were illuminated from an imaginary light source located behind you as you face the screen. Rendered images are photo-realistic, having depth, shadow, reflection, and texture.



Rendered object

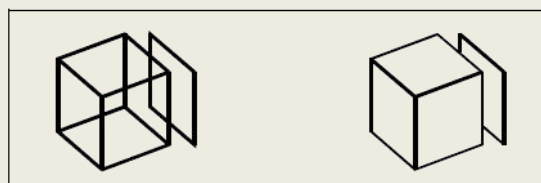
Full rendering creates a photo-realistic image of your model, complete with light sources, shadows, surface material properties, and reflections. You can illuminate your image with spotlights, distant lighting to simulate sunshine, and ambient light. If you choose not to customize the light sources, the program generates default light sources for you.

**Loading Render Toolbar:** Right click any tool bar you will get Toolbar list from that list select the render toolbar



### Render toolbar options:

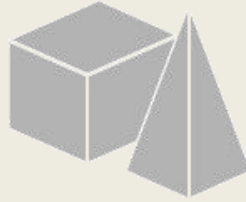
**Hide:** Creating a hidden-line view of your drawing removes all the lines that are hidden behind other surfaces when seen from your vantage point. When you remove hidden lines or shade a model, the program treats the entities differently, depending on how you created them. Wire-frame models always appear transparent, because they have no surfaces. Surface models appear filled, with surfaces applied to all visible sides.



Before creating a hidden-line image.

After removing hidden lines.

**Shade:** Creating a shaded image of your drawing removes hidden lines and then applies shading to the visible surfaces based on their entity color. Because they are intended to provide a quick visualization, shaded images do not have a light source and use continuous colors across surfaces, causing them to appear flat and unrealistic.



Shaded Objects

## Render:

Menu: View -> Render -> Render

Command: RENDER

Aliases :RR



Example of a rendered image

```
Command: render  
Rendered 14 Triangles in 0.21 sec
```

## Full Render:

Menu: View -> Render ->Full Render

Command: Full Render

Aliases :RR



Example of a rendered image

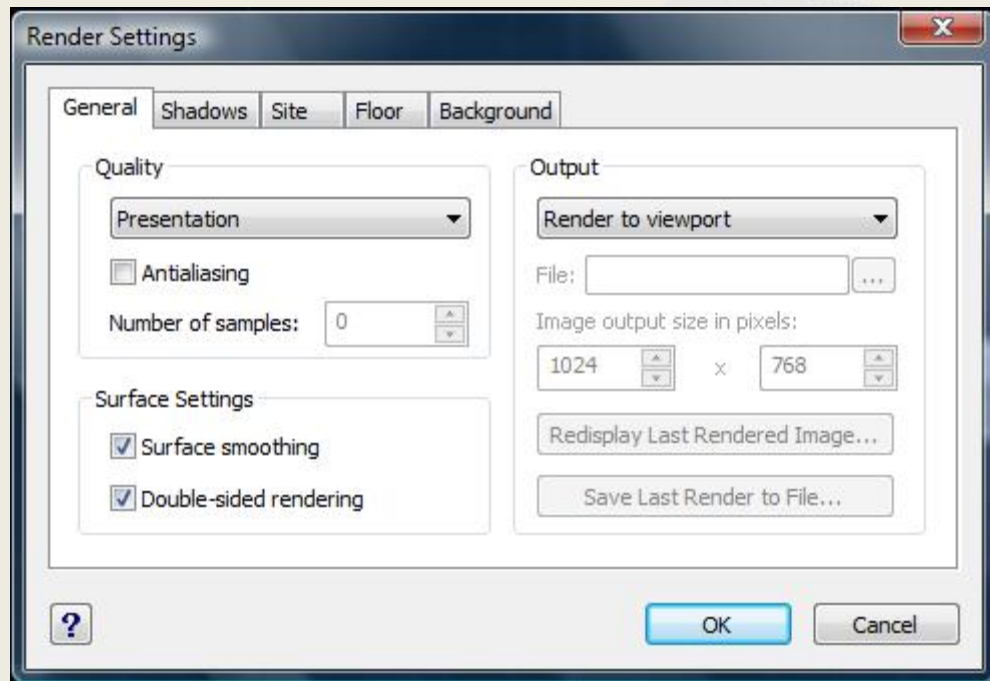
**Render preferences:** The Render Preferences command allows you to control several aspects of the rendering system and the Render Settings dialog box is divided into five different worksheets entitled: Rendering, Shadows, Floor, Site, and Render to File.

Menu: View -> Render -> Preferences

Command: Render Preferences

Aliases:-SETRENDER

The Rendering worksheet is shown below and is followed by a table listing each of the available controls and their function.



## MATERIALS:

Using materials for rendering bestows the rendered entity with strong realistic and stereoscopic sense.

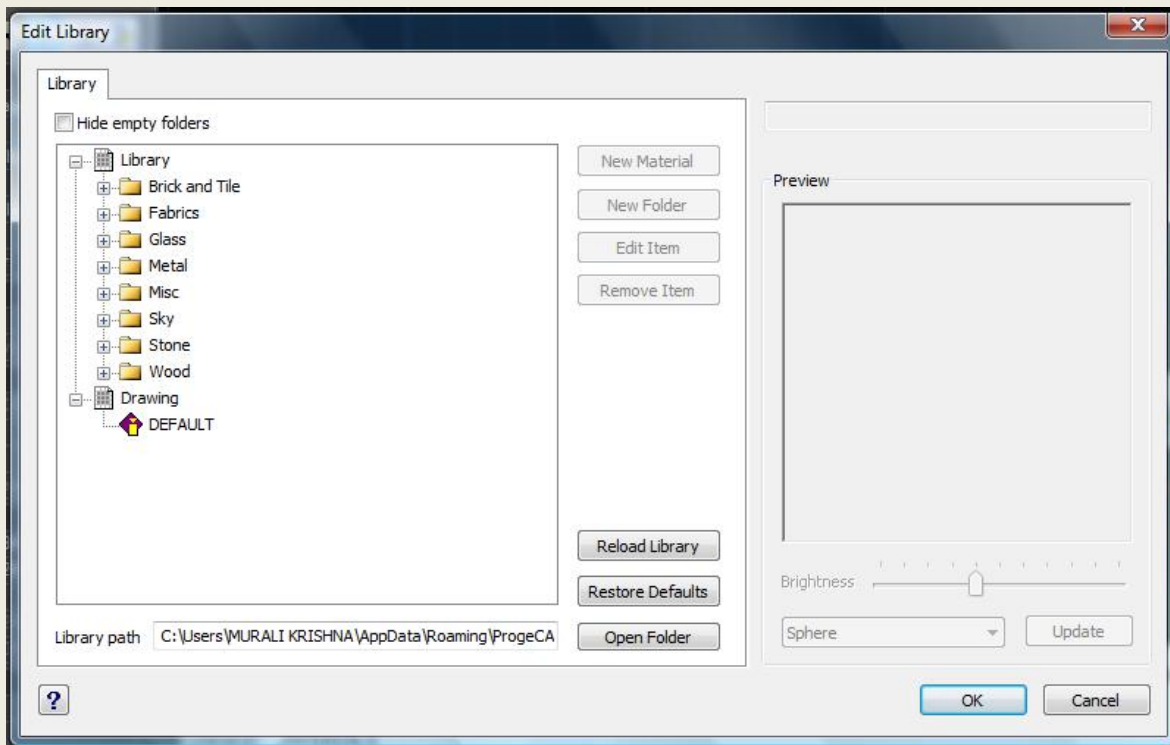
Several materials are provided with editable properties such as pattern type, color map and surface finish which decide how the material will be mapped to the surface of the entity in material library. If you have specified light to illuminate the entity, the material of the entity surface reflects the light clearly and generates shadows according to its own property, setting the materials and lighting properties together, you can get real visual effects of rendering. Thus you are required to make comprehensive modification for lighting, materials and backgrounds to obtain the expected visual effects.

