

# **GStarCAD 2009**

## **Tutorial**

**GreatStar Software**

**<http://www.staricad.com>**

## **Chapter 1 Introduction to GStarCAD**

### **1.1 Introduction to GStarCAD**

#### **1.1.1. system requirement**

You need the following software and hardware to install and run GStarCAD:

- Microsoft Windows 2000, 2003, XP, Vista (Windows2000, XP recommended);
- Intel Pentium(or faster) Processor ;
- Minimum 512M RAM (above 1G RAM recommended) ;
- More 300 MB Hard Disk(above 1G recommended);
- CD-ROM drive if installing from a CD;
- Mouse or other coordinates input devices;
- Color Monitor;
- Output Devices (Printer or Plotter).

#### **1. 1.2 Installation and Registry**

- **The installation of GStarCAD**

The GStarCAD installation program will lead you to finish the installation, the installation program will automatically create a folder which contains all programs, at the same time a program startup bar will be created at the “Start” menu of Windows, a shortcut icon will be created at desktop.

Users can use CD or copy installation program to your personal PC or the hard disc of server to install.

Install steps:

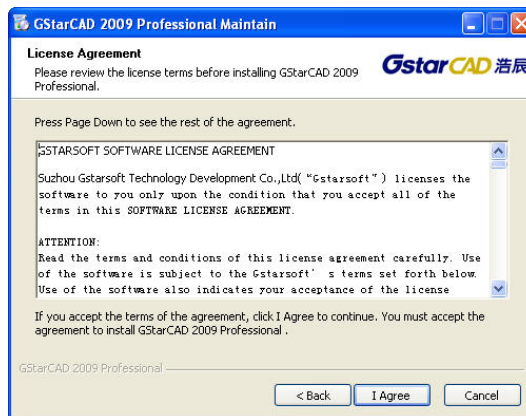
1.Enter the GStarCAD installation content, double-click the execute file, the installation setup interface is displayed and prompt that user system is preparing “Installation wizard” and do some preparation for installation.

2. After “Installation wizard” finishes installing, Installation wizard dialog is

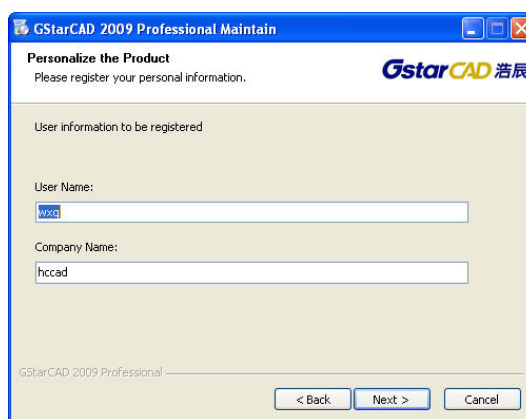
displayed. Click next button to continue installing.



3. “License Agreement” dialog box is displayed, as the below picture, if you agree with the agreement, click “I Agree” to continue

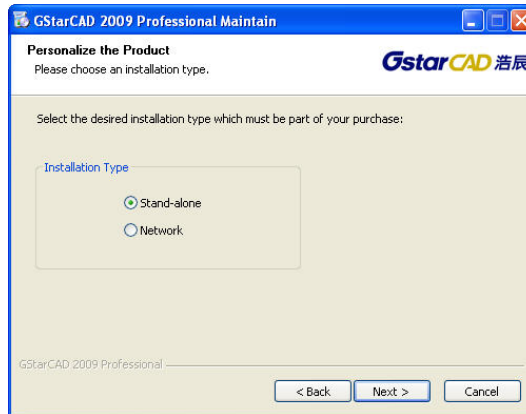


4. The user information dialog box is displayed, after you inputting user information, click “Next” button



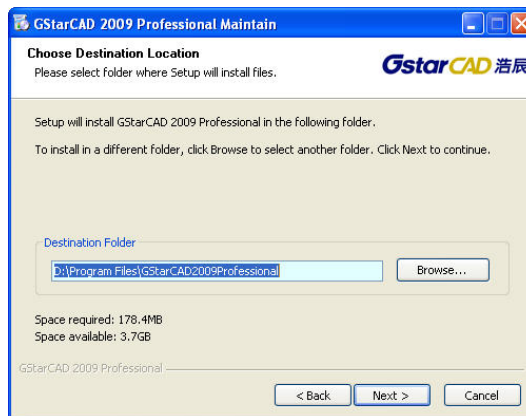
5. “Installation type” dialog box is displayed, as the below.

If you buy Stand-alone version or try software, you can use the default stand-alone to install; if you buy the network version, please select “Network” to install.

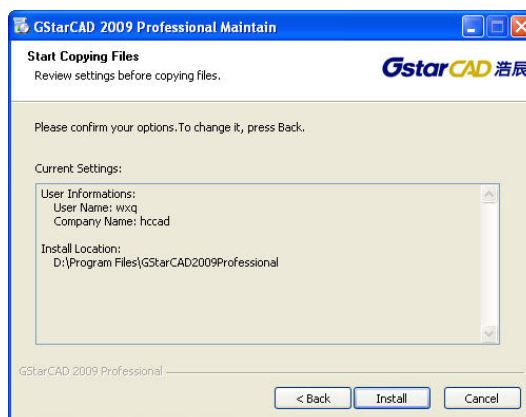


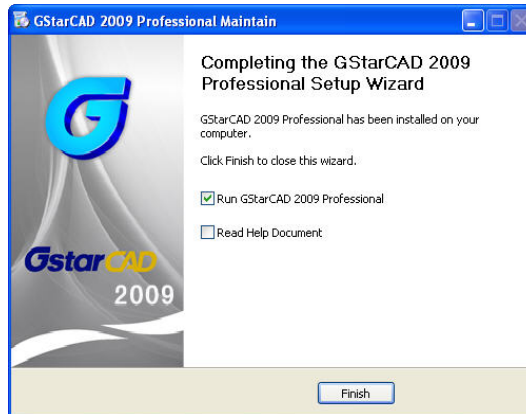
6. After you select the right installation type, click “Next” button, “Choose Destination Location” dialog box is displayed, as the below.

Click “Browse” button to select another folder.



7. Click “Next” button, “Start Copying Files” dialog box is displayed, user can check or modify the inputted information through this dialog box. If there are some information to modify, click “Back” button to return to the last dialog box to modify; If there are no information to modify, click “Next” button and the system begin to install the software.





Click “Finish” to complete the software installation

- **The registry of GStarCAD**

Click the icon on the desktop to start up GStarCAD. If you don't register software, at the title bar there is tryout software prompt

There are two authorization methods: Dongle and Registration Number

### 1、Dongle

Dongle is divided into stand-alone version and network version.

A stand-alone version belongs to a single customer, you needn't install driver, software will finish the register automatically.

Network version should be set up on the server (any computer in network), but the number of users should limit within the authorized number by the network lock.

If you install network version please select any a computer as the server, and install network lock driver at this server computer(the network lock driver is in the Driver content). When you install GStarCAD at customer end, please select “Network”, once you start CAD, and the software will automatically search for network lock to finish registering. If the software is still in the tryout situation, please input the server name or IP address at the program item of GStarCAD. If there still doesn't work well please check firewall settings or contact with us.

If you want to use network lock at different net, you should set a system variable at your customer end, the steps is below:

Start>Control panel>System>Advanced>Environment Variables>New a environment variable, the variable name is "NSP\_HOST", the variable value is the IP address of the

Server

### 2、Registration Number

The operation step about registration number is below:

- 1). Install the stand-alone GStarCAD
- 2) Click Help>Register, GStarCAD will generate a serial number, this serial number is corresponding to your computer



- 3) You should send the serial number to our company, we will calculate a activation number to you according your serial number.

Note: The tryout limit is 30 days, there is no function limitation in the tryout days, but you had better complete the registration before tryout expiration

- 4). Open “Activation” dialog box, input the activation number, then click Activate button to finish register
- 5). Restart GStarCAD, and check whether the software has been activated.

### 1.1.3. Learn About New Features in GStarCAD2009

The New Features Workshop introduces you to what's new in GStarCAD2009.

#### 1. Strong Re-development Environment:

##### ● GRX Programming Interface

GRX programming interface is provided which is compatible with ARX, based on C++, object-oriented re-development environment and application interface. GStarCAD2009 support all the development interface: AutoLISP, Visual LISP (including IDE), SDS (ADS), VBA, GRX (ARX)

##### ● VBA development interface

Microsoft VBA (Visual Basic for Application) is an object-oriented programming environment designed to provide rich development capabilities, which is a strong development tool. GStarICAD2009 also integrates VBA development tool, and provides ActiveX Automation suitable to VBA development

- **GStar Visual LISP IDE**

GStarCAD2009 provides the customers who are accustomed to using LISP with a visual development interface and environment. The VLISP integrated development environment (IDE) provides features to help ease the tasks of source-code creation and modification, program testing, and debugging. In addition, VLISP provides a vehicle for delivering standalone applications written in AutoLISP.

### **2. More Strong Print Functions:**

- HDI upgrades to 3.0 version. Re-developer can customize print interface in GStarICAD2009.
- We cooperate with Canon to develop Canon series print drivers
- Add DWF virtual print driver and realize the web publish function.
- Arrange drawings automatically.
- PLT output tool: Users can send PLT files to printer to print.

### **3. Perfect Management Functions:**

- New Drawing Wizard: User can select modes and also can select required template files
- Layer Translator: Change a drawing's layers to layer standards you specify.
- View Manager: Create, set, rename, modify, and deletes named views, and switch quickly between each views, It is helpful to reduce many unnecessary view adjust operation
- Alias Edit: GStarICAD2009 provides executable file “aliascommand.exe”. User can modify alias and need not to enable GStarCAD. In GStarCAD2009 user can customize system command such as: run、cmd、start and etc.
- Block Attribute Manager: Manage the attributes for a selected block definition, preview modification automatically, it is convenient to modify attribute block in

quantity.

- Block Count: User can know the block name and the number of inserted block in the selected object. It is compatible with the Bcount command in AutoCAD.
- The Upgrade of Design Center and Tool Palettes:

The blocks in design center can be dragged to drawing area; the blocks in drawing area can be dragged to tool palettes too. The input and output function of tool palettes is helpful to share and reuse.

#### **4. The Enhancement of Operation:**

- Dynamic Input
- Enhancement of Selection Function: window selection and crossing selection
- The Drag of Right Mouse Key

Use right mouse key to drag an entity after selecting the entity, the shortcut key is displayed. The menu contains: move here, copy here, paste as block, cancel. It is a new and easy operation mode.

- Text to Front: Bring text and dimensions in front of all other objects in the drawing
- **Quick Calculator**

Evaluate mathematical and geometric expressions. Different from common calculator, quick calculator is an expression generator. In order to get more flexibility, quick calculator will not give the answer immediately but let user input a editable expression then press Enter or equal sign to get the result. In addition, user can find the inputted expression from history record, and modify it or recalculate.

- **HaoChen Tools**

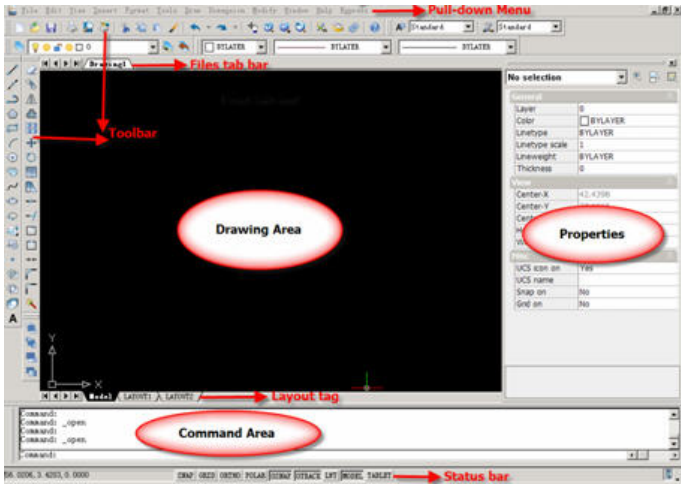
HaoChen Tools integrated many convenient drawing tools, user can use GStarCAD platform software to finish many professional design such as pipeline design.

## **1.2. Starting up GStarCAD**

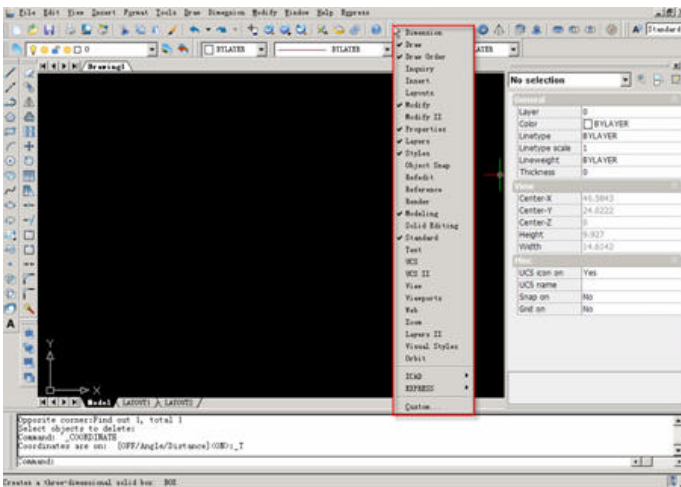
### **1.2.1 Working in GStarCAD**

#### **(1) Initial Interface of GStarCAD**





- Once users press right-button of mouse on Toolbar, Toolbar list will be displayed.
- Then select a Toolbar that users want to display on the screen.



(2) Introduce each kind of symbol agreement in GStarCAD commands:

- “/”: The Division symbol divide up different options of the command, each option

capital letter expressing abbreviation way, you can input this letter to execute this option directly.

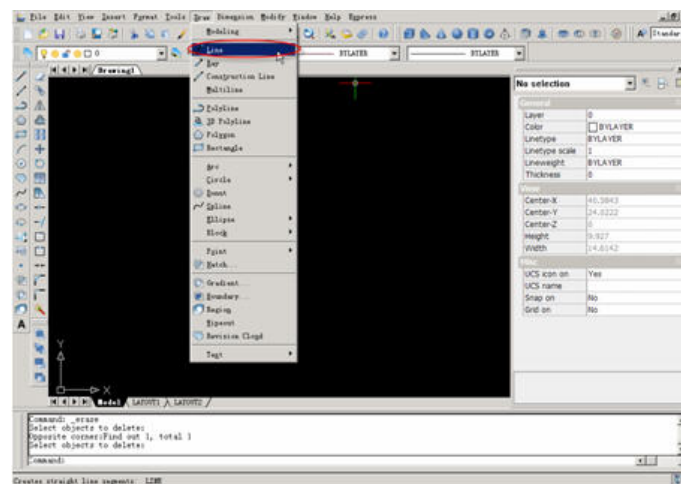
- “<”: The small parenthesis, imply the default input value or current execute the option in this parenthesis, if not meet to your requirement, then input the new value.
- After executing some commands, you can use “Enter”, Space Bar or right-button of mouse to repeat the command.

### 1.2.2 Using Commands

GStarCAD provides several methods to execute the command. Here is an example of “Drawing Line”.

#### (1) Pull-down Menu

Select “Draw >Line” in Pull-down Menu.



#### (2) Toolbar

Select the “Line” in the “Drawing” Toolbar.



#### (3) Input commands or shortening commands (Hotkeys)

Command: LINE

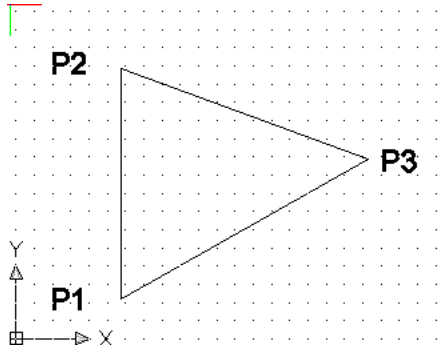
Start point of line: (pick P1)

Angle/Length/< End point>: (pick P2)

Angle/Length/Follow/Undo/< End point>: (pick P3)

Angle/Length/Follow/Undo/< End point>: (pick P1)

Angle/Length/Follow/Undo/< End point>: (Enter)



### Use function key

- F6 key (COORDINATE)

F6 key is used for ON/OFF the coordinate value. Once you press F6 key, coordinate value is ON. If you press it again, coordinate is OFF. When F6 key is ON, coordinate value of the crosshair is displayed on the status bar of lower screen.



- F7 key (GRID)

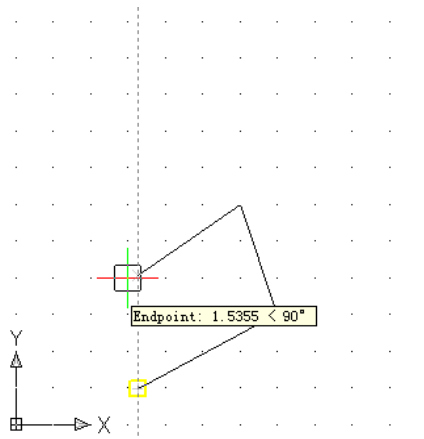
F7 key is used for ON/OFF the GRIP. When GRIP is ON, uniform grids are displayed on the screen

- F8 key (ORTHO)

F8 key is used for ON/OFF the Ortho function. When Ortho function is ON, a crosshair is moved only horizontally or vertically.

- F9 key (SNAP)

Snap function can be used only when Grid is ON. When Snap is ON, the mouse always moves on grids. When SNAP is OFF, the mouse moves randomly.

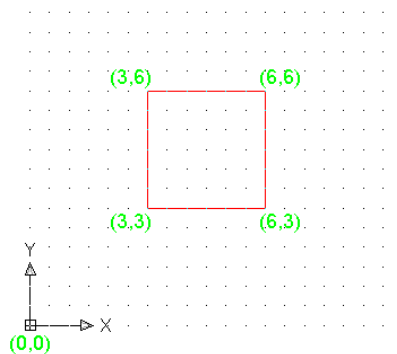


### 1.2.3 Working with Coordinates

#### (1) Absolute coordinates

In the GStarCAD, origin (0, 0) is always at lower-left corner of the screen. Absolute coordinates are used to input the coordinate value based on the origin. When you know the absolute coordinates of the drawing entity, you use it.

(Input method) X coordinate value, Y coordinate value.



Command: `_LINE`

Start point of line: 3,3

Angle/Length/<End Point>: 6,3

Angle/Length/Follow/Undo/<End Point>: 6,6

Angle/Length/Follow/Undo/<End Point>: 3,6

Angle/Length/Follow/Undo/<End Point>: 3,3

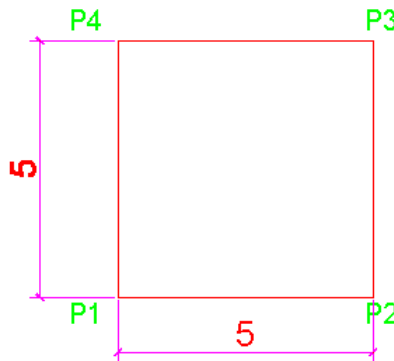
Angle/Length/Follow/Undo/<End Point>:

#### (2) Relative coordinates

Relative coordinates are used to input the incremental value of X and Y based on the

last selected point instead of the origin.

(Input method) @ X Incremental value, Y incremental value



Command: \_LINE

Start point of line: (pick P1)

Angle/Length/<End Point>: @5,0

Angle/Length/Follow/Undo/<End Point>: @0,5

Angle/Length/Follow/Undo/<End Point>: @-5,0

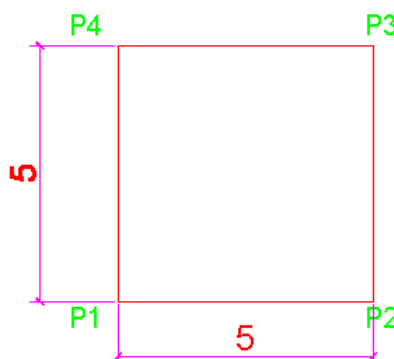
Angle/Length/Follow/Undo/<End Point>: @0,-5

Angle/Length/Follow/Undo/<End Point>:

(3) Relative polar coordinates

Relative polar coordinates are used to specify the point by using the distance and angle based on the reference point.

(Input method) @distance<ANGLE



Command: \_LINE

Start point of line: (pick P1)

Angle/Length/<End Point>: @5<0

Angle/Length/Follow/Undo/<End Point>: @5<90

Angle/Length/Follow/Undo/<End Point>: @5<180

Angle/Length/Follow/Undo/<End Point>: @5<270

Angle/Length/Follow/Undo/<End Point>:

## Chapter 2 File (I/O) Commands

### 2.1 New/ Newwiz

#### 1. NEW (Open a new drawing)

NEW command opens new drawing sheet on the screen as the preparing step of drawing.

Use “C:\ProgramFiles\GreatStarICAD 2008 Professional\ICAD” as a template drawing.

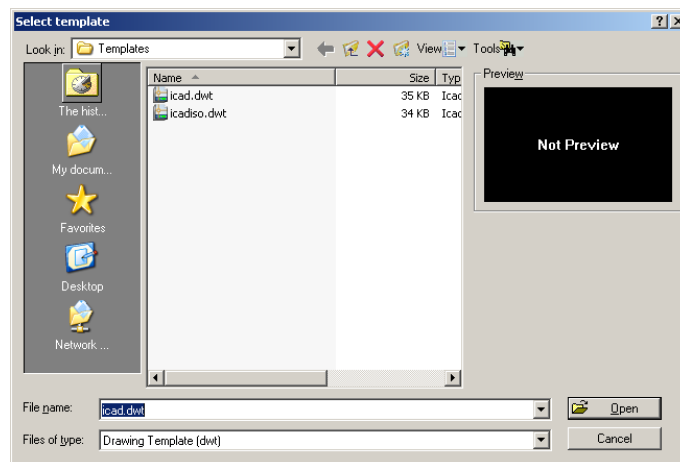
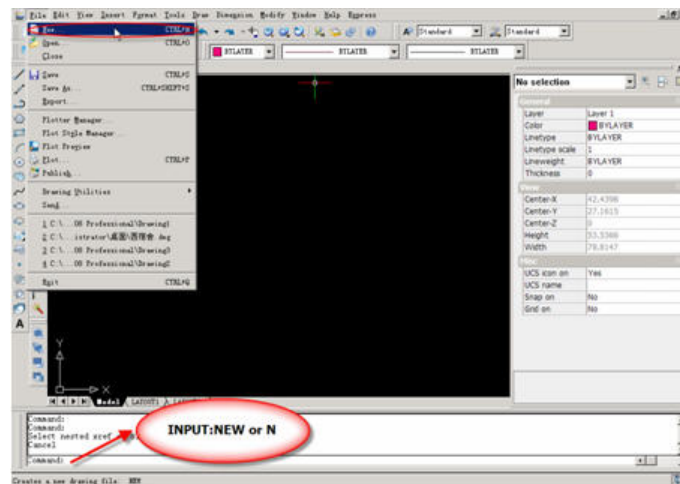
(1) Execution

Hot key: Ctrl+N

Pull-down menu: File >New

Commands: NEW or N

Toolbar: 



(2) What's TEMPLATE drawings?

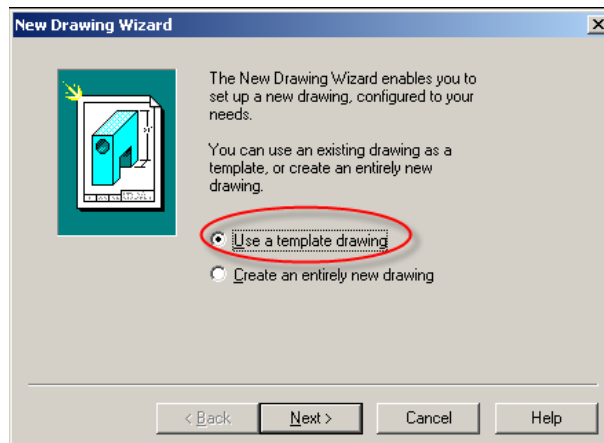
When you are drawing, you always make a border-line and a title which are written in the drawing sheet. If you make them in advance and save as “C:\ProgramFiles\GStarCAD 2008 Professional\\Templates ICAD.dwt”, you don’t need to make them again because pre-defined ICAD.dwt is automatically opened whenever the GStarCAD is started. This kind of drawing sheets that are made by using NEWWIZ command, are called as a TEMPLATE.

### 2. NEWWIZ

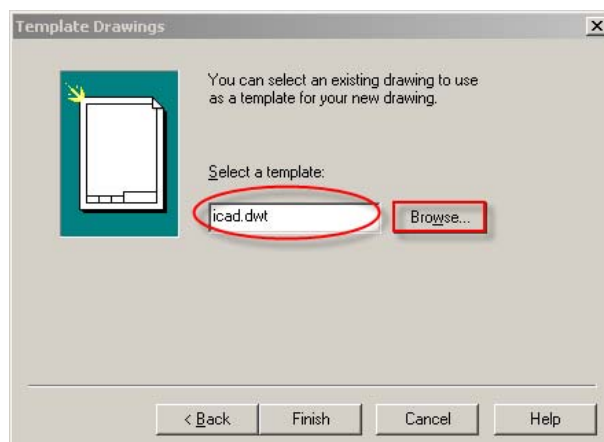
NEWWIZ command makes designers setup the specific drawing sheet as TEMPLATE. Or by using wizard functions, designers can setup environments of the drawing sheet such as the sheet size, unit, text size, etc.

Command: NEWWIZ

#### (1) The method of using TEMPLATE



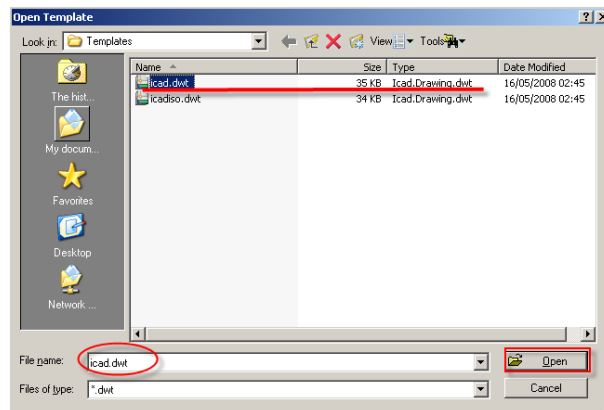
- After selecting “Use a template drawing”, click “Next”.



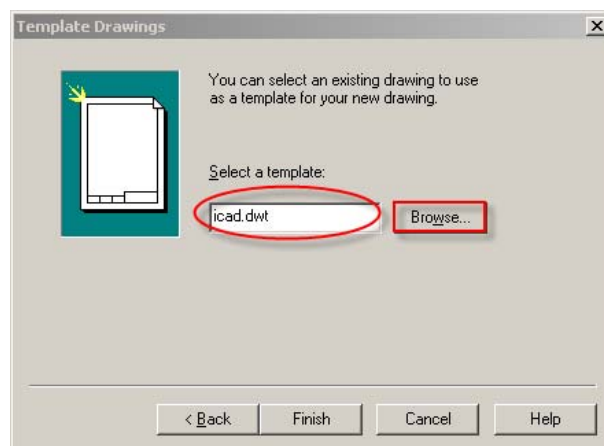
- Currently default TEMPLATE is set to icad.dwt. If the users want to set other



drawing sheet as TEMPLATE, click “Browse” button.

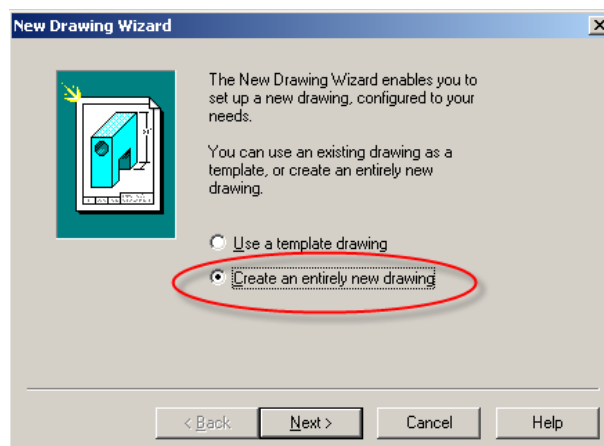


- Double-click the specific file as a TEMPLATE drawing sheet, or click the file and click “Open” button.

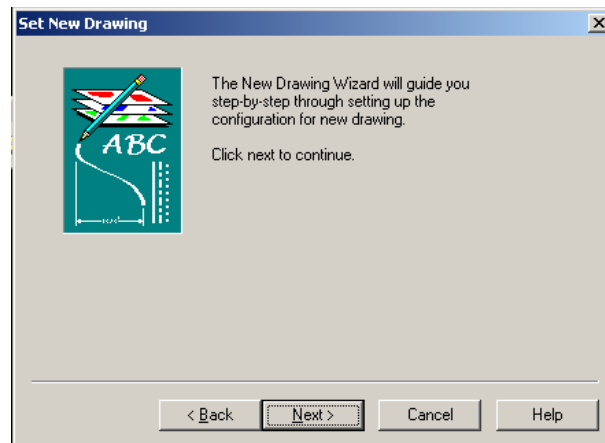


- Click the “Finish” button.

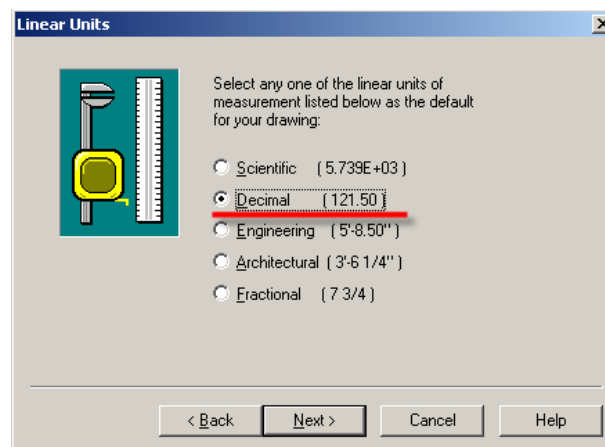
### (2) The method of using wizard



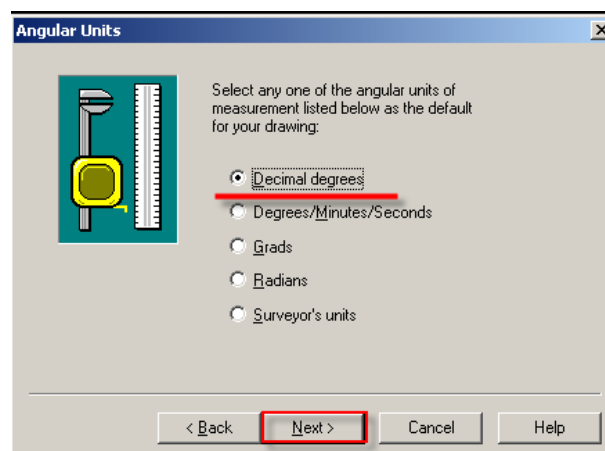
- After selecting “Create an entirely new drawing”, click the “Next” button.



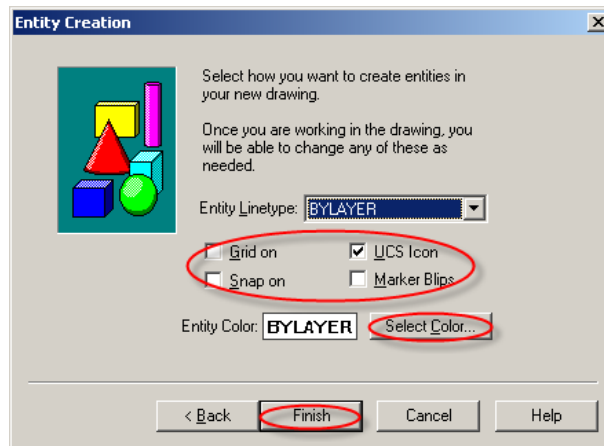
- Click the “Next” button.



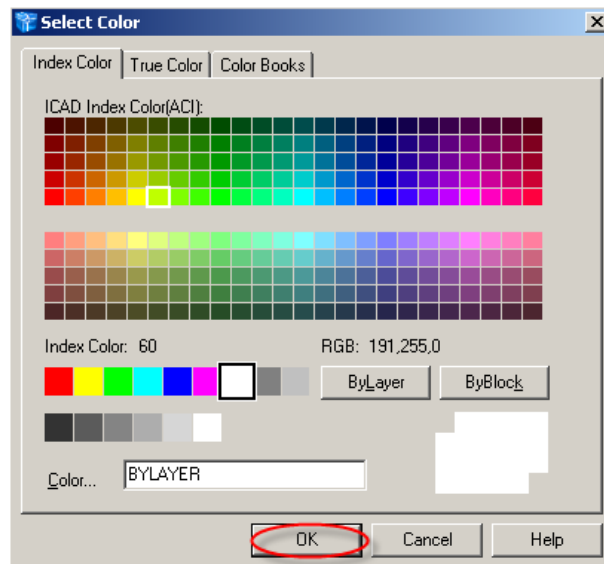
- Select linear units for drawing. Usually “Decimal” is default.
- Click the “Next” button.
- Select angular units for drawing. Usually “Decimal” is default.
- Click the “Next” button.



- Click the “Next” button.



- Set ON/OFF of Grid, Snap, UCS Icon and Blips functions.
- Select the line-type to be used for drawing in “Entity Linetype”.
- Once you click the “Select Color” button, select the color to be used for drawing.



- Click “Finish” button and end the wizard.

## 2.2 Open

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

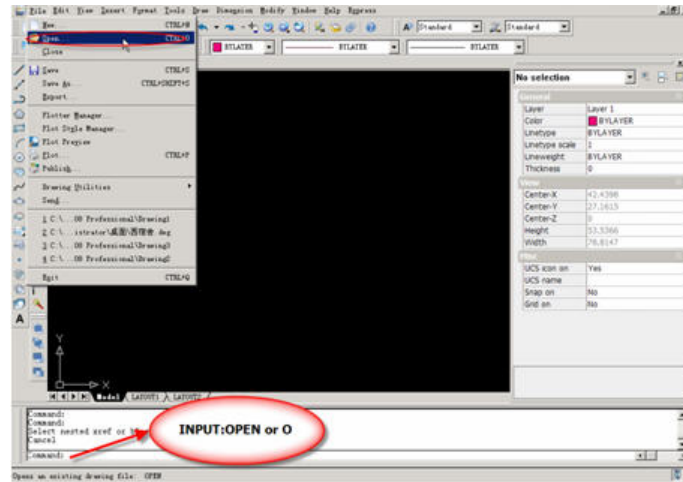
You can also open and check files that you suspect whether it was damaged.

Hot key: Ctrl+O

Command: OPEN or O

Pull-down menu: File> Open

Toolbar:



### 2.2.1 Open a existing drawing

#### Open an existing drawing

Use one of the following methods:

Pull-down menu: File > Open.

Toolbar:



Command: OPEN or O

Choose the type of file you want to open.

Choose the folder containing the desired file.

Do one of the following:

- Choose the drawing you want to open, and then click Open.
- Double-click the drawing you want to open.

If the drawing requires a password, enter the password, click OK to verify the password, and then click Open again.

### 2.2.2 Open damaged drawings

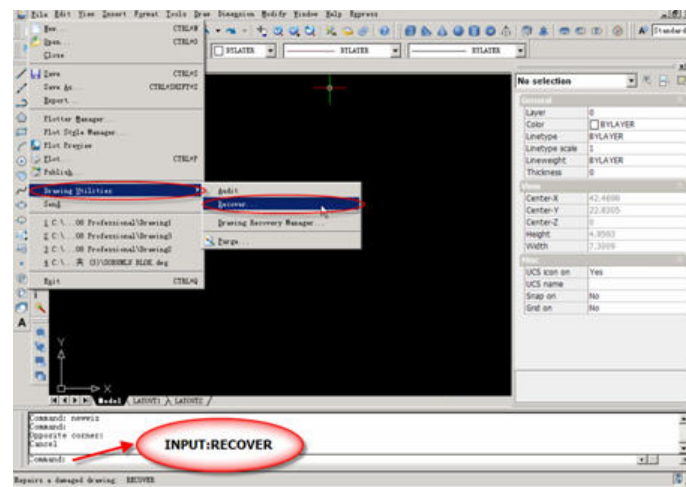
Files may be damaged for many reasons. GStarCAD allows you to check and recover damaged files.

#### Open a damaged file

Use one of the following methods:

Pull-down menu: File >Drawing utilities >Recover.

Command: RECOVER



Choose the type of file you want to recover.

Choose the directory containing the damaged file.

Choose the damaged file you want to recover.

Click Open.

To check a drawing file for errors

If you want to check the opening drawing, do one of the following:

Pull-down menu: File >Drawing utilities> Audit.

Command: AUDIT

Choose whether you want GStarCAD to fix any found errors automatically, and then press Enter.

## 2.3 Save/ Qsave/Save As

SAVE or SAVEAS commands make users save the file during working or after finishing drawing. When you save a drawing, your work is saved in a drawing (.dwg) file.

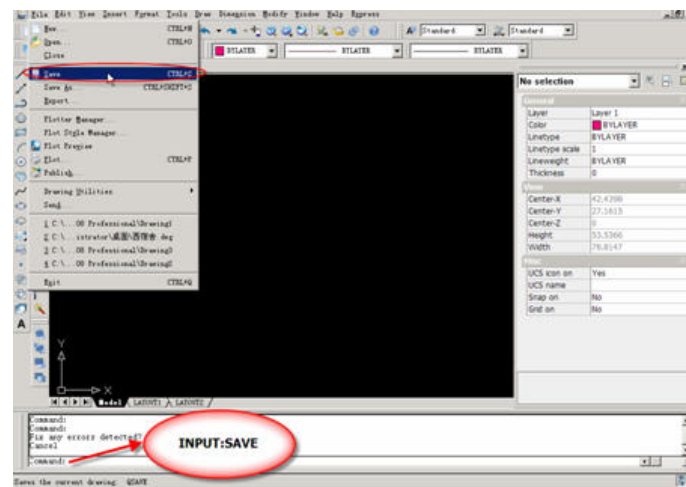
### 2.3.1 Save a drawing

Hot key: Ctrl+S

Pull-down menu: File>Save

Toolbar: 

Command: SAVE



- If you have saved your drawing for the first time, you can save it with a new name. In addition to drawing (.dwg) files, you can also save a drawing in a Drawing Exchange Format (.dxf) file or a drawing template (.dwt) file.
- If you created your drawing using a template, saving the drawing does not alter the original template.
- If you need protect your drawing files, you can save the drawing with a password so only you and those who know the password can open the drawing.

### 2.3.2. Save a drawing with a new name or file format

Hot key: Ctrl+Shift+S

Pull-down menu: File> Save as

Command: SAVE AS

Specify the name of the file you want to create

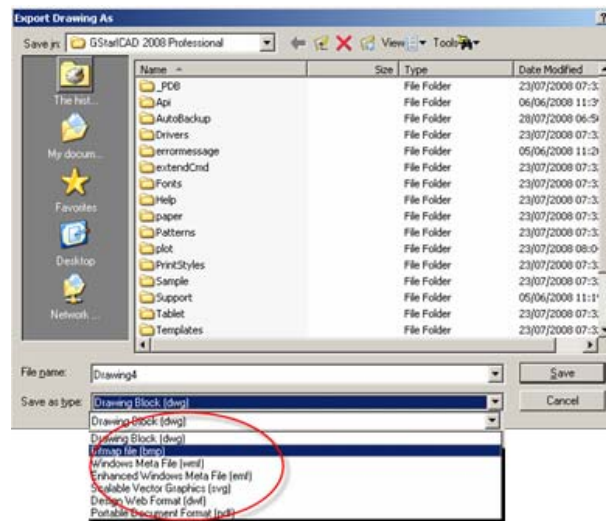
In the “Save drawing As” dialog box, choose the file format.

## 2.4 Export

EXPORT command is used to save the current drawing as a new file name or other file types. Usage is the same with the “SAVE” dialog box.

Pull-down menu: File> Export

Command: EXPORT



### Chapter 3 Draw Geometric Objects

Drawings are made up of objects. In general, you draw objects by specifying points with the pointing device or by entering coordinate values at the command prompt.