Menu: View ->Render -> Materials

Command: MATERIALS

Alias: RMAT

```
Command: _MATERIALS
Assign materials to: <Selection>/Layer/Color/Browse: s
Filter/<Select entities>:
Entities in set: 1
Filter/<Select entities>:
```

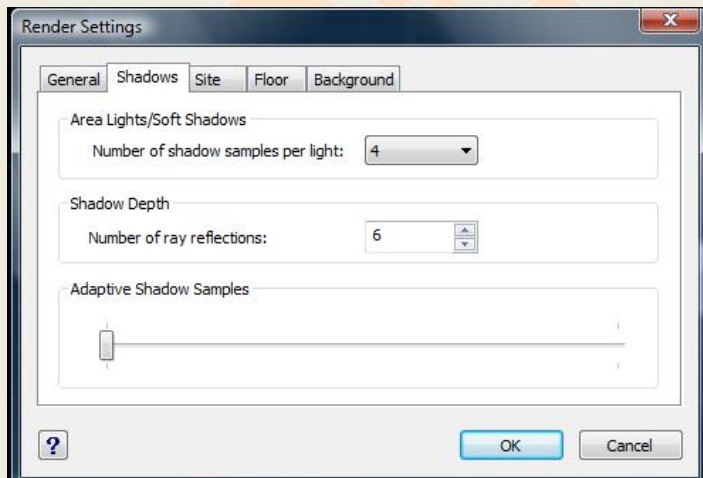


Material Assignment Dialog Box

In this dialog box you can apply and create new materials to your selected objects

**ADDING SHADOWS:** The Shadows worksheet appears on the Render Settings Dialog box and is used to control some of the aspects of the shadow computations in the rendering module.

Menu: View -> Render -> Preferences -> Shadows



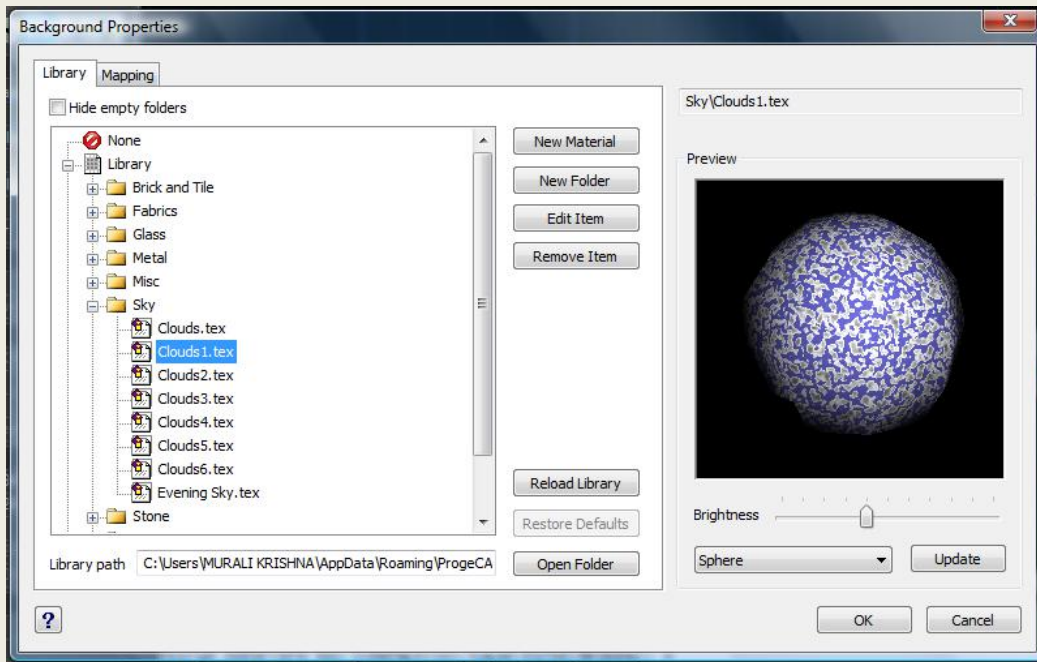
## BACKGROUND SETTING:

The Background Properties window allows you to specify the background or backdrop for your image. It is essentially identical to the Material Assignment window except that it applies to the

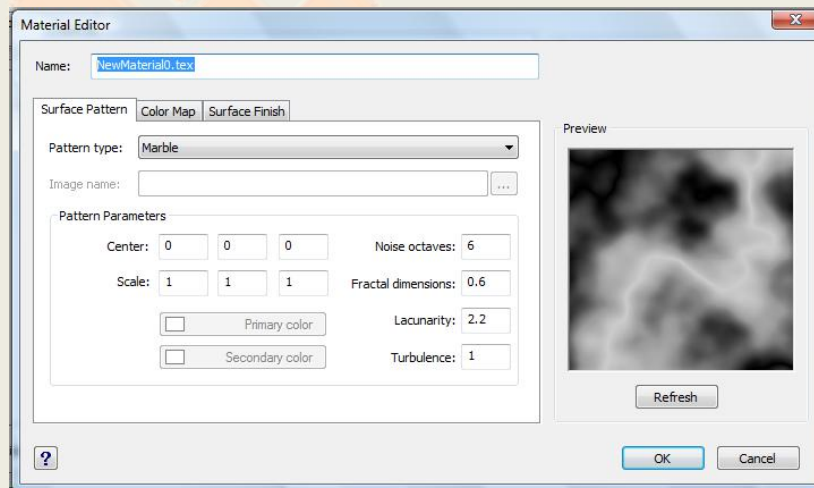
background instead of surfaces in the drawing. The background is an infinite plane behind the rest of your drawing. A variety of background options are available as described below. By default, no background is used and the background will appear black. Note that the background is an infinite, planar surface and will reflect off of any reflective surfaces in your model. The background is not affected by lighting however, so no shadows or highlights will be seen on the background.

Menu: View -> Render ->Background

Command:Background



**CUSTOMIZING MATERIAL:** In ActCAD you can customize the material properties like color, transparency by using material editor dialog box which is available in material Assignment dialog box.



## APPLYING LIGHTS:

Toolbar: Render ->Lights

Menu: View -> Render -> Light

Command: LIGHT

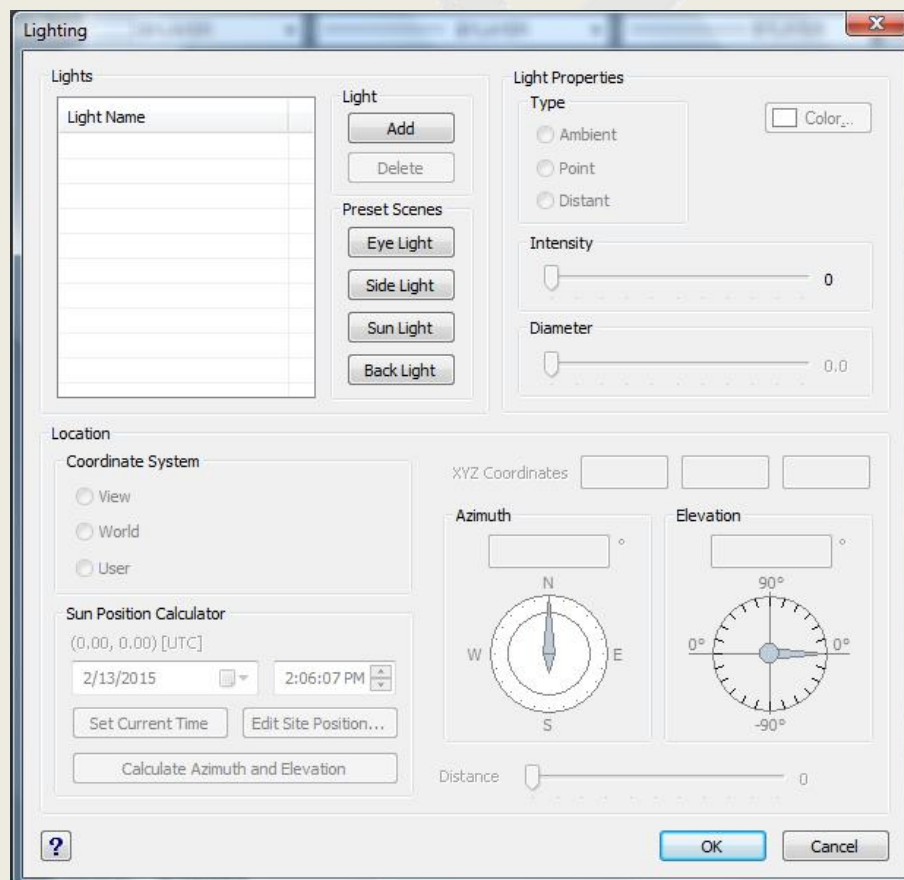
The Lighting command activates the Lighting dialog box which allows you to specify how your drawing or scene is lit. You can control placement of the lights, light color, and light intensity. The Lighting dialog box, illustrated below, includes four buttons in the upper left corner of the window that allow you to select a preset lighting configuration. To the right of those buttons is a list of all of the lights currently defined in the drawing and buttons that allow you to add or delete from the list of lights. The change the characteristics of a particular light in the drawing, you first click on the name of the light in the light list. In the upper right corner of the window, the name of the selected light will be shown, as well as the color of the light and the light type.

### Types of lights:

- Ambient light
- Point light
- Distant light

#### Ambient:

An ambient light illuminates the entire scene evenly and does not cast shadows or change with viewing angle. The parameters available for an ambient light are the light color and the light intensity. Ambient light can be used to fill in dark or shadowed areas of a drawing. It can also be used to make certain objects in a scene appear to glow. Note that too much ambient light will make the scene flat and lifeless since it flattens the shadows and highlights that provide depth cues.

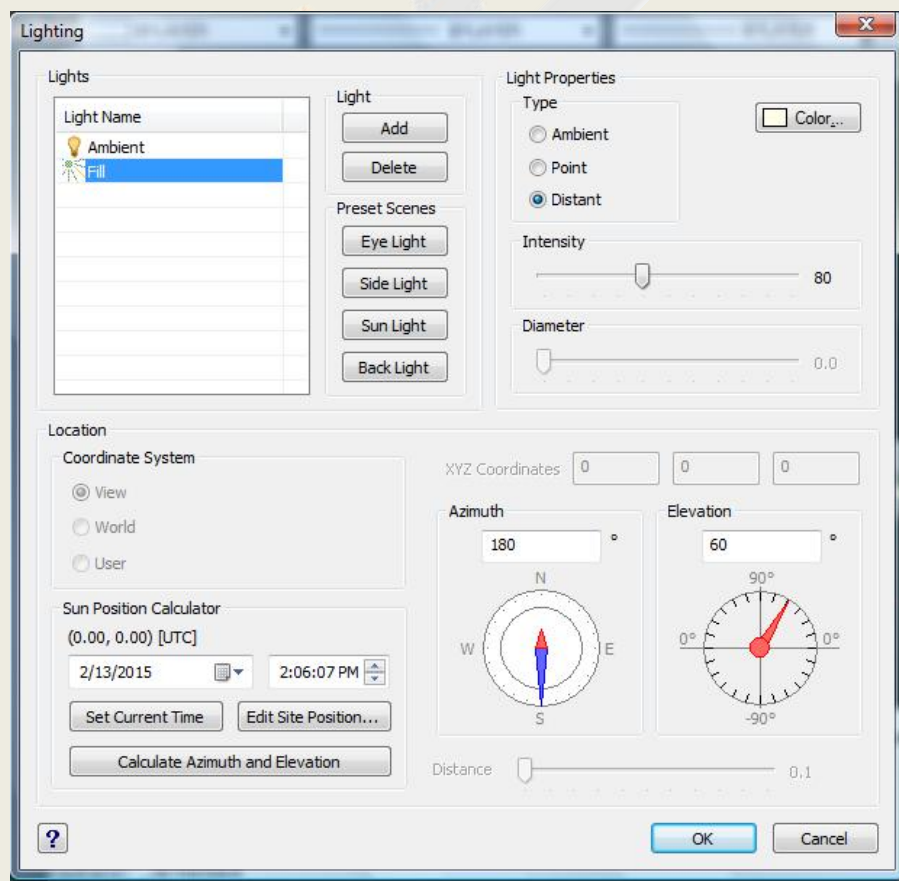


**Point:**

A point light represents light emanating from a specific point in (or out of) the drawing scene. Several additional parameters are available for the point light including its position, and its diameter. The diameter controls the size of the light source and hence the sharpness of the shadows cast by the light. A true point light source would have a diameter of zero and casts sharp-edged shadows. A larger diameter makes the light source into an area light source that will cast softer-edged shadows. Note that soft-edged shadows require additional rendering time

**Distance:**

A Distant light represents a light that is a large distance away from the surfaces in the drawing (e.g. the sun). All of the light rays are essentially parallel to each other so the shadows cast by the light have sharp edges. This option allows you to use a Sun Position Calculator that will calculate the position of the sun at any time of the day or year at any point on the earth's surface. With this type of light, the light's position is always calculated in the drawing's World Coordinate System. The Azimuth angle measures the compass bearing in the X-Y plane to the light and the Elevation angle measures the angle of the light relative to the X-Y plane horizon. By default, an azimuth angle of 0 degrees is in the +Y direction. However, this can be altered by adjusting the Site Orientation parameter on the Site Location tab of the Render Settings dialog.