You can create a range of objects, from simple lines and circles to spline curves, and ellipses. In general, you draw objects by specifying points with the pointing device or by entering coordinate values at the command prompt.

- Draw Linear Objects

A line, the most basic object, can be one segment or a series of connected segments.

- Draw Curved Objects

Curved objects are arcs, circles, polyline arcs, donuts, ellipses, and splines.

- Draw Construction and Reference Geometry

Construction lines and reference points are temporary objects you create to help you draw accurately.

- Create and Combine Areas (Regions)

Regions are two-dimensional enclosed areas that have physical properties such as centroids or centers of mass. You can combine existing regions into a single, complex region to calculate area.

- Create Revision Clouds

Revision clouds are polylines that consist of sequential arcs. They are used to call attention to parts of a drawing during the review stage.

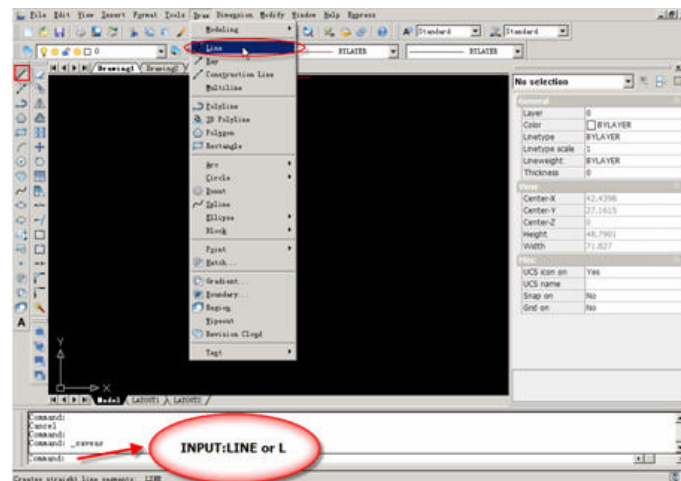
#### 3.1 Line

Pull-down-menu: Draw >Line

Command: LINE or L

Toolbar: 





With the Line command you can draw a simple line from one point to another. When you *pick* the first point and move the crosshair to the location of the second point you will see a *rubber band line* which shows you where the line will be drawn when the second point is picked. Line *objects* have two ends (the first point and the last point). You can continue picking points and GStarCAD will draw a straight line between each picked point and the previous point. Each line segment drawn is a separate *object* and can be moved or erased as required. To end this command, just press the Space bar or Enter key on the keyboard.

Command: line

Specify first point: (pick P1)

Specify next point or [undo]: (pick P2)


Specify next point or [undo]: ↵ (to end)



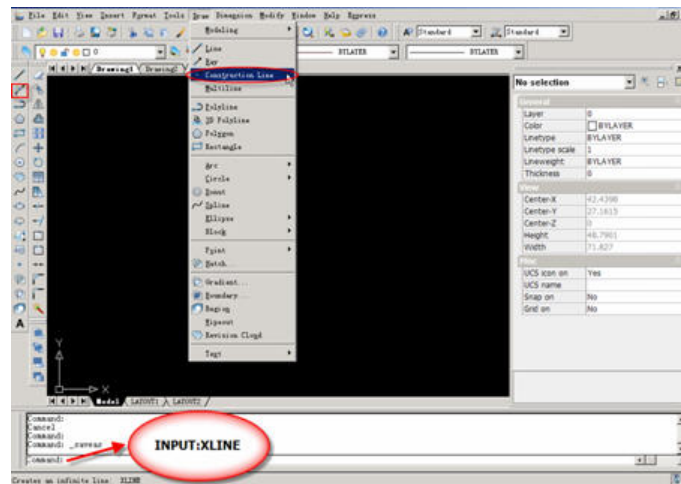
You can also draw lines by entering the co-ordinates of their end points at the command prompt rather than picking their position from the screen . This enables you to draw lines that are off screen, should you want. You can also draw lines using something called direct distance entry.

### 3.2 Construction line

Pull-down menu: Draw >Construction line

Toolbar: 

Command: Xline



The Construction Line command creates a line of infinite length which passes through two picked points. Construction lines are very useful for creating construction frameworks or grids within which to design.

Construction lines are not normally used as objects in finished drawings, it is usual, therefore, to draw all your construction lines on a separate layer which will be turned off or frozen prior to printing. Because of their nature, the Zoom Extents command option ignores construction lines.

#### Command Sequence

Command: XLINE

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: (pick a point)

Specify through point: (pick a second point)

Specify through point: (to end or pick another point)

You may notice that there are a number of options with this command. For example, the "Hor" and "Ver" options can be used to draw construction lines that are truly horizontal or vertical. In both these cases, only a single pick point is required because the direction of the line is predetermined. To use a command option, simply enter the capitalised part

of the option name at the command prompt. Follow the command sequence below to see how you would draw a construction line using the Horizontal option.

### Command Sequence

Command: XLINE

Hor/Ver/Ang/Bisect/Offset/<From point>: H

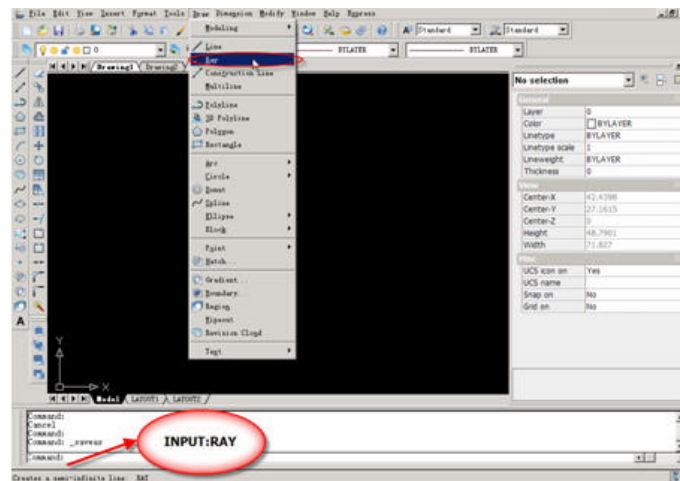
Through point: (pick a point to position the line)

Through point: (to end or pick a point for another horizontal line)

### 3.3 Ray

Pull-down menu: Draw >Ray

Command: RAY



The Ray command creates a line similar to a construction line except that it extends infinitely in only one direction from the first pick point. The direction of the Ray is determined by the position of the second pick point.

### Command Sequence

Command: RAY

Specify start point: (pick the start point)

Specify through point: (pick a second point to determine direction)

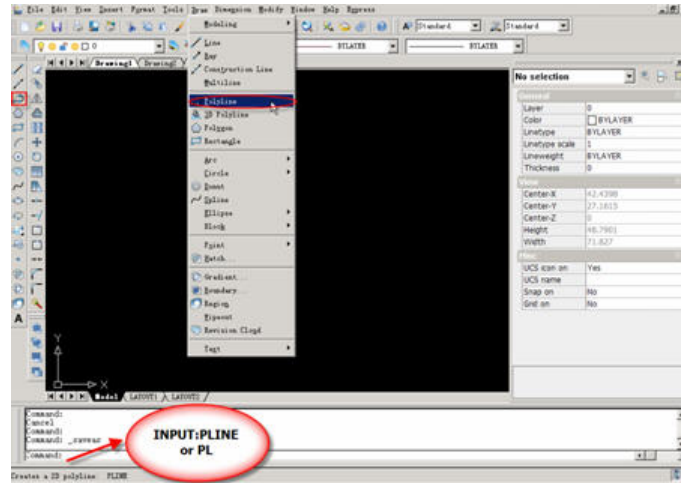
Specify through point: (to end or pick another point)

### 3.4 Polyline

Pull-down menu: Draw >Polyline

Toolbar: 

Command: Pline or PL



A polyline is a connected sequence of segments created as a single object. You can create straight line segments, arc segments, or a combination of the two. In practice the Polyline command works in the same way as the Line command allowing you to pick as many points as you like. Follow the command sequence below to see how this works.

Polylines are ideal for applications including the following:

- Contour lines for topographic, isobaric, and other scientific applications
- Wiring diagrams and printed circuit board layouts
- Process and piping diagrams
- Extrusion profiles and extrusion paths for 3D solid modeling

Polylines can be created with several commands including PLINE, RECTANG, POLYGON, DONUT, BOUNDARY, and REVCLOUD. All of these commands result in a LWPOLYLINE (lightweight polyline) object type. Command Sequence

Command: PLINE

Specify start point: (pick P1)

Current line-width is 0.0000

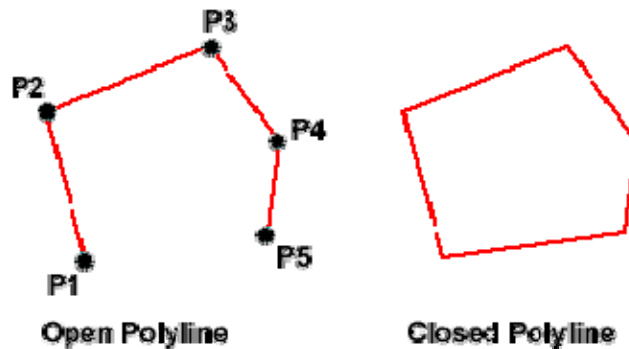
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: (pick P2)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (pick P3)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (pick P4)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (pick P5)

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (or C to close)



You can draw polylines of various widths by using the Width and Halfwidth options. You can set the width of individual segments and make them taper gradually from one width to another. These options become available after you specify a starting point for the polyline.

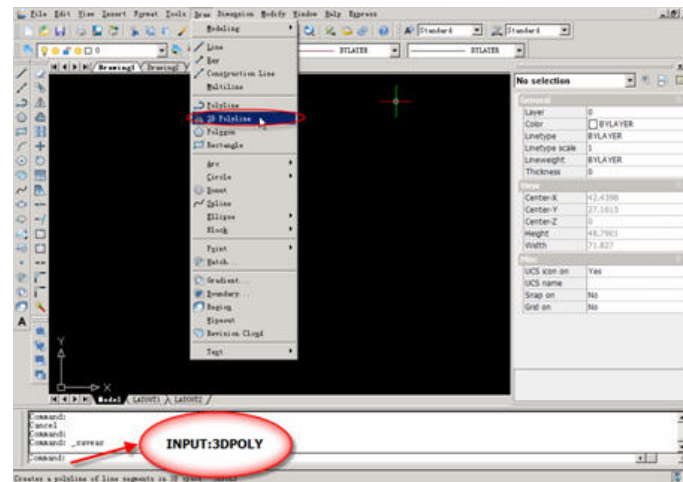
It is worth while taking some time to familiar with the Polyline command as it is an extremely useful command to know. Try experimenting with options such as Arc and Width and see if you can create polylines like the ones in the illustration above. The Undo option is particularly useful. This allows you to unpick polyline vertices, one at a time so that you can easily correct mistakes.

Polylines can be edited after they are created to, for example, change their width. You can do this using the PEDIT command, Modify>Object>Polyline from the pull-down menu.

### 3.5 3D Polyline

Pull-down menu: Draw>3D Polyline

Command: 3DPOLY



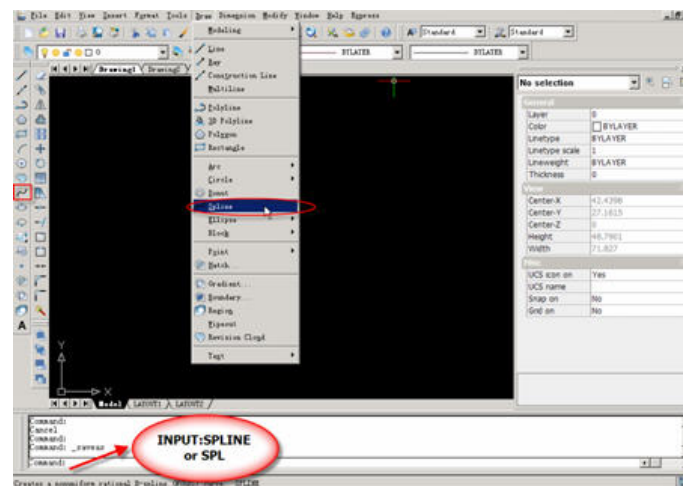
The 3D Polyline command works in exactly the same way as the Polyline command. The main difference between a normal polyline and a 3D polyline is that each vertex (pick point) of a 3D polyline can have a different value for Z (height). In normal (2D) polylines, all vertexes must have the same Z value.

### 3.6 Spline

Pull-down menu: Draw>Spline

Toolbar: 

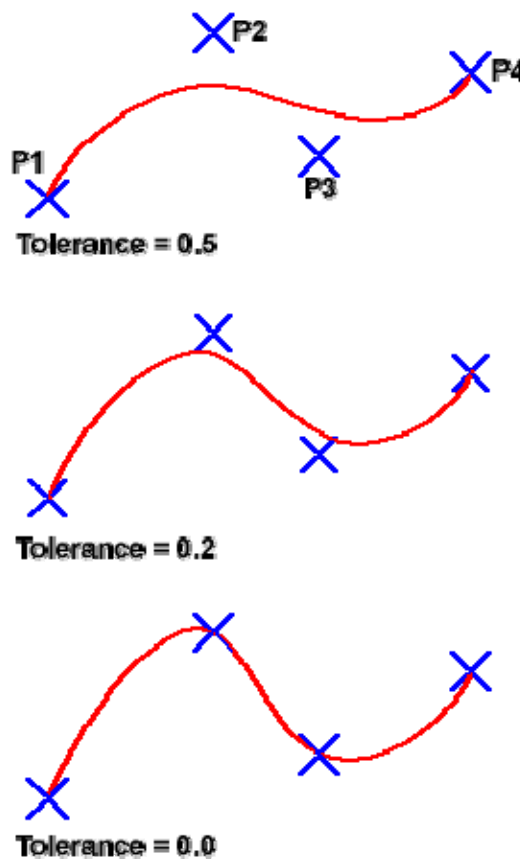
Command: SPLINE or SPL



The Spline command creates a type of spline known as a nonuniform rational B-spline, NURBS for short. A spline is a smooth curve that is fitted along a number of control points. The Fit Tolerance option can be used to control how closely the spline conforms to the control points. A low tolerance value causes the spline to form close to the control

points. A tolerance of 0 (zero) forces the spline to pass through the control points. The illustration on the right shows the effect of different tolerance values on a spline that is defined using the same four control points, P1, P2, P3 and P4.

Splines can be edited after they have been created using the SPLINEDIT command, Modify>Object>Spline from the pull-down menu. Using this command, you can change the tolerance, add more control points move control points and close splines, amongst other things. However, if you just want to move spline control points, it is best to use grips.



### Command Sequence

Command: SPLINE

Specify first point or [Object]: (Pick P1)

Specify next point: (Pick P2)

Specify next point or [Close/Fit tolerance] <start tangent>: (Pick P3)

Specify next point or [Close/Fit tolerance] <start tangent>: (Pick P4)

Specify next point or [Close/Fit tolerance] <start tangent>:


Specify start tangent: (pick a point)

Specify end tangent: (pick a point)

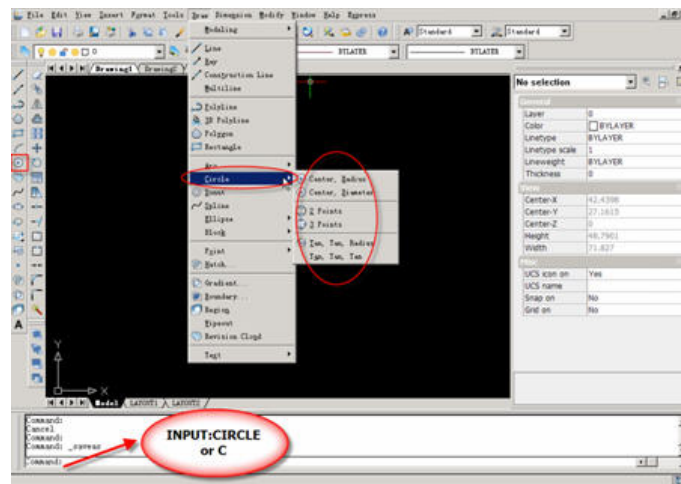
You can create linear approximations to splines by smoothing polylines with the PEDIT command, Modify>Object>Polyline from the pull-down menu

### 3.7 Circle

Pull-down menu: Draw >Circle

Toolbar: 

Command line: CIRCLE or C



The Circle command is used to draw circles. There are a number of ways you can define the circle. The default method is to pick the center point and then to either pick a second point on the circumference of the circle or enter the circle radius at the keyboard.

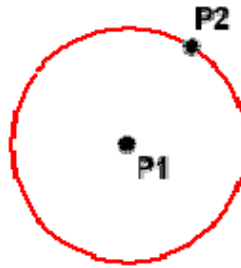
#### Command Sequence

Command: CIRCLE

[TwoPoints(2P)/ThreePoints(3P)/Tangency-Tangency-Radius(T)/Arc/Multiple]<Center of circle>:(pick P1)


[Diameter/Radius]<Radius> <17.4439>: (pick P2 or enter the exact radius)



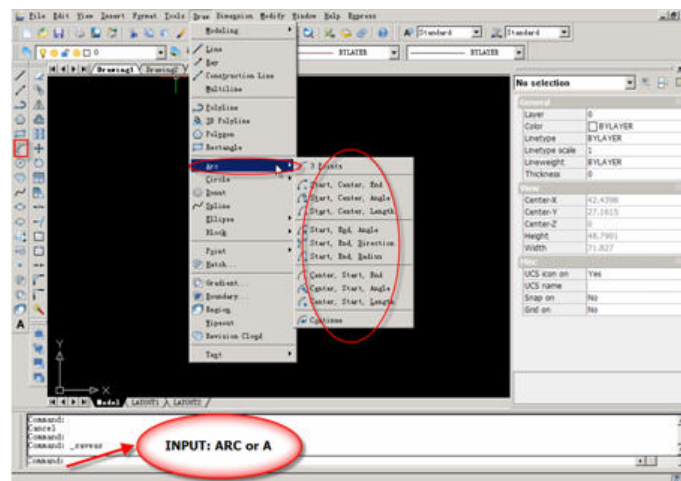


### 3.8 Arc

Pull-down menu location: Draw >Arc

Toolbar: 

Command line: ARC(A)



The Arc command allows you to draw an arc of a circle. There are numerous ways to define an arc, the default method uses three pick points, a start point, a second point and an end point. Using this method, the arc will start at the first pick point, pass through the second point and end at the third point. Once you have mastered the default method try some of the others. You may, for example need to draw an arc with a specific radius. All of the Arc command options are available from the pull-down menu.

#### Command Sequence

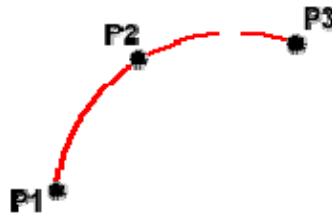
Command: ARC

Specify start point of arc or [Center]: (pick P1)

Specify second point of arc or [Center/End]: (pick P2)

Specify end point of arc: (pick P3)

It is also possible to create an arc by trimming a circle object. In practice, many arcs are actually created this way.



### 3.9 Donut

Pull-down menu: Draw >Donut

Command: DONUT

This command draws a solid donut shape, actually it's just a closed polyline consisting of two arc segments which have been given a width. GStarCAD asks you to define the inside diameter i.e. the diameter of the hole and then the outside diameter of the donut. The donut is then drawn in outline and you are asked to pick the center point in order to position the donut. You can continue picking center points to draw more donuts or you can hit to end the command. Surprisingly, donuts are constructed from single closed polylines composed of two arc segments which have been given a width. Fortunately GStarCAD works all this out for you, so all you see is a donut.