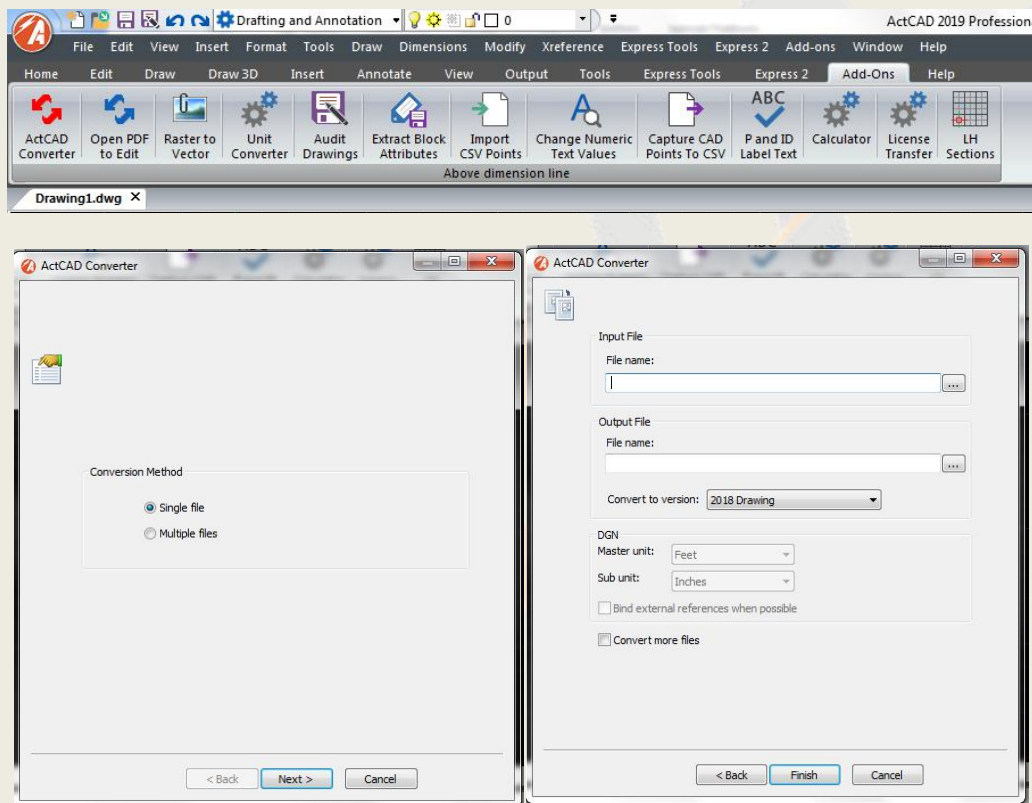


## Chapter 16: Addons

### ActCAD Converter:

ActCAD Drawings Converter works outside ActCAD to convert a bunch of dwg/dxf/dgn files from one format to another. This converter support all file versions from R2 to 2018. This conversion process is much faster compared to above tool and offers many additional functions.

It has an ActCAD converter that can convert DWG files to DXF or DGN file format in the add-ons tabs. You can either choose a single file or a number of files for conversion, after selecting you can select the location for storing the converted file format. Thus the required file format can be used according to the need. ActCAD supports DGN, DWG, DXF file formats. It can open DGN file and save them as DWG or DWG file

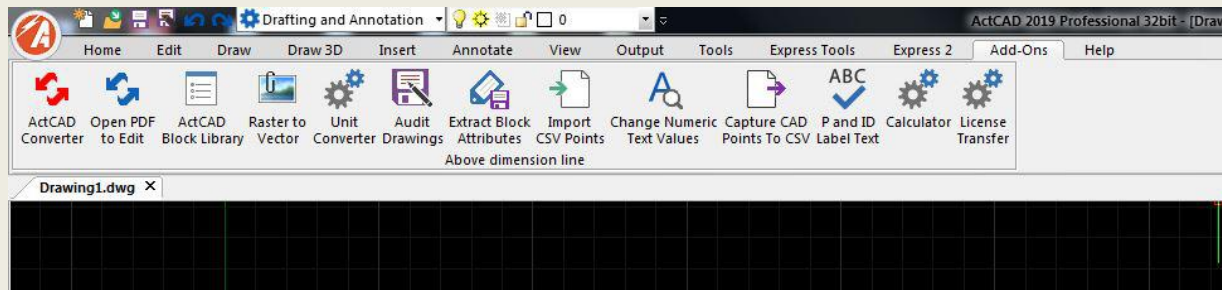


### Open PDF to Edit

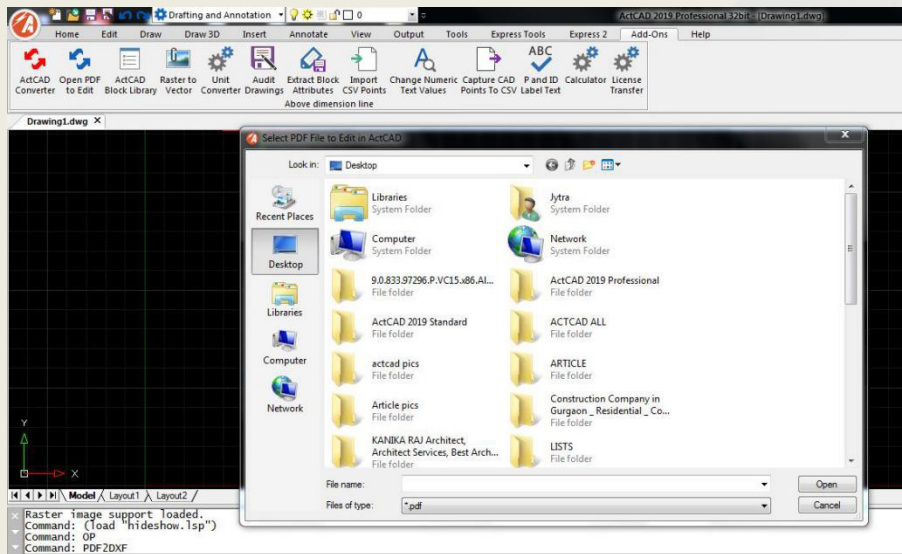
In ActCAD, you can open any existing vector PDF file directly and edit the entities just like a normal CAD drawing. All you have to do is to select the PDF file and click OK. ActCAD will convert the file to DXF at same location of original pdf and with same name. It will launch the converted drawing to ActCAD and fit it to graphics window. After that you can save the drawing to any of the supported dwg or dxf or any other file formats.

### Steps to convert PDF to DWG

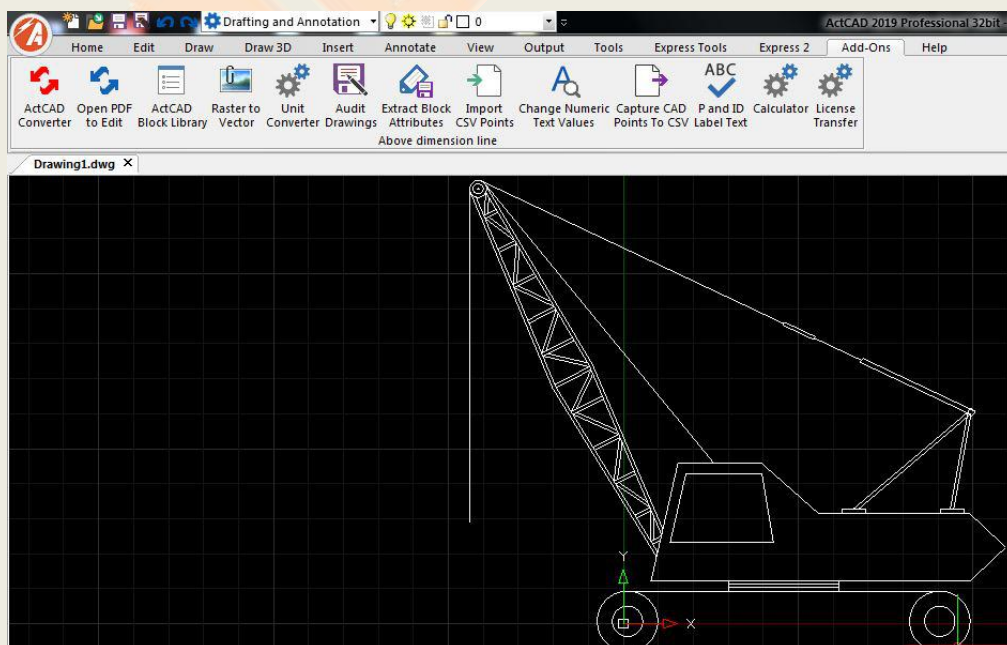
1. Ribbon --> Addon --> OPEN PDF to Edit.



## 2. Select the PDF file to be convert to DWG



## 3. After selecting the PDF file click on OPEN it will be converted to dwg.



## Raster to Vector

You will probably be able to load a raster file (like BMP, TIFF, PCX -file) into your CAD program - nowadays most CAD programs can import and display raster files. However, once you have imported a raster file into your CAD program you will find that all you can do is look at it or trace over it. You cannot change or edit it.

This is because CAD programs can only work with vector files. If you want to edit a raster file in your CAD program you must first convert it to a vector file. You do this using raster to vector conversion software. Once you have converted the raster file to vector you will be able to import it into your CAD program and edit it just like any other drawing you have created using your CAD program.

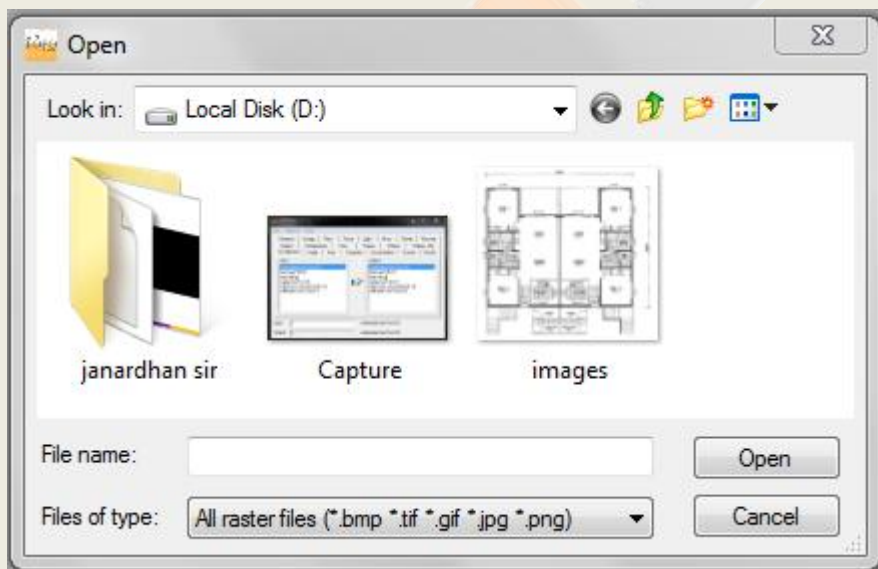
The list below summarises the process of converting a paper drawing to a CAD drawing using a raster to vector converter:

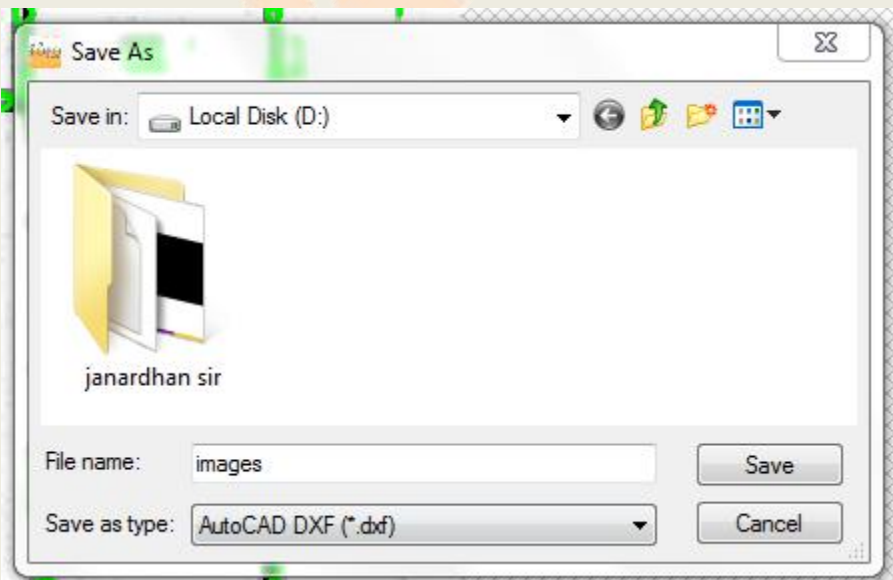
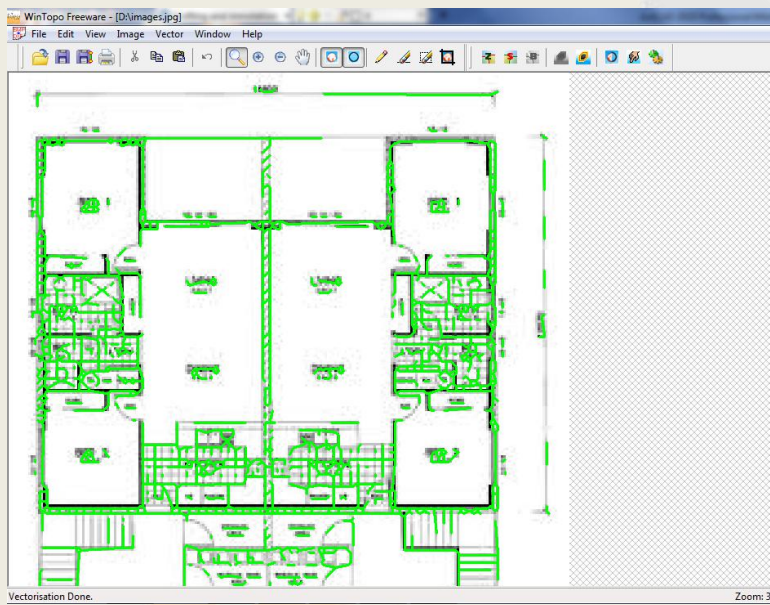
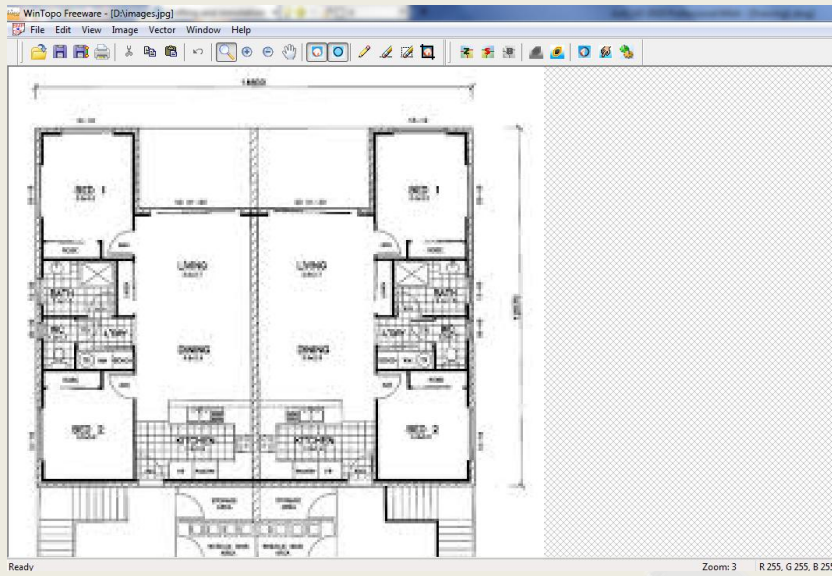
1. Create a raster file by scanning the paper drawing into RasterVect using a scanner.
2. Use RasterVect to convert the raster file into a vector file.
3. Import the vector file into your CAD program.
4. Edit the drawing in your CAD program.