#### Command Sequence

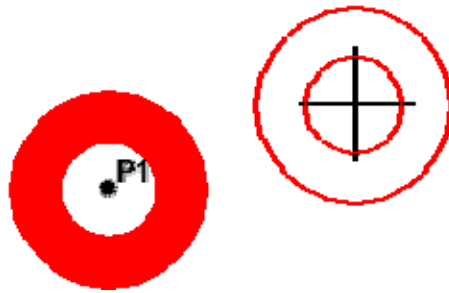
Command: DONUT

Specify inside diameter of donut <0.5000>: (pick any two points to define a diameter or enter the exact length)

Specify outside diameter of donut <1.0000>: (pick any two points to define a diameter or enter the exact length)

Specify center of donut or <exit>: (pick P1)

Specify center of donut or <exit>: (to end or continue to pick for more doughnuts)

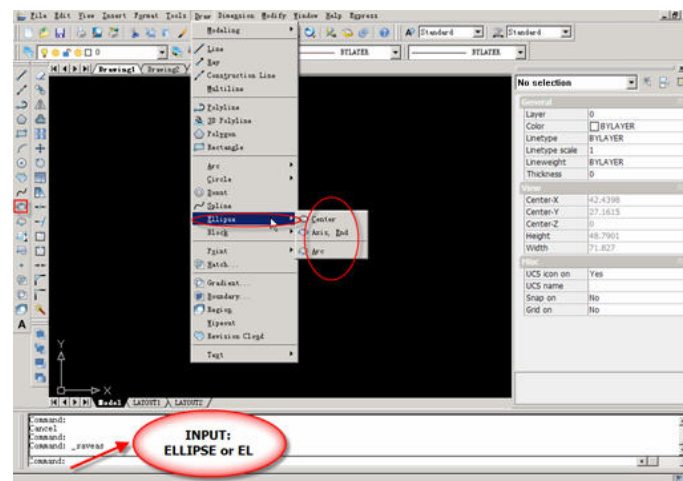


### 3.10 Ellipse

Pull-down menu: Draw >Ellipse

Toolbar: 

Command: ELLIPSE or ELL



The Ellipse command gives you a number of different creation options. The default option is to pick the two end points of an axis and then a third point to define the eccentricity of the ellipse. After you have mastered the default option, try out the others.

#### Command Sequence

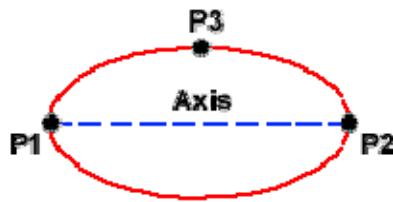
Command: ELLIPSE

Specify axis endpoint of ellipse or [Arc/Center]: (pick P1)

Specify other endpoint of axis: (pick P2)

Specify distance to other axis or [Rotation]: (pick P3)

The ellipse command can also be used to draw isometric circles



### 3.11 Ellipse Arc

Pull-down menu: Draw >Ellipse Arc

Toolbar: 

Command: **ELLIPSE** or **A**

The Ellipse Arc command is very similar to the Ellipse command, described above. The only difference is that, in addition to specifying the two axis end points and the "distance to other axis" point, you are prompted for a start and end angle for the arc. You may specify angles by picking points or by entering values at the command prompt. Remember that angles are measured in an anti-clockwise direction, starting at the 3 o'clock position.

In truth, the Ellipse Arc command is not a new or separate command; it is just an option of the Ellipse command and it therefore has no unique command line name. It is curious why Autodesk considered this option important enough to give it its own button on the Draw toolbar. Still, there it is.

#### Command Sequence

Command: ELLIPSE

Specify axis endpoint of ellipse or [Arc/Center]: A

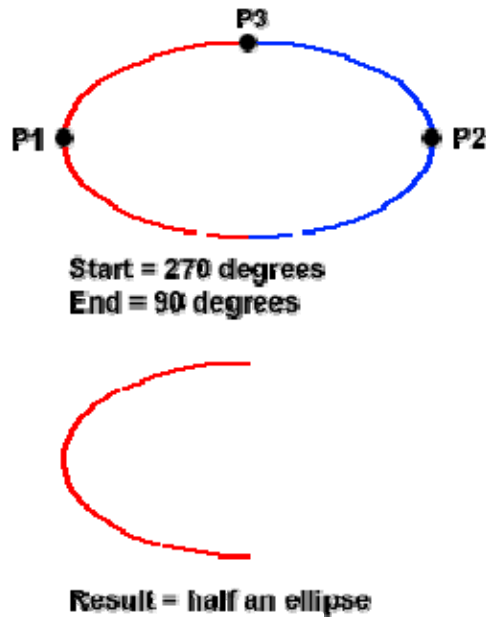
Specify axis endpoint of elliptical arc or [Center]: (pick P1)

Specify other endpoint of axis: (pick P2)

Specify distance to other axis or [Rotation]: (pick P3)

Specify start angle or [Parameter]: 270

Specify end angle or [Parameter/Included angle]: 90

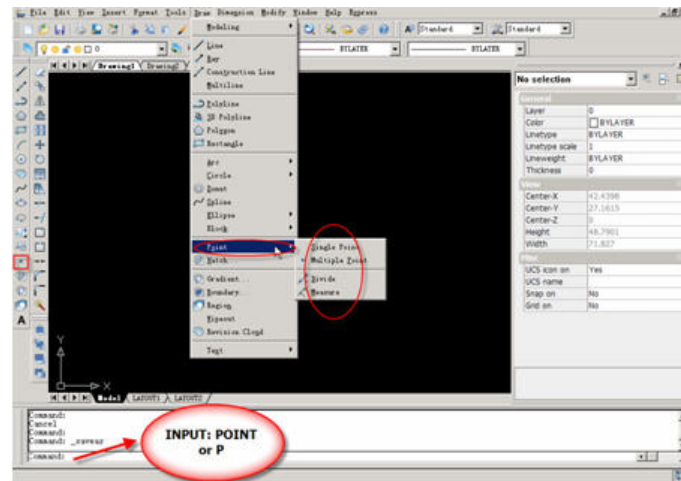


### 3.12 Point

Pull-down menu: Draw >Point

Toolbar: 

Command: POINT or P



Point objects are useful as nodes or reference geometry for object snaps and relative offsets.

You can set the style of the points and their size relative to the screen or in absolute units. Changing the style of points

- Make them more visible and easier to differentiate from grid dots
- Affect the display of all point objects in the drawing

- Require using REGEN to make the change visible

### Command Sequence

Command: POINT

Specify a point: (pick any point)

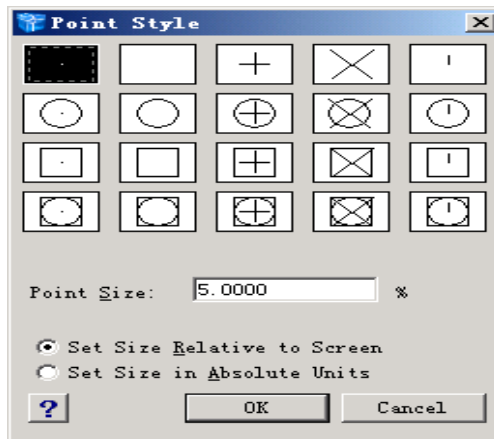
### Set Point Style

Pull-down: Format>Point Style...

Command: DDPTYPE

To change the point style, just pick the picture of the style you want and then click the "OK" button. You will need to use the Regen (or RE) command. Any new points created after the style has been set will automatically display in the new style.

One interesting aspect of points is that their size can be set to an absolute value or relative to the screen size, expressed as a percentage. The default is for points to display relative to the screen size, which is very useful because it means that points will remain the same size, irrespective of zoom factor. This is particularly convenient when drawings become complex and the drawing process requires a lot of zooming in and out



## 3.13 Rectangle

Pull-down menu: Draw >Rectangle

Toolbar: 

Command: RECTANGLE or REC

The Rectangle command is used to draw a rectangle whose sides are vertical and

horizontal. The position and size of the rectangle are defined by picking two diagonal corners. The rectangle isn't really an GStarCAD object at all. It is, in fact, just a closed polyline which is automatically drawn for you.

### Command Sequence

Command: RECTANG

[Chamfer/Elevation/Fillet/Rotated/Square/Thickness/Width] or <Select first corner of rectangle>:(pick P1)

Other corner of rectangle: (pick P2)



The Rectangle command also has a number of options. Width works in the same way as for the Polyline command. The Chamfer and Fillet options have the same effect as the Chamfer and Fillet commands,

### 3.14 Polygon

Pull-down menu: Draw >Polygon

Toolbar: 

Command line: POLYGON or POL

The Polygon command can be used to draw any regular polygon from 3 sides up to 1024 sides. This command requires four inputs from the user, the number of sides, a pick point for the center of the polygon, whether you want the polygon inscribed or circumscribed and then a pick point which determines both the radius of this imaginary circle and the orientation of the polygon. The polygon command creates a closed polyline in the shape of the required polygon.

This command also allows you to define the polygon by entering the length of a side

using the Edge option. You can also control the size of the polygon by entering an exact radius for the circle. Follow the command sequence below to see how this command works.

### Command Sequence

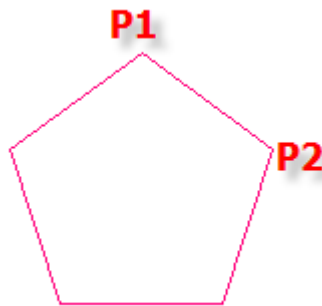
Command: POLYGON

[Multiple/Width of line]<Number of sides> <5>:

Specify by: [Edge]<Center of polygon>:e

First endpoint of edge: P1

Second endpoint of edge: P2



### 3.15 Multiline

Pull-down menu: Draw >Multiline

Command: MLINE

The Multiline command is used to draw multilines. This process of drawing is pretty much the same as drawing polylines, additional line segments are added to the multiline as points are picked. As with polylines, points can be unpicked with the Undo option and multilines can be closed.

When you start the Multiline command you also have the option to specify the Justification, Scale and Style of the multiline. The Justification option allows you to set the justification to "Top", the default, "Zero" or "Bottom". When justification is set to top, the top of the multiline is drawn through the pick points, as in the illustration below. Zero justification draws the center line of the multiline through the pick points and Bottom draws the bottom line through the pick points. Justification allows you to



control how the multiline is drawn relative to your setting out information. For example, if you are drawing a new road with reference to its center line, then Zero justification would be appropriate.

The Scale option allows you to set a scale factor, which effectively changes the width of the multiline. The default scale factor is set to 1.0 so to half the width of the multiline, a value of 0.5 would be entered. A value of 2.0 would double the width.

The Style option enables you to set the current multiline style. The default style is called "Standard". This is the only style available unless you have previously created a new style with the Multiline Style command. Follow the command sequence below to see how the Multiline command works and then try changing the Justification and Scale options.

### Command Sequence

Command: MLINE

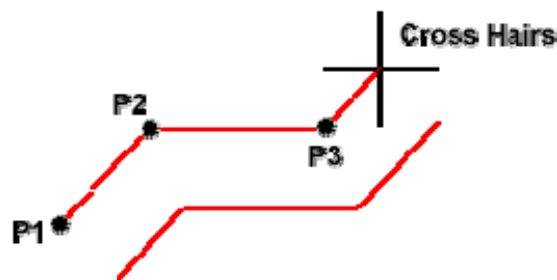
Current settings: Justification = Top, Scale = 20.00, Style = STANDARD

Specify start point or [Justification/Scale/STyle]: (Pick P1)

Specify next point: (Pick P2)

Specify next point or [Undo]: (Pick P3)

Specify next point or [Close/Undo]: (to end or continue picking or C to close)



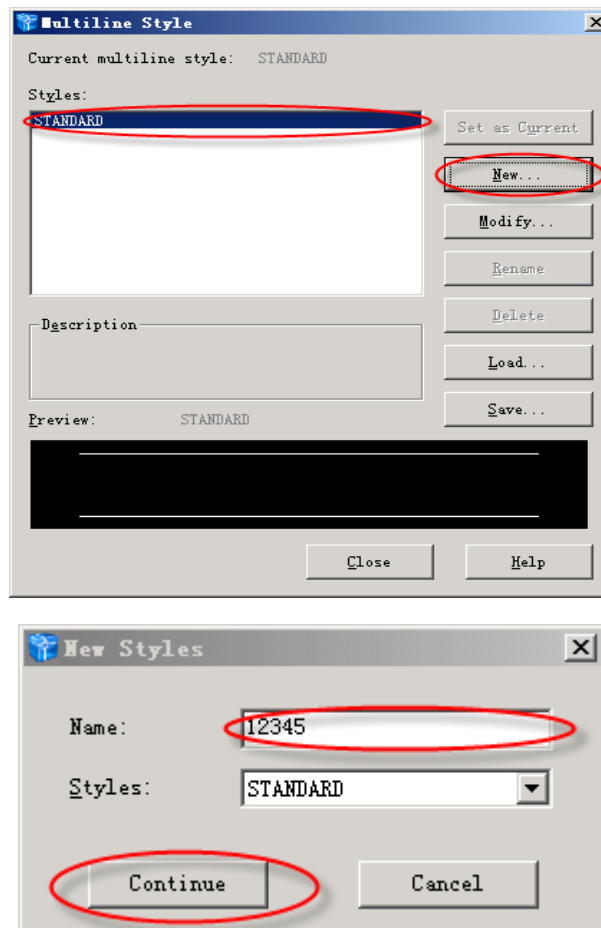
### Set Multiline Style

Pull-down menu: Format>Multiline Style

Command: MLSTYLE

The Multiline style command is used to create new multiline styles, which can then be

used with the Multiline command. When you start the command for the first time, you will see the Multiline Styles dialogue box indicating that the Standard style is "Current". To create a new style, click "New", enter a new style name in the "Name" edit box



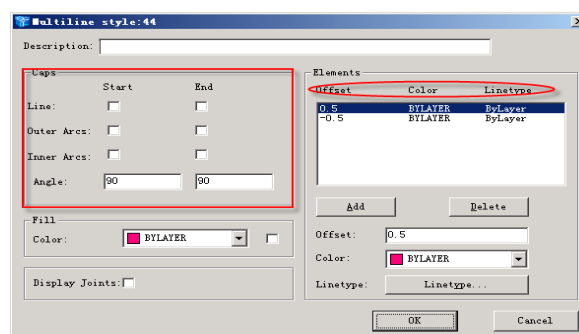
Then click "Continue". So the next step is to change the style to suit your own purposes. You will now see the Element Properties dialogue box appear. This dialogue box allows you to add new line elements or delete existing ones and to control the element offset, color and linetype. Click the "Add" button to add a new element. A new line element now appears with an offset of 0.0, in other words, this is a center line. Highlight the top element in the "Elements" list and change the offset to 1.0 by entering this value in the "Offset" edit box. Now do the same with the bottom element remembering to enter a value of -1.0 because this is a negative offset. You now have a multiline that is 2 drawing units wide with a centerline. Let's now change the color and linetype of the center line.

Highlight the 0.0 offset element by clicking it in the "Elements" list. To change the color, simply click on the Color... button and select an appropriate color from the palette. When a color has been selected, click the "OK" button on the palette to return to the Element Properties dialogue box.

Changing the linetype is a little more complicated because we will need to load the required linetype first. However, click on the "Linetype..." button to proceed.

The Select Linetype dialogue box appears with just a few solid linetypes listed, ByLayer, ByBlock and Continuous. Click on the "Load..." button. The Load or Reload Linetypes dialogue box now appears. Scroll down the list of linetypes until you find one called "Hidden". Highlight Hidden and then click the "OK" button. You will now see the Hidden linetype appear in the "Loaded linetypes" list in the Select Linetype dialogue box, which should now look similar to the one shown above. Finally, highlight Hidden and click the "OK" button. Your Element Properties dialogue box should now look similar to the one in the illustration above. To complete our new style, we will add some end caps and a solid fill. Click on the "Multiline Properties..." button to proceed.

In the Multiline Properties dialogue box, click in the "Line" check boxes under "Start" and "End". This will have the effect of capping the ends of the multiline with a 90 degree line. As you can see from the dialogue box, you can change this angle if you wish to give a chamfered end. Next, click the "On" check box in the "Fill" section and then click on the Color button and select the fill color from the palette. The Multiline Properties dialogue box should now look like the one in the illustration on the left. Finally, click the "OK" button in the Multiline Properties dialogue box and again in the Multiline Style dialogue box. You are now ready to draw with your new multiline.



### 3.16 Revision Cloud

Pull-down menu: Draw>Revision Cloud

Toolbar: 

Command line: REVCLOUD

The Revcloud command is used to draw a "freehand" revision cloud or to convert any closed shape into a revision cloud.

#### Command Sequence

Command: REVCLOUD

Minimum arc length: 15    Maximum arc length: 15    Style: Normal

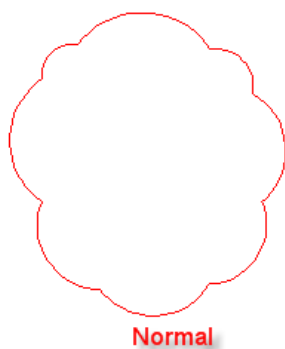
Specify start point or [Arc length(A)/Object/Style] <Object>:(Pick P1)

Guide crosshairs along cloud path...

Move the mouse to form a closed shape; the command automatically ends when a closed shape is formed.


Revision cloud finished.

You can use the "Arc length" option to control the scale of the revision cloud. This is achieved by specifying the minimum and maximum arc length. The "Object" option is used to transform any closed shape, such as a polyline, spline or circle into a revision cloud. You can use "Style" option set the style as Normal or Calligraphy .



### 3.17 Region

Pull-down menu: Draw >Region

Toolbar: 

Command: REGION

A region is a surface created from objects that form a closed shape, known as a loop. The Region command is used to transform objects into regions rather than actually drawing them (i.e. you will need to draw the closed shape or loop first). Once a region is created, there may be little visual difference to the drawing.

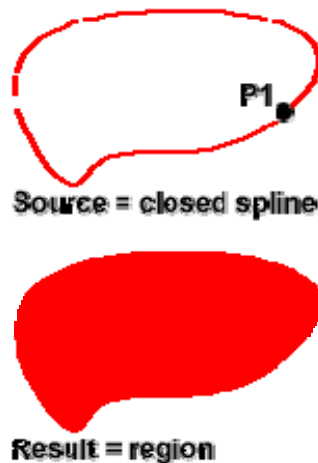
Before starting the Region command, draw a closed shape such as a rectangle, circle or any closed polyline or spline.

### Command Sequence

Command: REGION

Select objects: (Pick P1)

1 region have been created



### 3.18 Hatch

You can hatch an area using a predefined hatch pattern, define a simple line pattern using the current linetype, or create more complex hatch patterns. One type of pattern is called solid, which fills an area with a solid color.

You can also create a gradient fill, which uses a transition between shades of one color or between two colors. Gradient fills can be used to enhance presentation drawings, giving the appearance of light reflecting on an object.

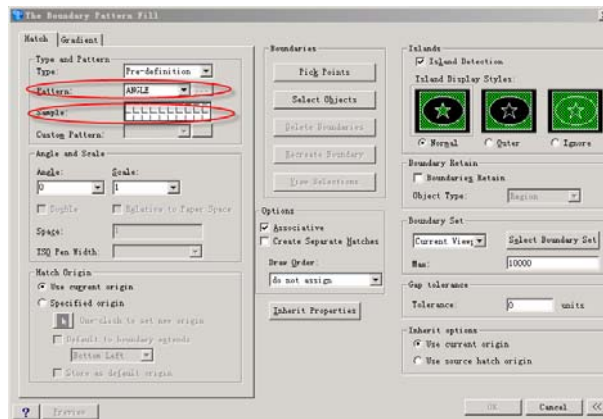
Pull-down menu: Draw> Hatch

Toolbar: 

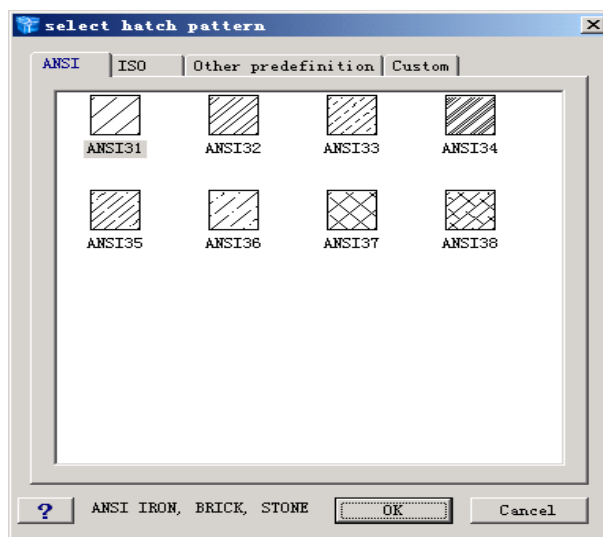
Command: BHATCH

### The usage of hatch

Command: \_BHATCH

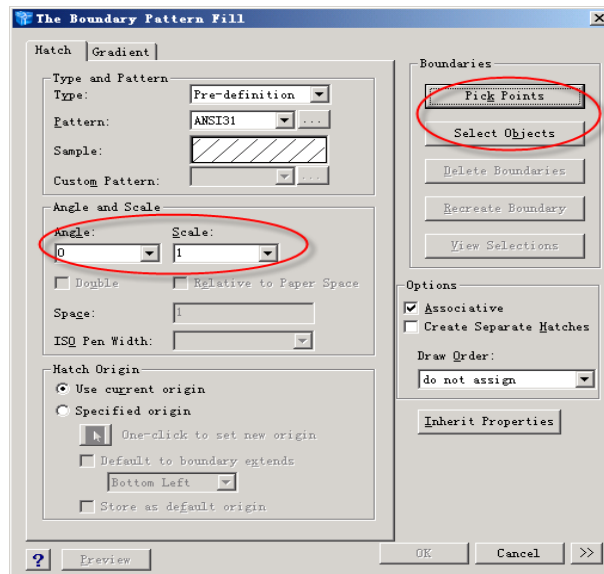


- Firstly, in order to determine the type of hatching lines, select proper hatching patterns in the “Pattern” tab.