RasterVect saves many hours of tedious work.

Process :

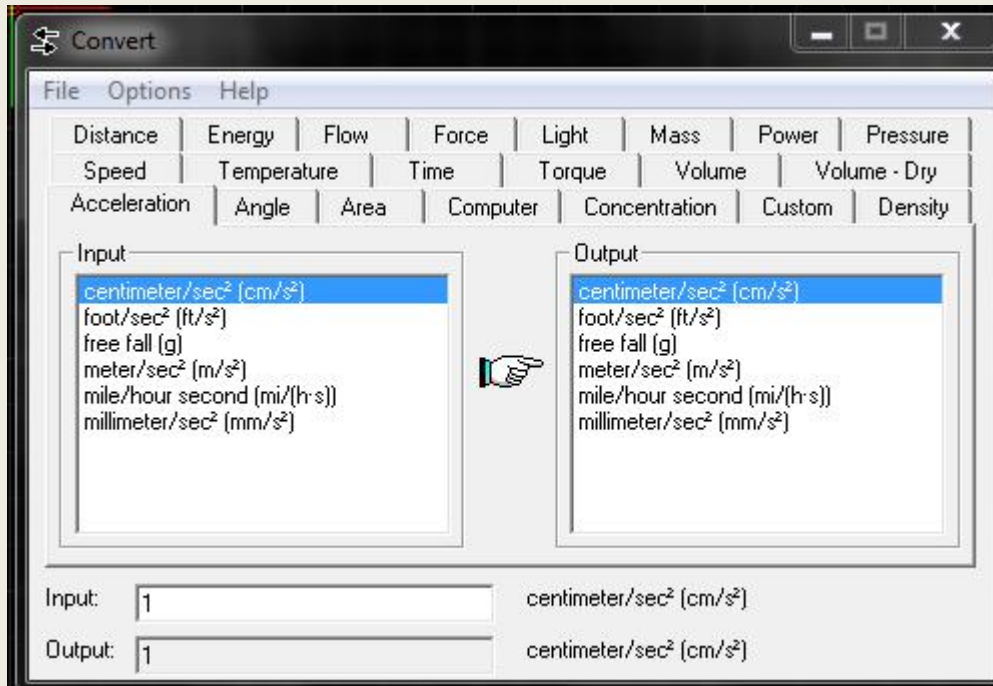
Select a file to convert Raster to vector and Click on OPEN





## Unit Converter

While working on design projects, you may have to convert some engineering units frequently. ActCAD has a ready tool inside add-on menu for easy unit conversions for all common engineering units.

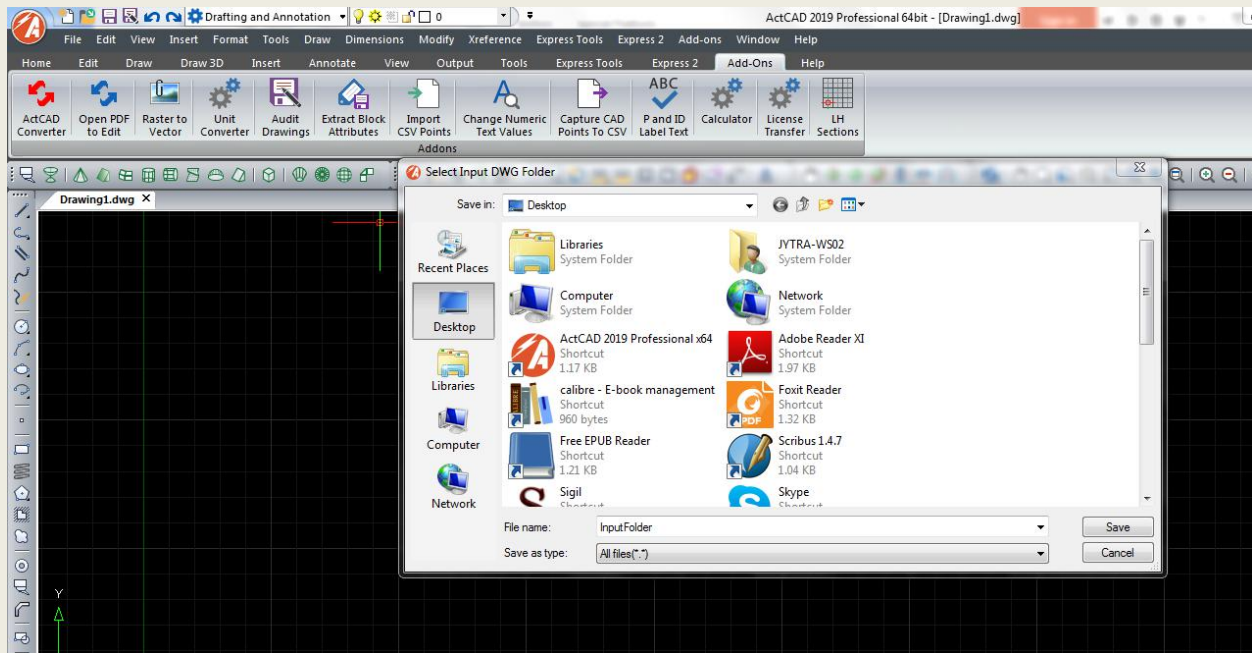


## Audit Drawings

ActCAD has a special tool to audit & fix any internal errors in your dwg/dxf files in batch mode. Select the location of input folder and start the process. ActCAD will open each drawing and do database audit and automatically fix any internal errors and save the drawing to output folder. The process will repeat in a loop to avoid all manual operations during the conversion process. Just relax and observe the conversion process.