If you select “SOLID” in the “Pattern” tab, then click “OK” button.

- In order to change scale and angle, input the value of “Scale” and “Angle” in the “Hatch” tab.



- In order to do hatching, if you select "Pick Points " button in the "Hatch" tab, pick an internal point of hatching area; if you select "Select Objects " button in the "Hatch" tab, pick the boundary of hatching area.

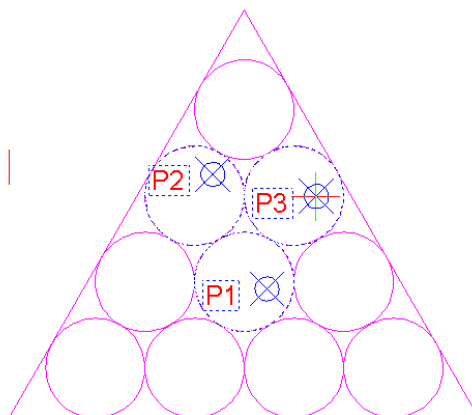
I "Pick Points:

Select a point to define a boundary or hatch area: (pick P1)

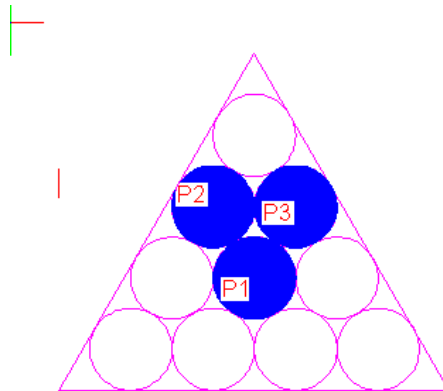
Select a point to define a boundary or hatch area: (pick P2)

Select a point to define a boundary or hatch area: (pick P3)

Select a point to define a boundary or hatch area:



Then press "OK" in the "Hatch" tab.



### II "Select Objects "

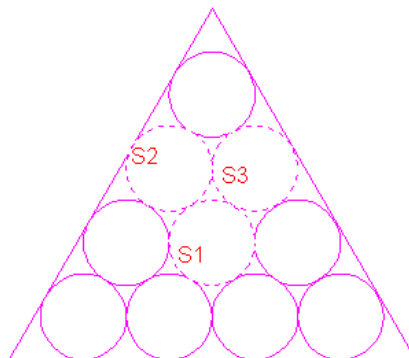
Select entities:(select S1)

Select entities:(select S2)

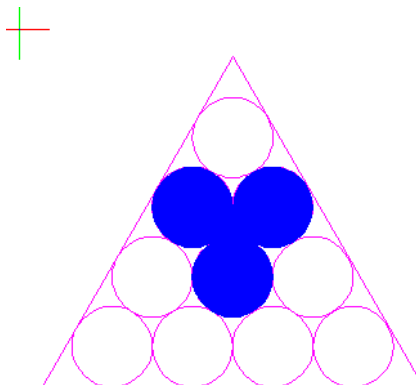
Select entities:(select S3)

Select entities:

□



Then press “OK” in the “Hatch” tab.





### 3.19 Wipeout

Pull-down menu: Draw>Wipeout

Command: WIPEOUT

A Wipeout is an image type object. Most commonly it is used to "mask" part of a drawing for clarity. For example, you may want to add text to a complicated part of a drawing. A Wipeout could be used to mask an area behind some text so that the text can easily be read, as in the example shown on the right.



The Wipeout command can be used for 3 different operations. It can be used to draw a wipeout object, as you might expect, but it can also be used to convert an existing closed polyline into a wipeout and it can be used to control the visibility of wipeout frames.

#### Command Sequence

Command: WIPEOUT

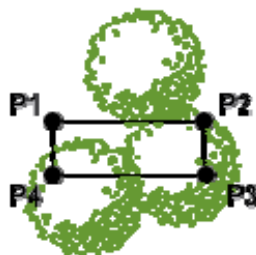
Specify first point or [Frames/Polyline] <Polyline>: (Pick P1)

Specify next point: (Pick P2)

Specify next point or [Undo]: (Pick P3)

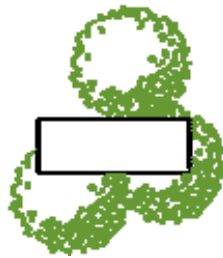
Specify next point or [Close/Undo]: (Pick P4)

Specify next point or [Close/Undo]:



You can use as many points as you wish in order to create the shape you need. When you have picked the last point, use right-click and Enter (or hit the Enter key on the keyboard) to complete the command and create the wipeout.

You may find that it is easier to draw a polyline first and then convert that polyline into a wipeout. To do this, start the Wipeout command and then Enter to select the default "Polyline" option. Select the polyline when prompted to do so. Remember, polylines must be closed before they can be converted to wipeouts.



### 3.20 Block

Block is a combination of several entities and is treated as one entity by inserting it into a drawing. Block consists of visual entities such as lines, arcs and circles and attributes of them. It is saved as part of the drawing file. When you use Block, you can make the drawing quickly and reduce the file size. If you make common-used symbols into libraries, you can make the symbols by inserting BLOCK without drawing them again from first.

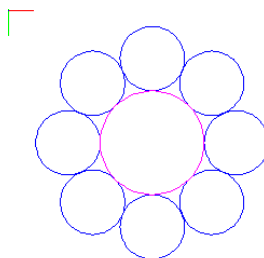
#### 1. Create block

Pull-down menu: Draw >Block >Make

Command: BLOCK

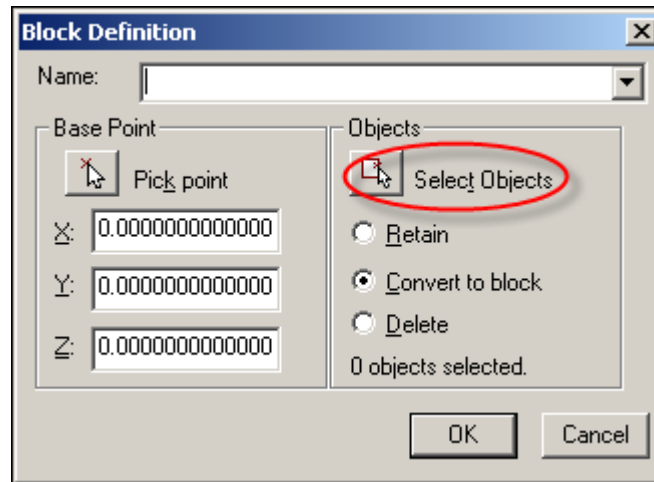
Toolbar: 

After you drawing the following drawing on the screen, set them as BLOCK.



- Set the drawing as BLOCK.

Command: \_BLOCK

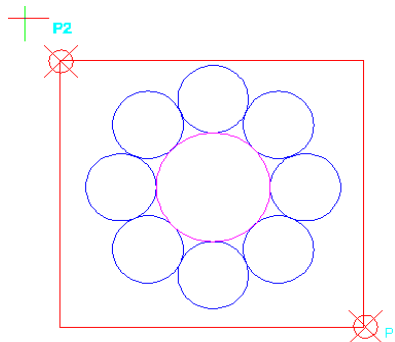


Press “Select” button.

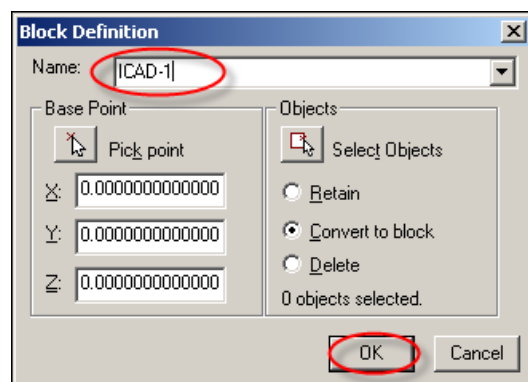
Select entities for block: (pick P1)

Opposite corner: (pick P2)

Select entities for block:



- Then input the name of the block, press “OK” button.



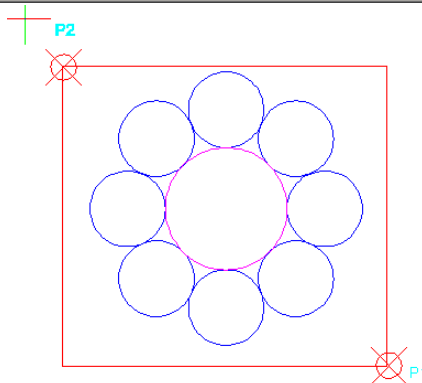
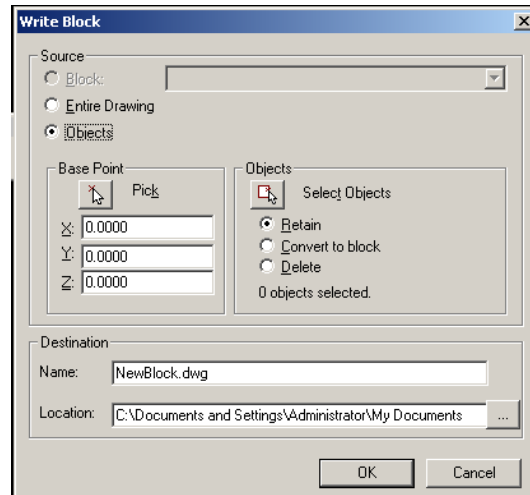
## 2. WBLOCK

Insert a BLOCK that is created by BLOCK command into the current drawing.

Command: WBLOCK

### The usage of Wblock

Command: WBLOCK



Select entities for block: (pick P1)

Opposite corner: (pick P2)

Select entities for wblock:

### 3. Insert block

Inserting block is used to insert the BLOCK that is created by WBLOCK command or other external drawings into the current drawing.

Pull-down menu: Insert>Block

Command: INSERT or DDINSERT

Toolbar: 

- Select the BLOCK or other external drawings to insert.
- Input the insertion point of BLOCK, scale factor and rotation angle. When you select "Position block when inserting", you can't input the parameters. In that case, you can input the parameters when you insert BLOCK into the screen.

When you select "Explode block upon insertion", BLOCK is inserted as exploded entities instead of one entity.

## Chapter 4 Modifying Entities

### Introduction

Drawings are rarely completed simply by drawing lines, circles etc. Most likely you will need to modify these basic drawing objects in some way in order to create the image you need. GStarCAD provides a whole range of modify tools such as Move, Copy, Rotate and Mirror. As you can see, the command names are easily understandable. If you just need information quickly, use the toolbar below to go straight to the information you need.

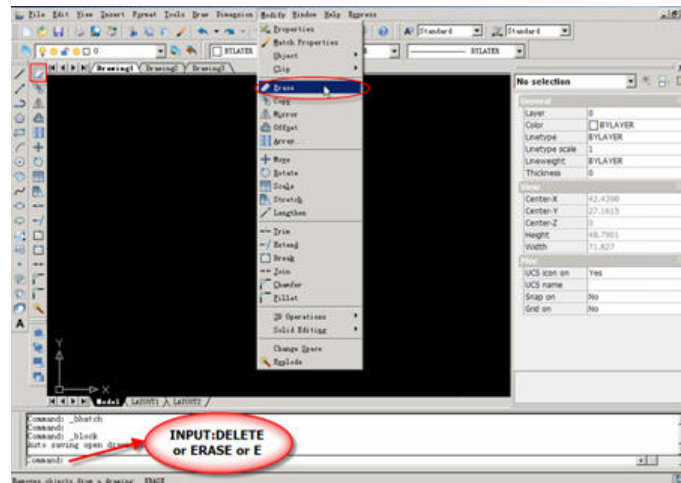


### 4.1 The Erase Command

Pull-down menu: Modify>Erase

Toolbar: 

Command: ERASE or E



The Erase command is one of the simplest commands and is one of the most used. The command erases (deletes) any selected object(s) from the drawing. Remember you can always get deleted objects back by typing U or undo, from the Standard toolbar or by using the OOPS command.

### Command Sequence

Command: ERASE


Select objects: (select objects to delete)

Select objects: (to end the selection and erase the object)

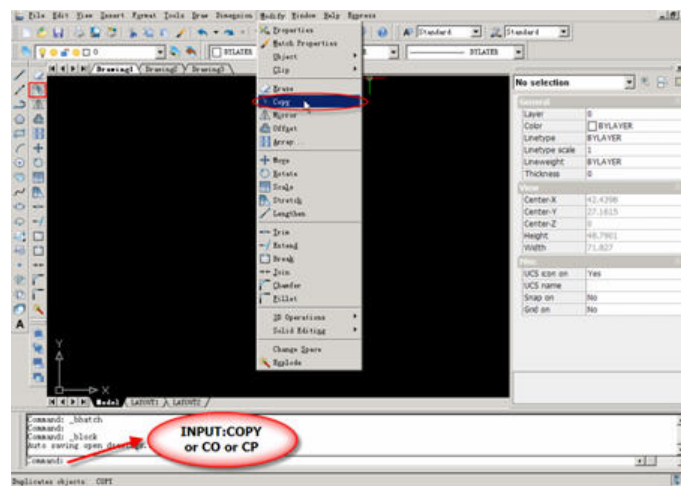
If you simply want to erase the last object you created you can type L at the "Select objects" prompt. The last object will be highlighted and you can then select more objects or to end the command.

### 4.2 The Copy Command

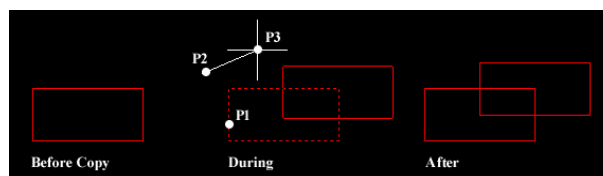
Pull-down menu: Modify>Copy

Toolbar: 

Command line: COPY or CP



The Copy command can be used to create one or more duplicates of any drawing object or objects which you have previously created. Copy is a very useful and time-saving command because you can create very complex drawing elements and then simply copy them as many times as you like.



#### Command Sequence

Command: COPY

Select objects: (pick object to copy, P1)

Select objects: (to end selection)

Vector/<Base point>: (pick P2)

Displacement point: (pick P3)

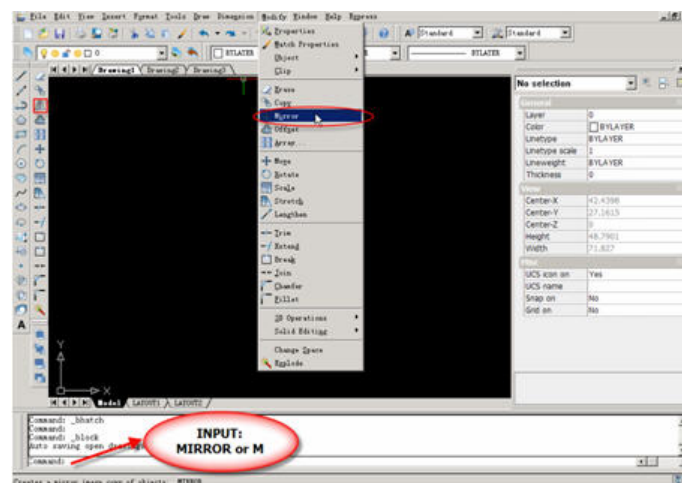
Notice that the "Base point", P2 and the "Second point", P3 do not have to be picked on or near the object. The two points are simply used to indicate the distance and direction of the copied object from the original object.

### 4.3 The Mirror Command

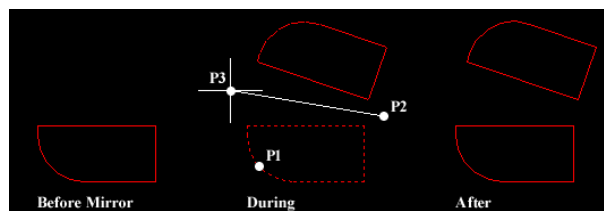
Pull-down menu: Modify>Mirror

Toolbar: 

Command line: MIRROR or MI



The Mirror command allows you to mirror selected objects in your drawing by picking them and then defining the position of an imaginary mirror line using two points.



#### Command Sequence

Command: MIRROR

Select objects: (pick object to mirror, P1)

Select objects: (to end selection)



Start of mirror line: (pick P2)

End of mirror line: (pick P3)

Delete the original entities? <N>

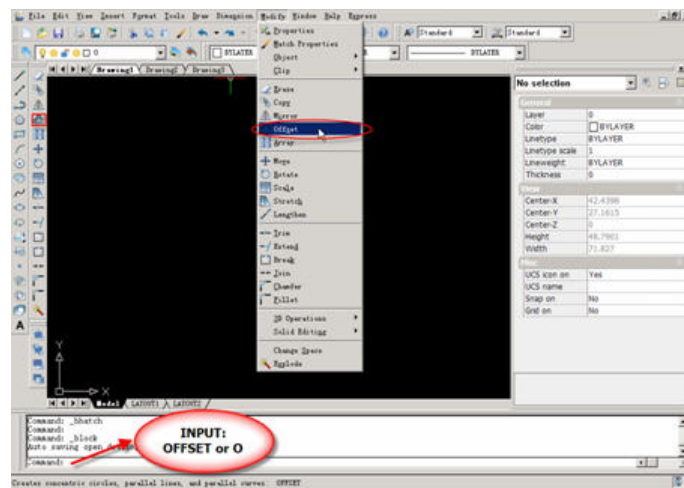
In order to create perfectly horizontal or vertical mirror lines use Ortho.

### 4.4 The Offset Command

Pull-down menu: Modify>Offset

Toolbar: 

Command: OFFSET or O



Offset is probably one of the most useful commands for constructing drawings. The Offset command creates a new object parallel to or concentric with a selected object. The new object is drawn at a user defined distance (the offset) from the original and in a direction chosen by the user with a pick point. You can offset lines, arcs, circles, ellipses, 2D polylines, xlines, rays and planar splines.

#### Command Sequence

Command: OFFSET

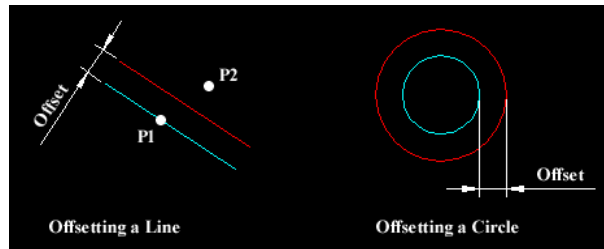
Parallel: [Through point]<Distance> <13.2025>:(input distance or use the default value)

Select objects: (select object, P1)

[Both sides]<Side for parallel copy>: (pick direction, P2)

In the illustration below, a line has been offset to the right through a distance "Offset" by picking a point to the right of the original line. The result is a new line (red) to the

right of the original.




Circles can be offset inside or outside of themselves to create a new circle which is concentric (has the same center point) with the original circle. In the illustration, a new circle (red) has been created outside of the original by picking a point outside of the original circle. The radius of the new circle is the offset distance "Offset" plus the radius of the original circle.

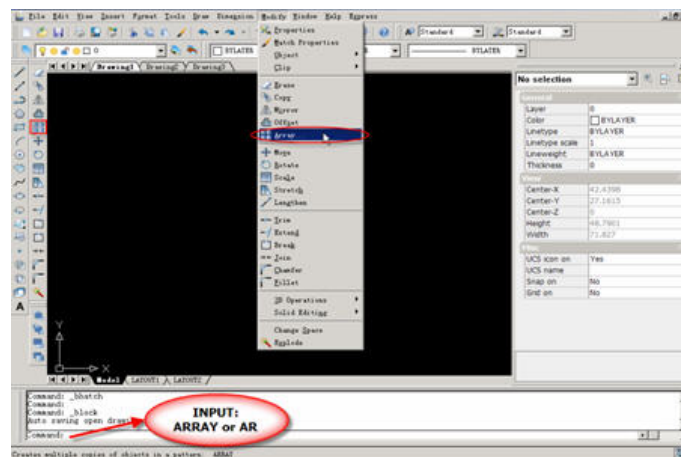
In the illustration above, the new objects and original objects are shown in different colors for clarity. In reality, offset objects inherit their object properties from the original object.

### 4.5 The Array Command

Pull-down menu: Modify>Array

Toolbar: 

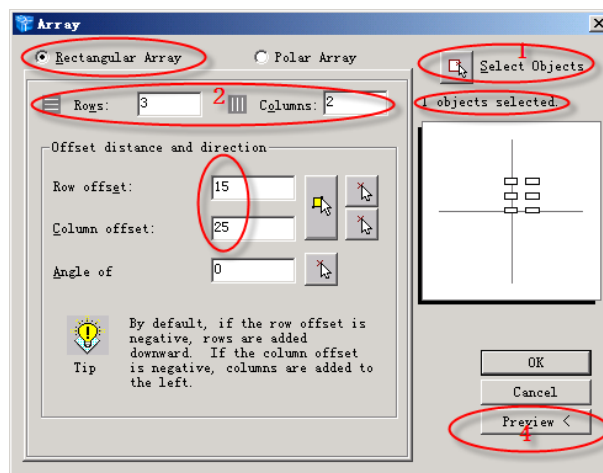
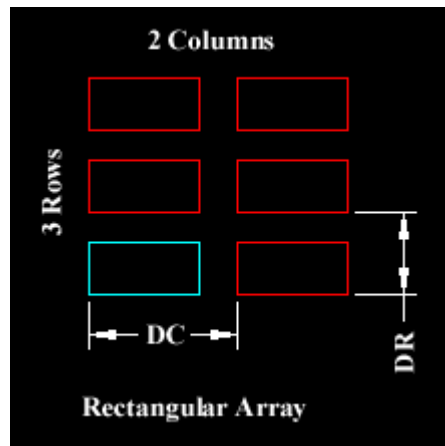
Command: ARRAY or AR



The Array command makes multiple copies of selected objects in a rectangular matrix (columns and rows) or a polar (circular) pattern.

#### The Rectangular Array

The illustration shows the results of a rectangular array with two columns and three rows. The distance between rows is indicated with the dimension DR and between columns with DC. When creating rectangular arrays it is important to remember that new rows are created above the original object and new columns are created to the right of the original object. The resulting array is, therefore, always created with the original object in the bottom left hand position with respect to the current co-ordinate system. In the illustration, the original object is shown in cyan and the new objects in red. It is possible to create arrays which do not align with the World Co-ordinate System by setting a User Co-ordinate System first.



Now start the Array command. First of all, make sure that the Rectangular Array button is selected and that you are looking at the Rectangular Array dialogue box and now follow the steps below:

1. Click the Select objects button. The dialogue box will temporarily disappear enabling

you to select the rectangle you just drew. Press the Return button on your keyboard to complete the selection. You are now returned to the dialogue box and the message immediately below the Select Objects button should read "1 objects selected".

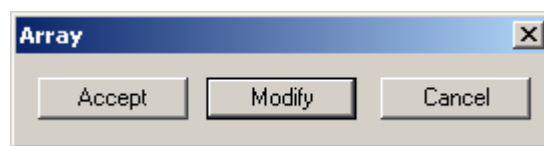
2. Enter the number of rows required in the Rows edit box. For this example, enter the value "3". Notice that the schematic preview on the right hand side of the dialogue box updates to reflect the values you are entering.

3. Enter the number of columns required in the Columns edit box. Enter the value "2".

4. Enter the row offset in the Row Offset edit box. This is the distance DR in the illustration above. Note that this is not the distance between rows. In this example, our rectangle is 10 units high and we will enter a row offset of 15. The result will be a 5 unit gap between rectangles.

5. Enter the column offset in the Column Offset edit box. The same parameters apply as for the row offset. Enter a value of 25 to give a 5 unit gap between our rectangles.

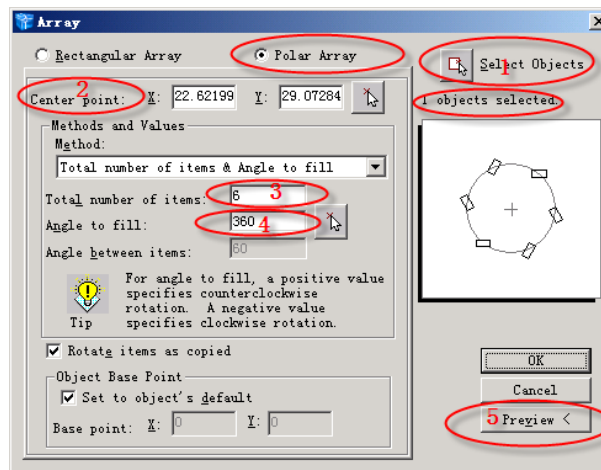
6. Click on the Preview button. Once again, the dialogue box disappears and the specified array is temporarily drawn so that we can preview it. We are now offered 3 choices. If the array isn't quite right, click the Modify button to return to the Array dialogue box. If you are happy with the array, click the Accept button, the array will be permanently drawn and the command is ended.



You should now have an array that looks similar to the one in the illustration above consisting of 6 rectangles arranged in 3 rows and 2 columns. As you can see, this command is very powerful and can save lots of time if used carefully.

### **The Polar Array**

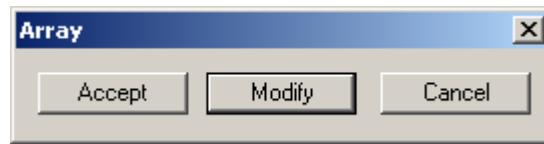
The polar array works in a similar way to the rectangular array. The main difference is that rather than specifying the number and offset for rows and columns, you must specify a center point and the total number of objects in the array (including the original object).



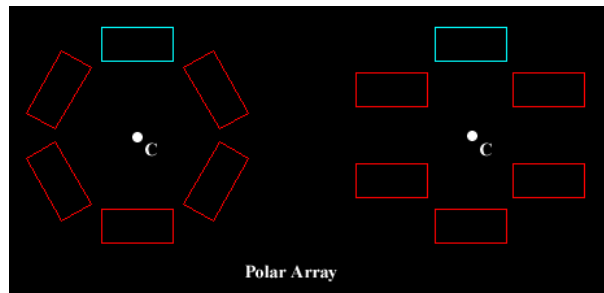
You can try this for yourself by drawing another rectangle as described above. This time, though, locate the rectangle in the center top half of the drawing area. Now, start the Array command, click the Polar Array radio button and follow the steps below:

1. Click the Select objects button. The dialogue box will temporarily disappear enabling you to select the rectangle you just drew. Press the Return button on your keyboard to complete the selection. You are now returned to the dialogue box and the message immediately below the Select Objects button should read "1 object selected".
2. Specify the center point for the array. This is the point C in the illustration below. You can do this by entering x and y co-ordinates into the appropriate edit boxes if you know what these values should be. However, this is rarely the case and most often you will want to click the Pick Center Point button to pick a point from the drawing area. Pick a point somewhere below the rectangle you have just drawn.
3. Enter a value for the total number of items. For this example, enter the value "6". Notice that once again, the schematic preview updates to reflect the values you have entered.
4. Make sure that the rotate items as copied checkbox is checked.
5. Click on the Preview button. Once again, the dialogue box disappears and the specified array is temporarily drawn so that we can preview it. We are now offered 3 choices. If the array isn't quite right, click the Modify button to return to the Array dialogue box. If you are happy with the array, click the Accept button, the array will be

permanently drawn and the command is ended.



The array you have just drawn should look something like the one illustrated below, left. Take some time to play around with the other options in the Array dialogue box to see what they can do.



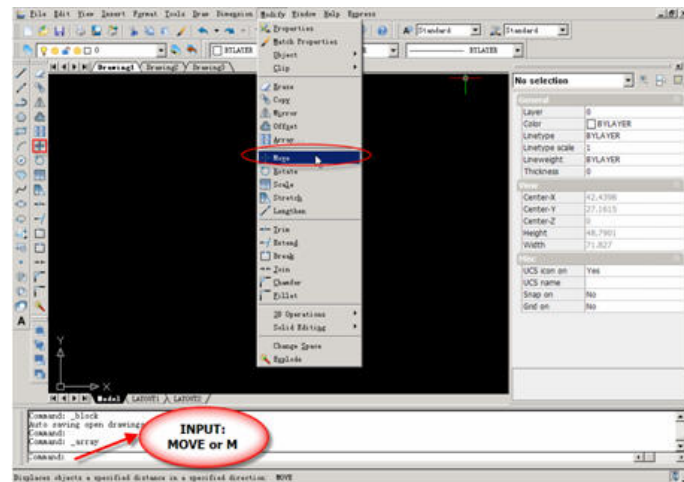
The illustration above shows the results of a polar array using an original rectangle copied six times about a center point C, through an angle of 360 degrees (full circle). The only difference between the array on the left and the one on the right is that the Rotate items as copied checkbox was not checked in the right hand example. All new objects created with the Array command will inherit the object properties of the original object.

### 4.6 The Move Command

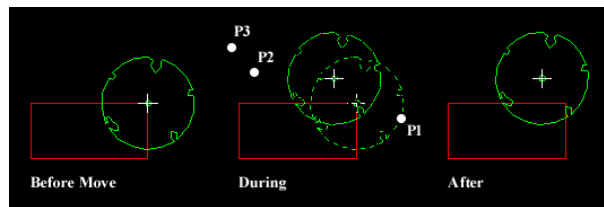
Pull-down menu: Modify>Move

Toolbar: 

Command line: MOVE or M



The Move command works in a similar way to the Copy command except that no copy is made, the selected object(s) is simply moved from one location to another.



### Command Sequence

Command: MOVE

Select objects to move: (pick object to move, P1)

Vector/<Base point>: (pick P2)

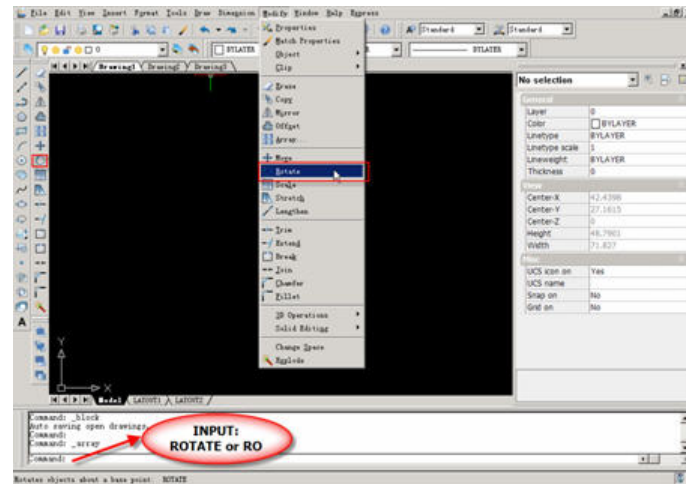
Displacement point: (pick P3)

### 4.7 The Rotate Command

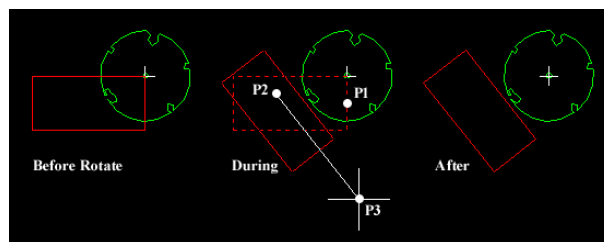
Pull-down menu: Modify>Rotate

Toolbar: 

Command: ROTATE or RO



The Rotate command allows an object or objects to be rotated about a point selected by the user. GStarCAD prompts for a second rotation point or an angle which can be typed at the keyboard.



### Command Sequence

Command: ROTATE

Select objects to rotate: (pick object to rotate, P1)

Rotation point: (pick base point, P2)

[Base angle]<Rotation angle>: (pick second point P3 or enter angle)

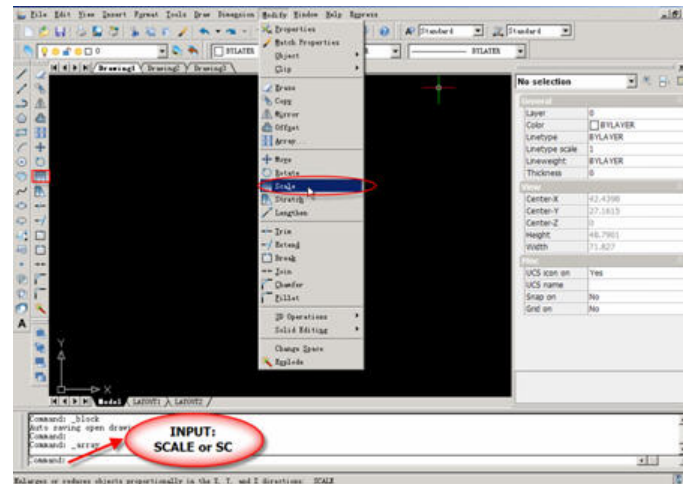
## 4.8 The Scale Command

Pull-down menu: Modify>Scale

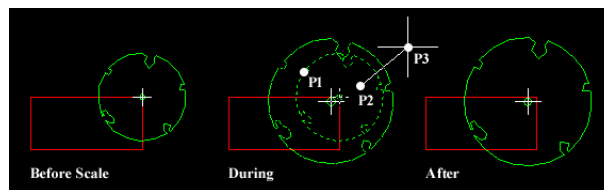
Toolbar: 

Command: SCALE or SC





The Scale command can be used to change the size of an object or group of objects. You are prompted for a pick point about which the selection set will be scaled. Scaling can then be completed by picking a second point (not always easy because it can sometimes be difficult to precisely control the scaling) or by entering a scale factor at the keyboard. For example: a scale factor of 2 will double the size of the objects in the selection set and a factor of 0.5 will half them.



### Command Sequence

Command: SCALE

Select objects: (pick objects to be scaled, P1)


Specify base point: (pick base point, P2)

[Base scale/Scale factor]<Scale factor>: (pick second point, P3 or enter scale factor)

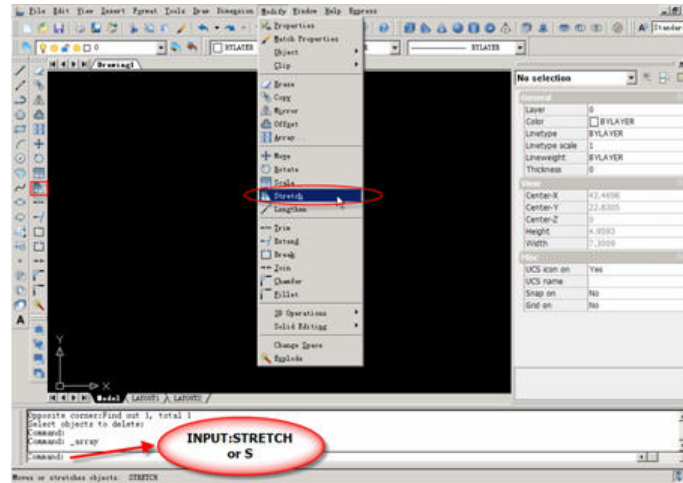
In the example shown above, the original tree symbol has been enlarged by dynamically scaling it using pick points to determine the change in scale. If you want to scale an object precisely, it is much easier to enter a scale factor using the keyboard.

## 4.9 The Stretch Command

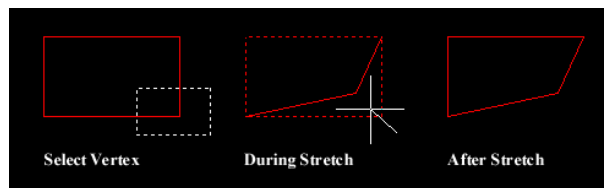
Pull-down menu: Modify>Stretch

Toolbar: 

Command: STRETCH or S



The Stretch command can be used to move one or more vertices of an object whilst leaving the rest of the object unchanged. In the example below, a rectangle has been stretched by moving one vertex to create an irregular shape.



### Command Sequence

Command: STRETCH

Select objects to stretch by crossing-window or crossing-polygon:

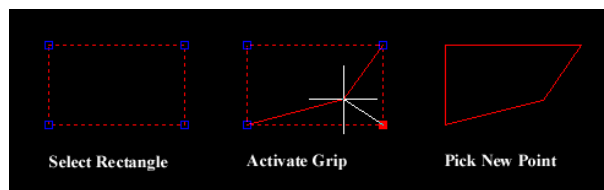
Specify base point or displacement: (pick base point)

Specify second point of displacement: (pick second point)

To select vertices to stretch, you must use a crossing window or polygon.

### Stretching with Grips

Although the Stretch command can be very useful, it has largely been superseded by the use of "Grips" which allow this sort of modification to be made much more intuitively.

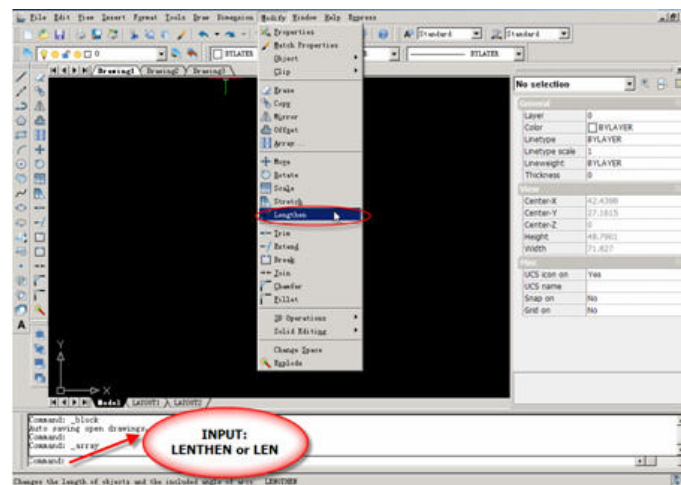


To stretch an object using grips, simply select the object by clicking on it (you can do this without starting a command). The object becomes highlighted and small square grips appear at each vertex and various snap points, depending upon the object type. Click a grip to activate it and click again to reposition it. When you have completed your modifications, use the Escape key (Esc) at the top left of your keyboard to deselect the object and release grips.

### 4.10 The Lengthen Command

Pull-down menu: Modify>Lengthen

Command line: LENGTHEN or LEN



The Lengthen command can often be used instead of either the Trim or Extend commands. Indeed the end result is exactly the same. The Lengthen command can be used to either lengthen or shorten Lines, Arcs, open Polylines, elliptical Arcs and open Splines without the use of cutting or boundary edges.

#### Command Sequence

Command: LENGTHEN

Edit length: [DYnamic/Increment/Percent/Total]<Select objects to list length>: DY

Select an object to change or [Undo]: (select a line or arc)

Specify new end point: (pick new end point)


Select an object to change or [Undo]: (to end)

The command sequence above demonstrates the use of the Dynamic Lengthen option which is probably the most useful for general purpose drafting. However, the other options are worth getting to know because they can save lots of time and effort.