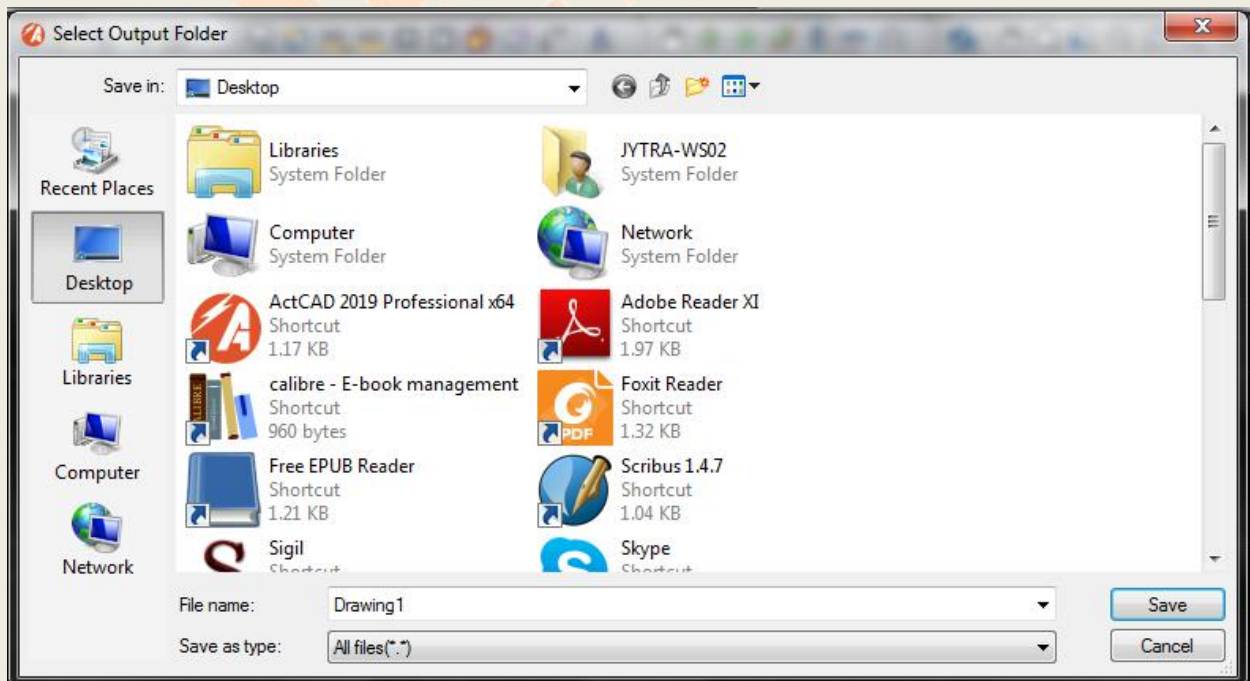
Your DWG file contains a database of all the information within your drawing. As you work on your drawing this database is constantly updated by adding new information and deleting old information.

Due to all this editing, errors can occur and need fixing before the DWG becomes corrupt and issues start appearing. In extreme cases you may find that your DWG won't open at all, if this is the case then you need to "recover" your drawing



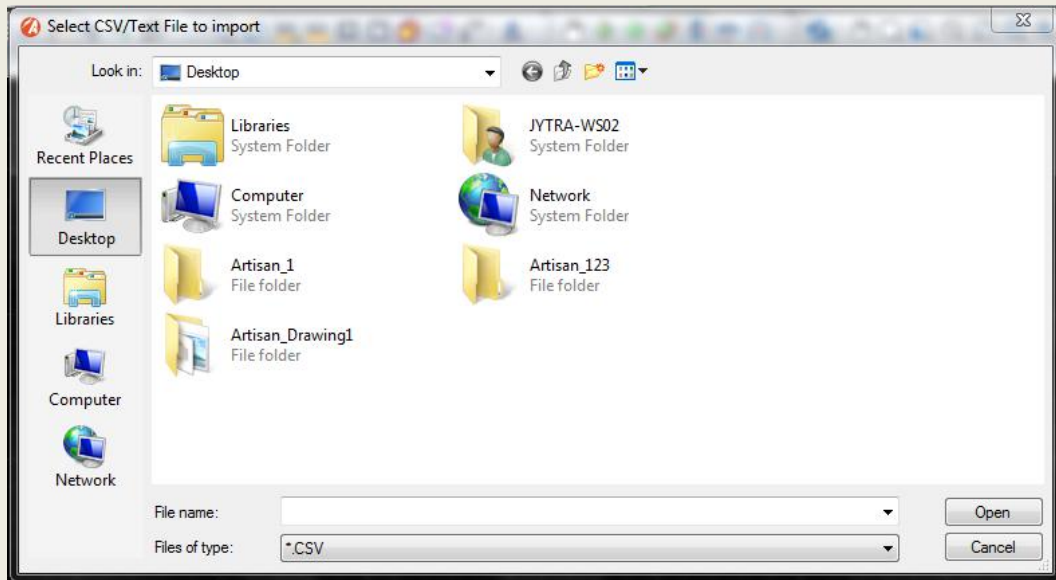
## Extract Block Attributes

This is another useful utility in ActCAD to extract all Block Attributes to an external Excel file with one button click. No need to select any complex templates and remember the syntax like regular data extraction tool. The attributes as in attributes editor will be copied to excel file in same sequence for ready use in other applications like SAP, BOM tables etc.