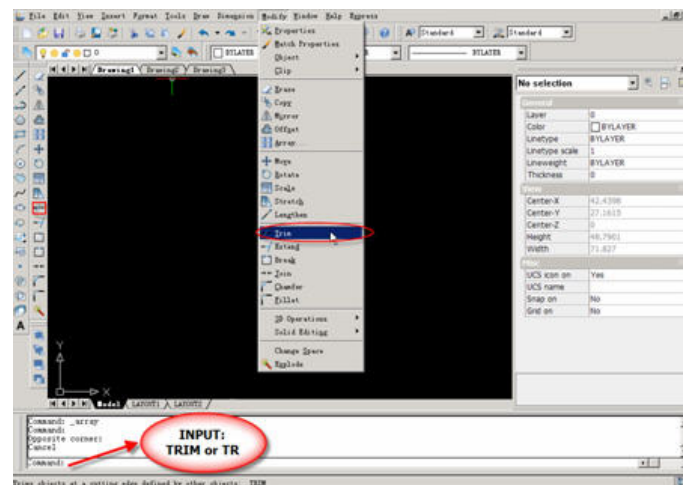
The Total option allows you to change the total length of a line to any value that you specify. The Percent option allows you to change a line length using a percentage.

### 4.11 The Trim Command

Pull-down menu: Modify>Trim

Toolbar: 

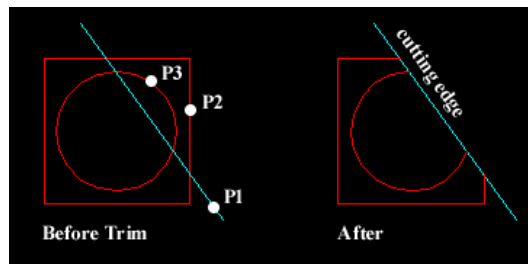
Command: TRIM or TR



The Trim command can be used to trim a part of an object. In order to trim an object you must draw a second object which forms the "cutting edge". Cutting edges can be lines, xlines, rays, polylines, circles, arcs or ellipses. Blocks and text cannot be trimmed or used as cutting edges. The illustration on the below shows the Trim command in action. The red square and circle have been drawn using the Polygon and Circle commands respectively. In order to trim these objects, a line has been drawn (cyan in the illustration), this forms the cutting edge. The Trim command, unlike most other modify commands requires that two separate object selections are made. The cutting edges are selected first (there can be one or more) and then the objects to be trimmed

are selected. In the example below, the line is selected first because it forms the cutting edge and then the square and circle are selected.

The Trim command is slightly more complicated than many other modify commands. To get a better understanding of how it works, draw a square, circle and line as illustrated below and then follow the command sequence below. Don't forget to watch the command line at each stage of the process.



### Command Sequence

Command: TRIM

Select objects or <select all>: (select the cutting edge, P1)

Select the object to Trim or shift-select to Extend or [Edge/Fence/Project]: (pick the part of the square which you want to trim, P2)

Select the object to Trim or shift-select to Extend or [Edge/Fence/Project/Undo]: (pick the circle, P3)

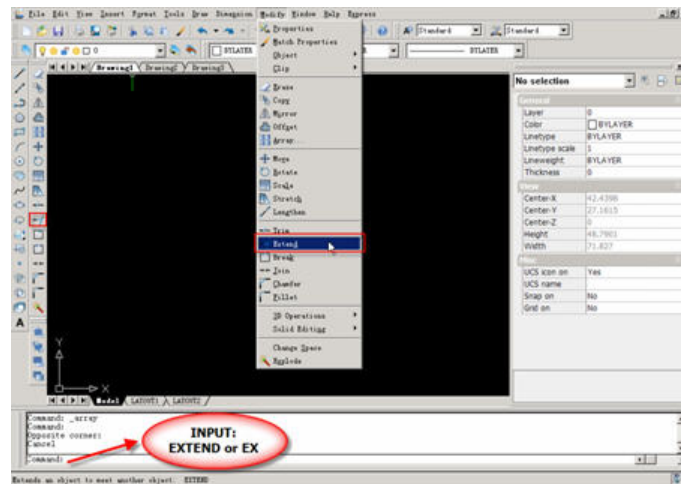
Notice that at each trimming step you are given the option to undo the previous trim. This can be very useful if you inadvertently pick the wrong object.

## 4.12 The Extend Command

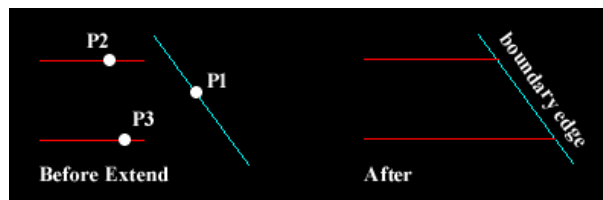
Pull-down menu: Modify>Extend

Toolbar: 

Command: EXTEND or EX



This command extends a line, polyline or arc to meet another drawing object (known as the boundary edge). In the illustration on the below, two lines (red) are extended to meet another line (cyan) which forms the boundary edge. This command works in a similar way to the Trim command, described above. Two selections are made, one for the boundary edge(s) and one for the object(s) to extend.



Lines and other objects can be extended in one of two directions. In the illustration on the right, the red line could be extended either to the right or to the left. You can tell GStarCAD in which direction to extend by picking a point to the right or left of the midpoint respectively. GStarCAD does not intuitively know where the boundary edge lies so you must explicitly indicate the direction of extension by picking either one side or other of the midpoint.

### Command Sequence

Command: EXTEND

Select objects or <select all>: (select the boundary edge, P1)

Select the object to Extend or shift-select to Trim or [Edge/Fence/Project]:

Select the object to Extend or shift-select to Trim or [Edge/Fence/Project/Undo]: (pick the object which you want to be extended, P2)

Select the object to Extend or shift-select to Trim or [Edge/Fence/Project/Undo]: (pick another object which you want to be extended, P3)

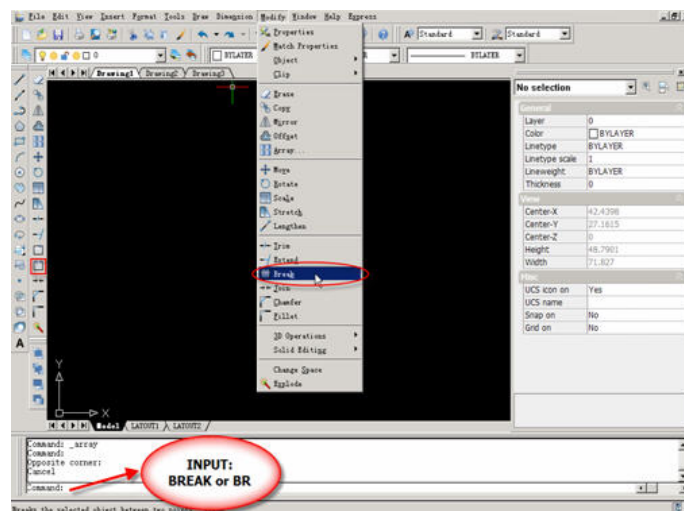
Sometimes you may get the message "Object does not intersect an edge" or "No edge in that direction". If this happens it means that you are either picking the wrong end of the object or the object you are trying to extend will not meet the boundary edge. The solution is either to pick near the end you want to extend or to move the boundary edge so that the extended line will intersect with it.

### 4.13 The Break Command

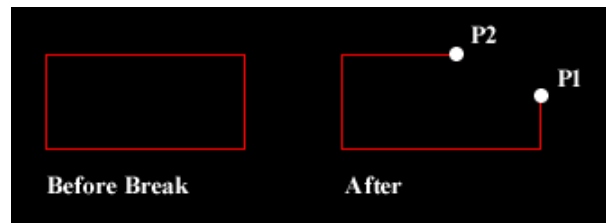
Pull-down menu: Modify>Break

Toolbar: 

Command: BREAK or BR



The Break command enables you to break (remove part of) an object by defining two break points. In the illustration below, a corner of a rectangle has been removed. The Break command can be used with lines, polylines, circles, arcs, ellipses, splines, xlines and rays. When you break an object, you can either select the object using the first break point and then pick the second break point, or you can select the object and then pick the second break point.



### Command Sequence

Command: BREAK

[First break point]<Second break point>: F


Specify first break point: (pick the first break point P1)

Specify second break point: (pick the second break point P2)

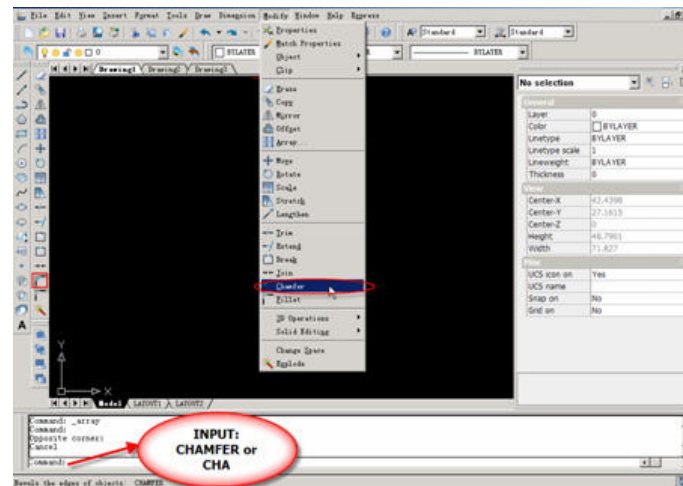
The section of the object is removed and the command ends.

### 4.14 The Chamfer Command

Pull-down menu: Modify>Chamfer

Toolbar: 

Command: CHAMFER or CHA



The Chamfer command enables you to create a chamfer between any two non-parallel lines as in the illustration below or any two adjacent polyline segments. Usually, the Chamfer command is used to set the chamfer distances before drawing the chamfer. Follow the command sequence below where the chamfer distances are changed to 20 before the chamfer is made.





### Command Sequence

Command: CHAMFER

Chamfer dist1=15.0000, dist2=15.0000:

[Distance/Angle/Trim/mEthod/Polyline/Multiple] <Select first entity>: D

Chamfer distance on first entity <15.0000>:10

Chamfer distance on second entity <10.0000>:10

Chamfer dist1=10.0000, dist2=10.0000:

[Distance/Angle/Trim/mEthod/Polyline/Multiple] <Select first entity>: (pick P1)

Select second entity: (pick P2)

The chamfer is made and the command ends.

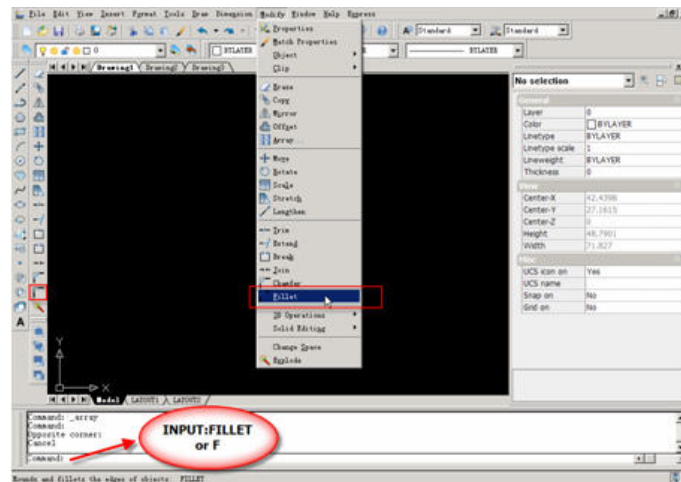
Notice from the command sequence that there are a number of options which can be used to control the way the Chamfer command behaves. The Polyline option can be used to chamfer all vertexes of a polyline simultaneously. The Distance option allows you to specify the two chamfer distances. Angle allows the angle between the first line and the chamfer to be specified. Trim is used to control whether the original lines are trimmed to the chamfer or remain as they are. Finally, Method is used to toggle the command between Distance and Angle mode. When Angle mode is used, the chamfer is defined using one distance and an angle rather than two distances.

## 4.15 The Fillet Command

Pull-down menu: Modify>Fillet

Toolbar: 

Command: FILLET or F



The Fillet command is a very useful tool which allows you to draw an arc between two intersecting lines or adjacent polyline segments. You first need to use the command to set the required radius and then select the two lines.

### Command Sequence

Command: FILLET

Fillet radius=25.0000: [Radius/Polyline/Trim/Multiple]<Select first entity>:R

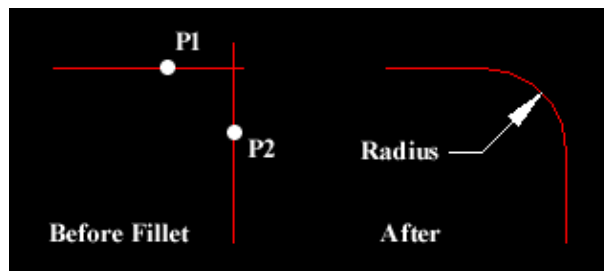
Fillet radius <25.0000>:10

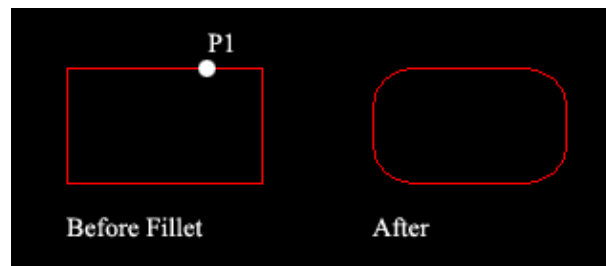
Fillet radius=10.0000: [Radius/Polyline/Trim/Multiple]<Select first entity>:(pick P1)

Select second entity: (pick P2)

The Fillet command can also be used to fillet arcs and circles. The "Polyline" option also allows you to fillet all vertices of a polyline with a single command.

For example, you can easily create the lozenge shape shown on the below from a simple rectangle. Since rectangles are just closed polylines, you can use the Polyline option of the Fillet command to fillet all polyline vertexes simultaneously.





### Command Sequence

Command: FILLET

Fillet radius=10.0000: [Radius/Polyline/Trim/Multiple]<Select first entity>: P

Select 2D polyline to fillet: (pick P1)

4 lines were filleted

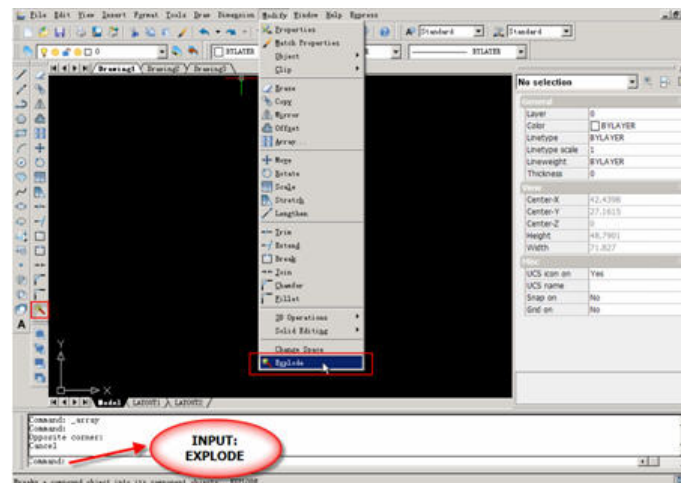
Tip: Make sure that the radius you specify will fit the objects you select, otherwise the fillet command will not work.

### 4.16 The Explode Command

Pull-down menu: Modify>Explode

Toolbar: 

Command: EXPLODE




The Explode command is used to "explode" single object back to their constituent parts. In other words, the command is used to return blocks, polylines etc. (which may be composed of a number of component objects) back to their individual component parts. The change has no visible effect.

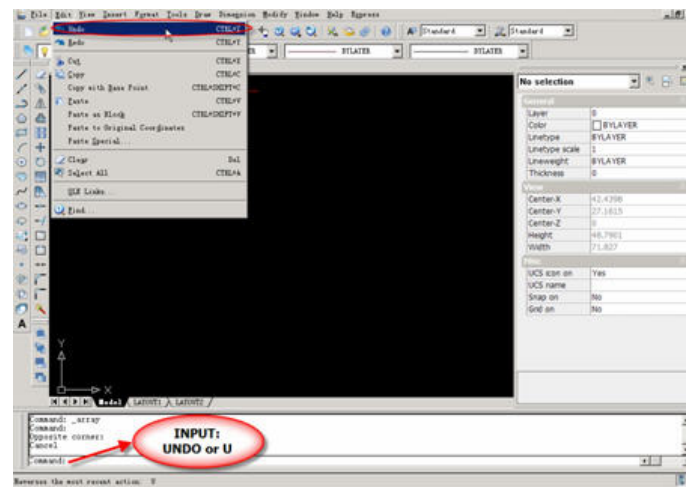
### 4.17 Undo

With UNDO command, Users can inversely cancel executed commands step by step or user-defined number of steps. Undo command is unlimited time undo.

Pull-down menu: Edit >Undo

Toolbar: 

Command : UNDO or U



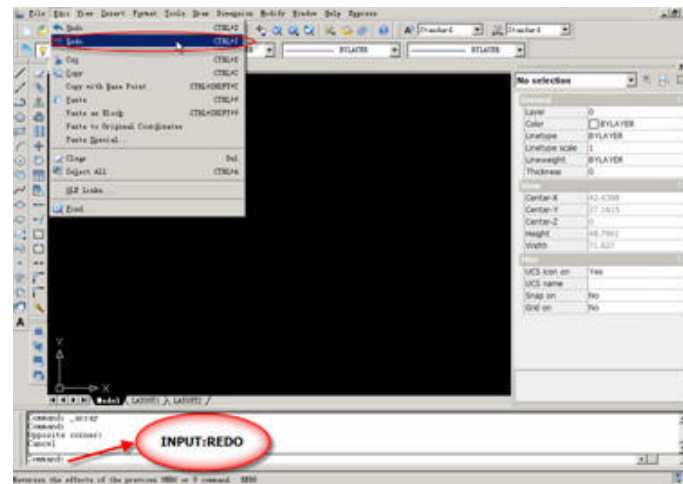
### 4.18 Redo

REDO command is used to restore canceled commands by UNDO step by step. Redo command is unlimited time redo.

Pull-down Menu: Edit → Redo

Toolbar: 

Command : REDO



Commands that are canceled by UNDO previously, can be restored by REDO command.

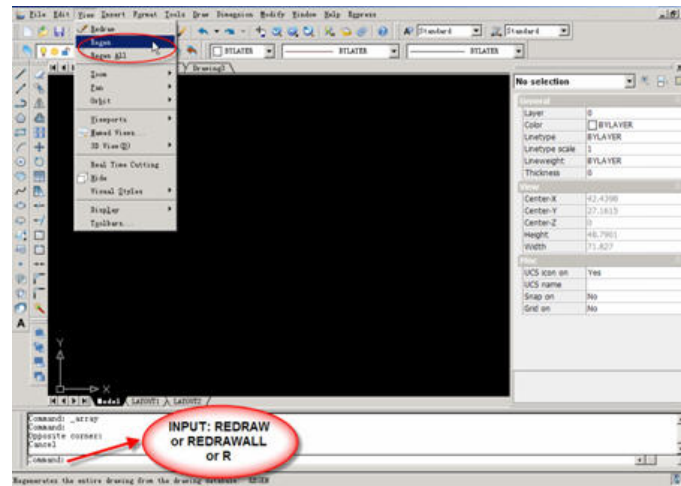
## Chapter 5 Viewing Entities

### 5.1 Redraw

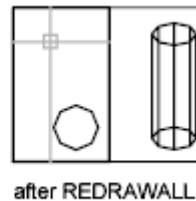
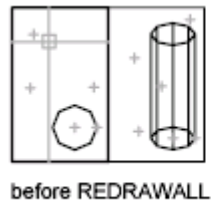
Refresh the display in all viewports

Pull-down menu: View>Redraw

Command : REDRAW or REDRAWALL or R



When BLIPMODE is on, marker blips left by editing commands are removed from all viewports.