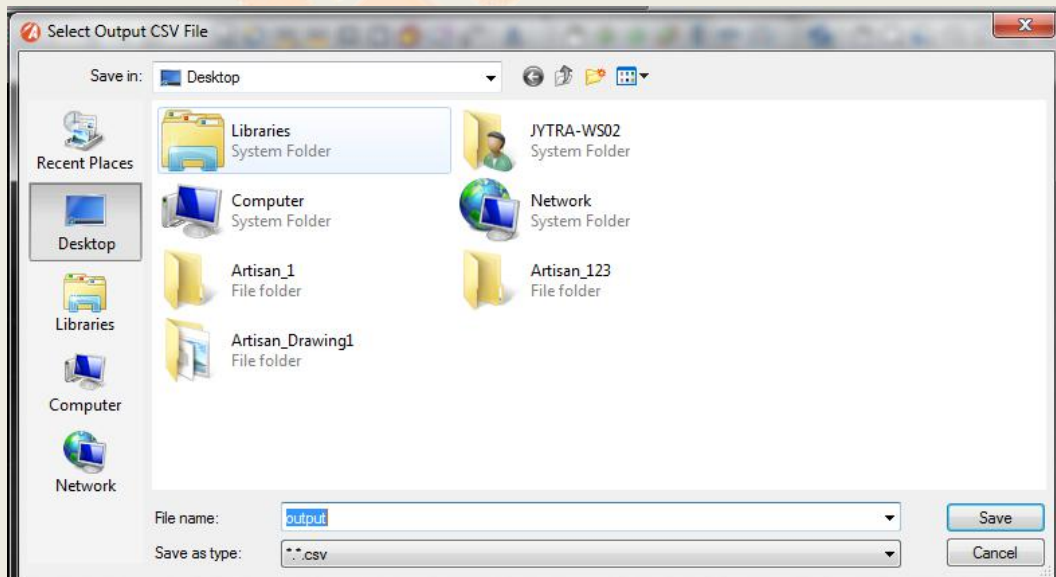
## Import CSV Points

Many times users have point cloud data stored in Excel/csv files. ActCAD has a provision to directly import all these points with one button click as geometric points to working drawing at correct 3D locations. You may have to adjust your point styles option to view the imported points inside your graphics window.



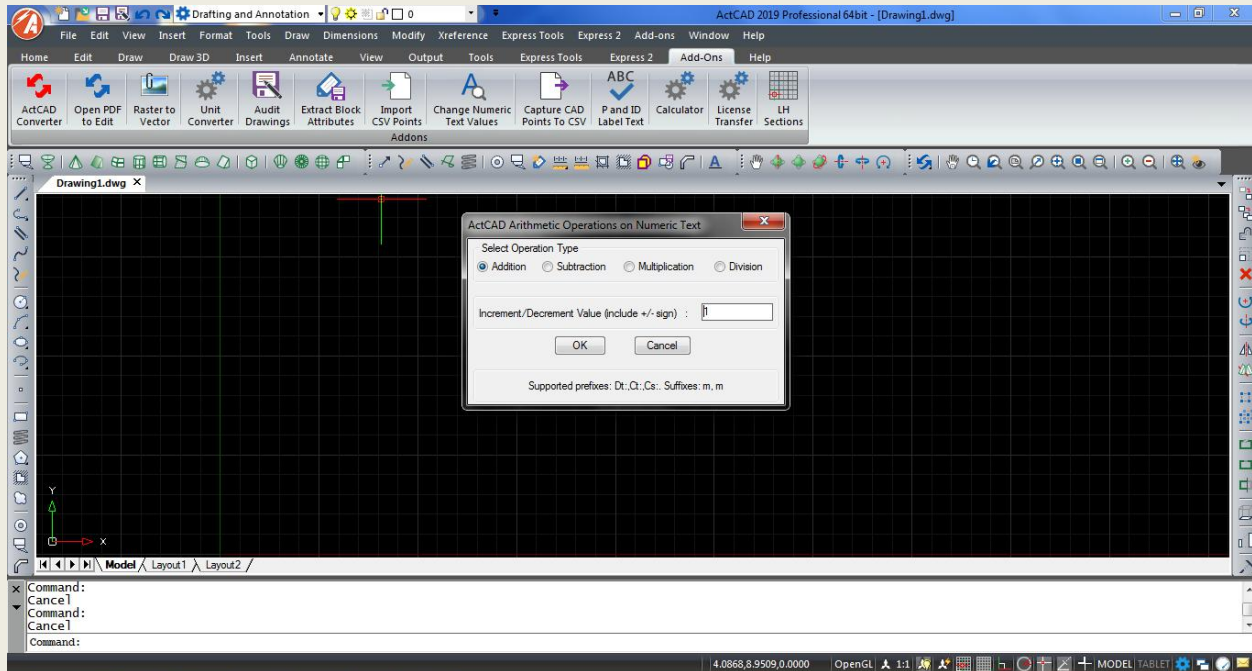
## Capture CAD points to CSV

This program is just opposite to above tool. You can click a series of points inside your drawing (like hole coordinates, road levels, block locations etc.) and their coordinates will be captured to external csv file. You can continue the selections as long as you need and program will continue to capture the points. In order to correlate the points data, ActCAD will add a serial number to each point inside the drawing and same numbers will be maintained in csv file.



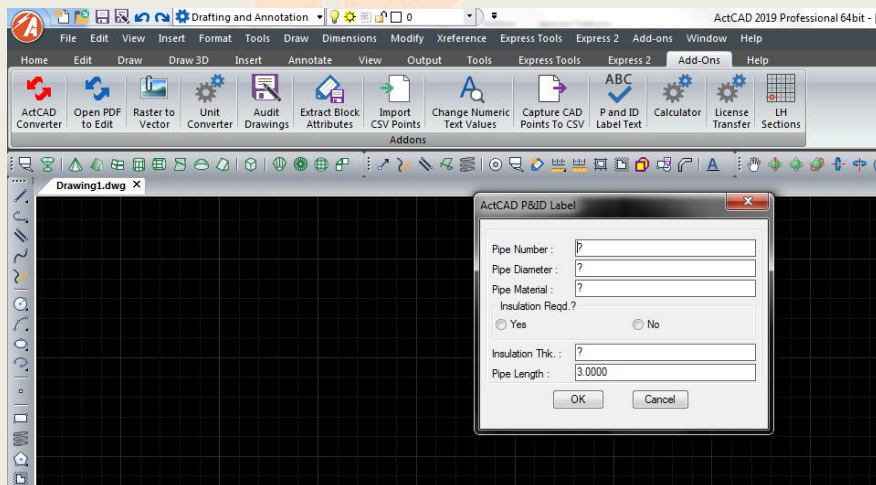
## Change Numeric Text Values

Sometimes in drawings, you may have to modify a series of number text with some modifications like addition, subtraction, multiplication, division etc. Manually editing each text will take lot of time. ActCAD has an in-built program to modify your texts in drawing in batch mode. You can set the operation and parameter before starting the change. This feature not only save your time but increase the accuracy and eliminate any human errors.



## P & ID Label Text Creator

ActCAD has a tool to create P&ID label text in standard format by asking few values from the user. You can orient the text at any angle as per pipe lines with few clicks.



## LH Sections

LH Sections is an additional and helpful tool for Civil Engineers and Surveyors who are having coordinates of the Land Surface and want to execute them in CAD Software as L Sections or H Sections. LH Sections feature in ActCAD needs your CSV File with coordinate inputs to access and draw the Section Profile accordingly. The user can configure its CSV Data with the user friendly interface.

	Value	Column Name
Position of Chainage (X) Column:	2	Chainage (m)
Position of G.L. (Y) Column:	3	G.L. (m)
Position of Invert (Y2) Column:	4	Invert Level (m)
Additional Column 1 :	6	Diameter (mm)
Additional Column 2 :	7	Slope (1 in N)
Position of Row Start:	5	Drawing Details Description: ???
Position of Row End:	53	
X (H) Scale Factor >>> 1:	1	
Y (V) Scale Factor >>> 1:	1	

Submit Values

Thank  
you