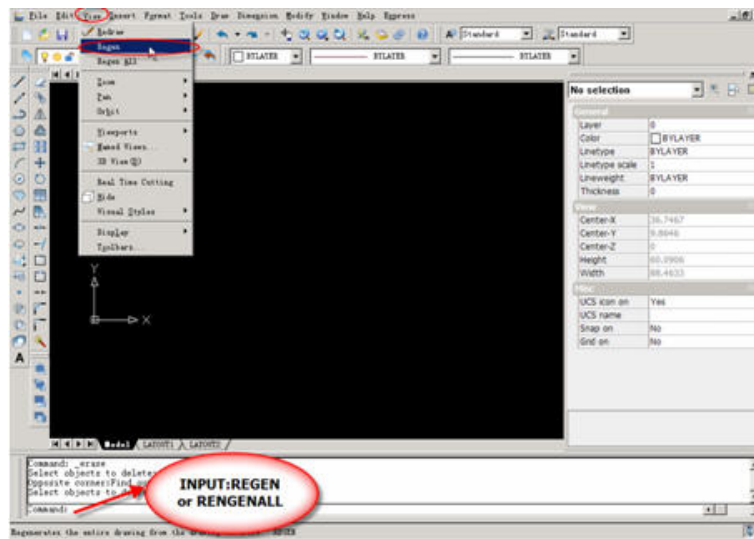


### 5.2 Regen

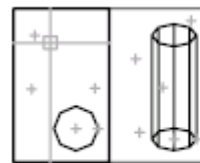
Regenerate the drawing and refresh all viewports

Pull-down menu: View> Regen

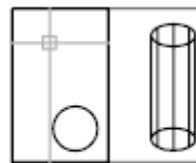
Commands: REGEN or REGENALL



REGENALL regenerates the entire drawing and recomputes the screen coordinates for all objects in all viewports. It also reindexes the drawing database for optimum display and object selection performance.



before REGENALL



after REGENALL

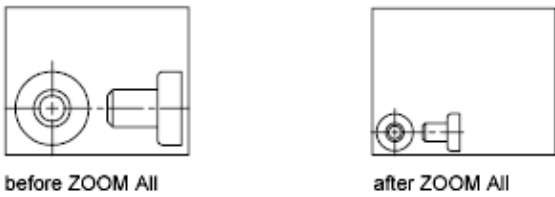
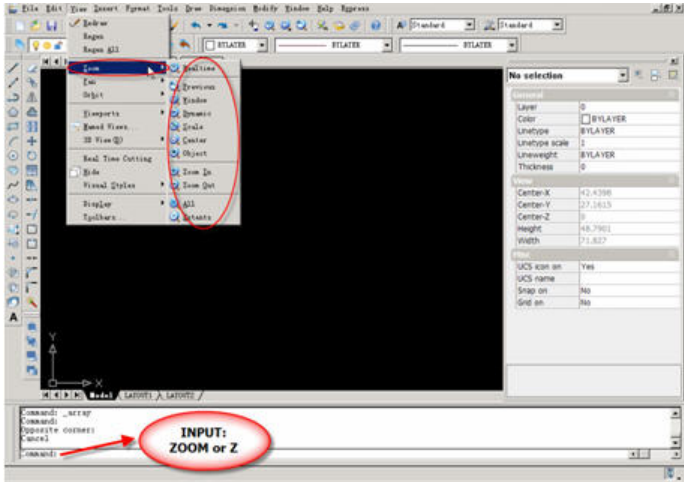
### 5.3 Zoom

Increase or decrease the apparent size of objects in the current viewport

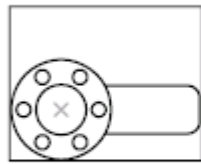
Pull-down menu: View Zoom

Toolbar:

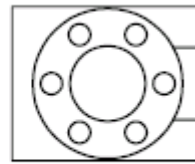
Commands: ZOOM or Z







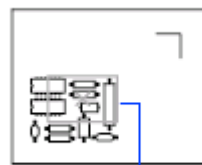
before ZOOM center



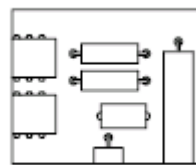
after ZOOM center,  
magnification increased

### Dynamic

Zoom to display the generated portion of the drawing with a view box. The view box represents your viewport, which you can shrink or enlarge and move around the drawing. Positioning and sizing the view box pans or zooms to fill the viewport with the image inside the view box.



view box



new view

The panning view box is displayed first. Drag it to the location you want and click. The zooming view box is then displayed. Resize it and press ENTER to zoom, or click to return to the panning view box.

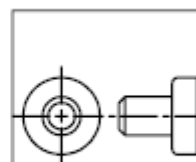
Press ENTER to fill the current viewport with the area currently enclosed by the view box.

### Extents

Zoom to display the drawing extents and result in the largest possible display of all the objects.



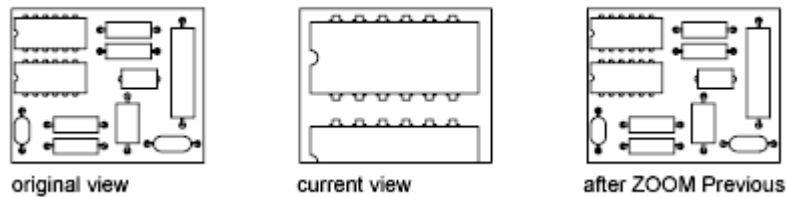
before ZOOM Extents



after ZOOM Extents

### Previous

Zoom to display the previous view. You can restore up to 10 previous views.



Note: If you change the visual style, the view is changed. If you enter ZOOM Previous, it restores the previous view, which is shaded differently but not zoomed differently.

### Scale

Zoom the display at a specified scale factor.

Enter a scale factor (nX or nXP): Specify a value

Enter a value followed by x to specify the scale relative to the current view. For example, entering .5x causes each object to be displayed at half its current size on the screen.



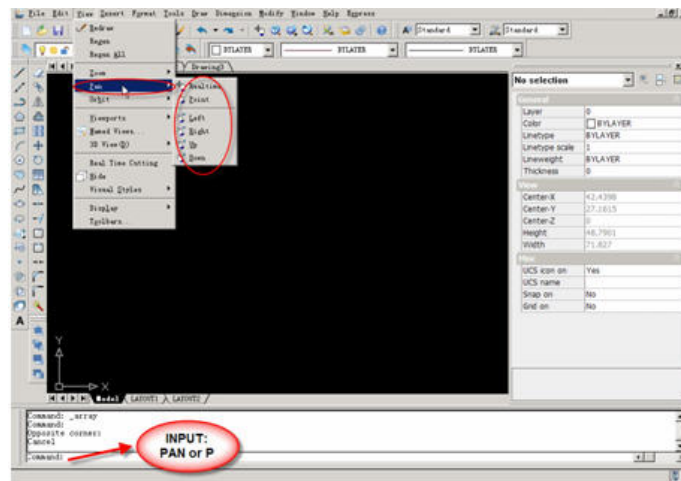
## 5.4 Pan

PAN is used to move the view in the current viewport

Pull-down menu: View>Pan

Toolbar: 

Commands: PAN or P



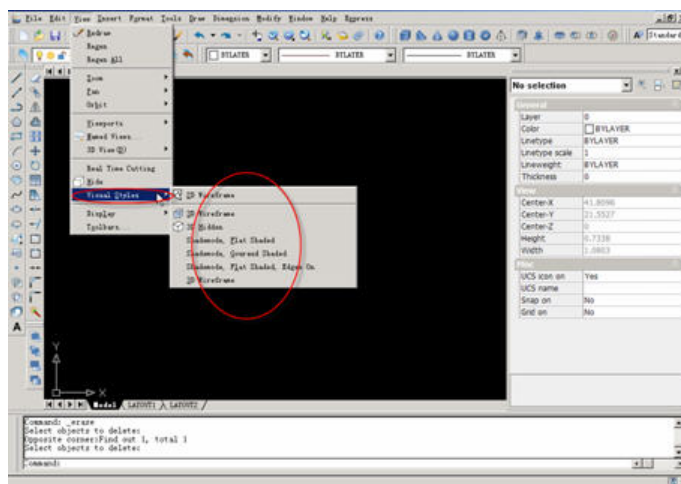
### The usage of pan

When you use wheel-mouse, you can use “PAN” function by pressing the wheel of the mouse. Then a mouse icon is changed like a hand. If you want to use the wheel for panning, the value of button pan should be “on”.

### 5.5 Visual Style

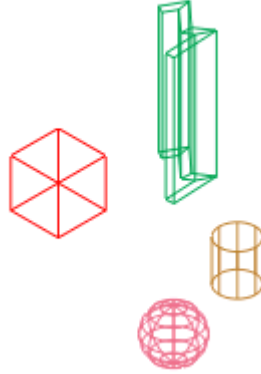
A visual style is a collection of settings that control the display of edges and shading in the viewport. Instead of using commands and setting system variables, you change the properties of the visual style. As soon as you apply a visual style or change its settings, you can see the effect in the viewport.

Pull-down menu: View>Visual style

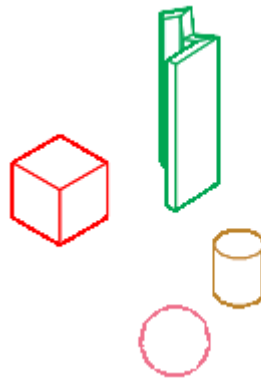


- 2D Wireframe: Display the objects using lines and curves to represent the boundaries, Raster and OLE objects. linetypes, and lineweights are visible.

- 3D Wirefram: Display the objects using lines and curves to represent the boundaries.



- 3D Hidden: Display the objects using 3D wireframe representation and hides lines representing back faces.

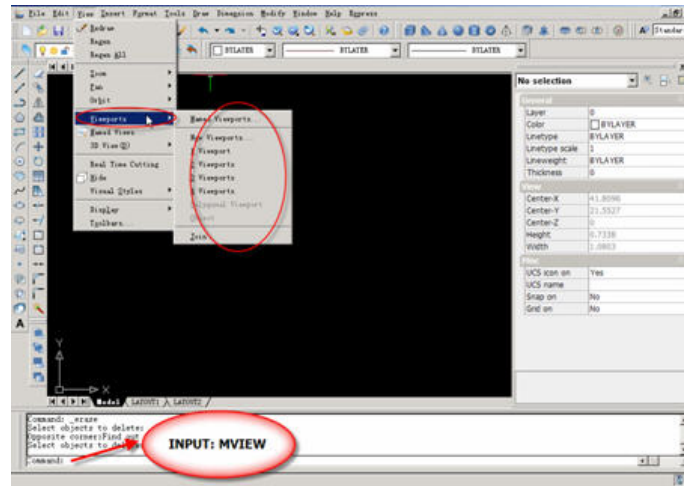


## 5.6 Viewports

Create layout viewports

Pull-down menu: View>Viewports

Command: MVIEW



### The usage of viewports

Command: MVIEW

Viewports: [ON/OFF/Fit/2/3/4/Polygon/Object] <First corner>:

#### Corner of Viewport

Specify the first corner of a rectangular viewport.

Specify opposite corner

#### On

Make a selected viewport active. An active viewport displays objects in model space.

The MAXACTVP system variable controls the maximum number of viewports that can be active at one time. If your drawing contains more viewports than the number specified in MAXACTVP, you must turn one off to make another one active.

Select objects: Select one or more viewports

#### Off

Make a selected viewport inactive. Objects in model space are not displayed in an inactive viewport.

Select objects: Select one or more viewports

#### Fit

Create one viewport that fills the layout to the edges of the printable area. When the paper background and printable area are turned off, the viewport fills the display.



fit

### Polygon

Create an irregularly shaped viewport using specified points. The prompts are similar to those displayed when you specify a polygonal clip boundary for external references, but you can specify arcs when you create a polygonal viewport boundary.

Viewports: [ON/OFF/Fit/2/3/4/Polygon/Object]<First corner>:P

ENTER to use last point [Follow] <Start of polyline>:

[Arc/Distance/Follow/Halfwidth/Width]<Next point>:

### Object

Specify a closed polyline, ellipse, spline, region, or circle to convert into a viewport. The polyline you specify must be closed and contain at least three vertices. It can be self-intersecting, and it can contain an arc as well as line segments.

Select object to clip viewport: Select an object

### 2

Divide the specified area horizontally or vertically into two viewports of equal size.

Enter viewport arrangement [Horizontal/Vertical] <Vertical>: Enter h or press Enter

Specify first corner: Specify a point or press ENTER



2/vertical

First Corner

Position and size new viewports using the window selection method; the viewports are fit into the selected area.

### 3

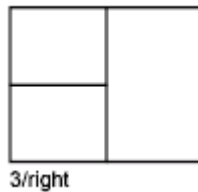
Divide the specified area into three viewports.

Enter viewport arrangement

[Horizontal/Vertical/Above/Below/Left/<Right>: Enter an option or press ENTER

The Horizontal and Vertical options split the specified area into thirds. The other options split the area into three viewports: one large viewport and two smaller ones. The Above, Below, Left, and Right options specify where the larger viewport is placed.

Specify first corner or [Fit] <Fit>: Specify a point or press ENTER



First Corner

Position and size new viewports using the window selection method; the viewports are fit into the selected area.

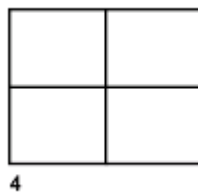
Fit

Size the viewports to fill the drawing area.

**4**

Divide the specified area horizontally and vertically into four viewports of equal size.

Specify first corner or [Fit] <Fit>: Specify a point or press ENTER



First Corner

Position and size new viewports using the window selection method; the viewports are fit into the selected area.

Fit

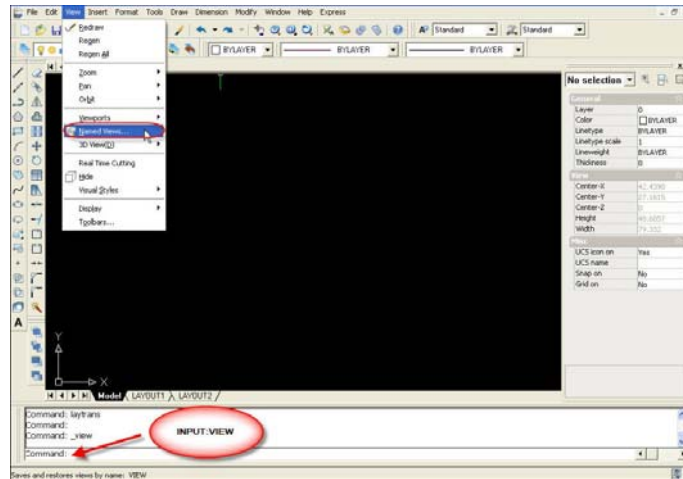
Size the viewports to fill the drawing area.

## 5.7. View Manager

Create, set, rename, modify, and deletes named views, and switch quickly between each views, It is helpful to reduce many unnecessary view adjust operation

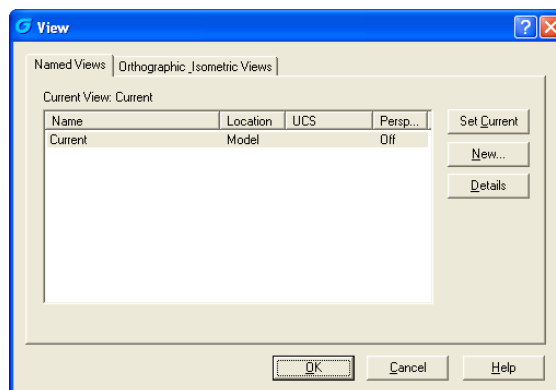
Command: View

Menu: View> Named Views



There are two tabs in this dialog box, named view tab and orthographic isometric view tab.

### 1) Named Views Tab



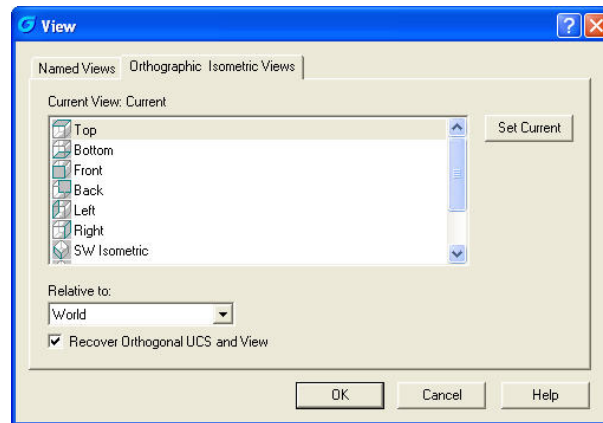
**Set Current:** Restore the selected view.

**New:** Create a named view. And “new view” dialog box will emerge

**Details:** Display “detail of view” dialog box

### 2) Orthographic Isometric View Tab





**Set Current:** Restore the selected view.

**Current View:** Display the name of current view

**Relative to:** Specify the base coordinate system which is used to define the orthographic view, the WCS is the default base coordinate system.

**Recover Orthographic UCS and View:** Restore the relative UCS when set orthographic view as current

## Chapter 6 Dimensions

Dimensioning is the process of adding measurement annotation to a drawing.

You can create dimensions for a variety of object types in many orientations. The basic types of dimensioning are:

Linear

Radial (radius and diameter)

Angular

Ordinate

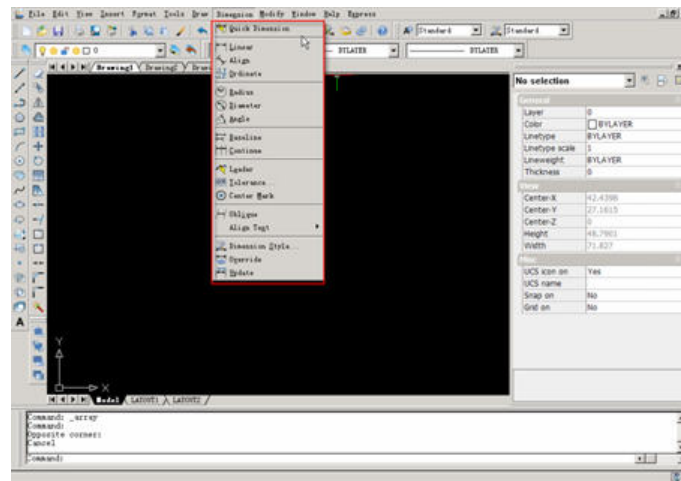
Arc Length

The correct use of dimension tools is the key to produce clear and concise measured drawings.

If you just need to quickly find a description of the various dimension commands, click the appropriate button on the Quick Find toolbar below.



You can also use dimension commands in the pull-down menu.



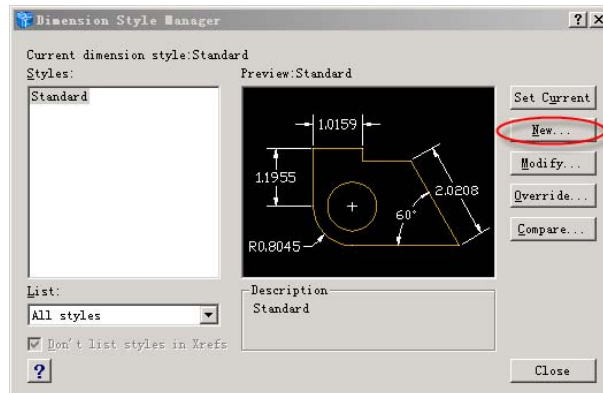
### 6.1 Dimension Style

Dimension styles are the main method used to control the appearance of dimensions.

Using styles you can change the text font, the arrow head style, the relative position of the text, the scale of dimensions and many other parameters.

Pull-down menu: Format>Dimension style or dimension>Dimension style

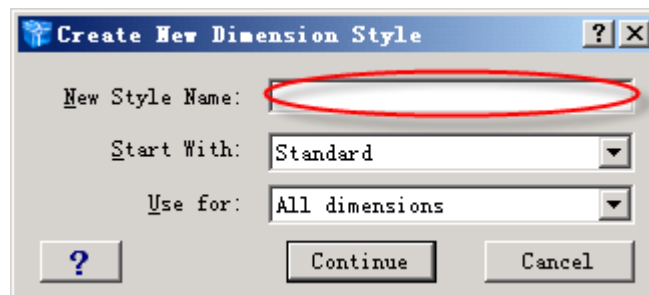
Commands: DDIM or DIMSTYLE



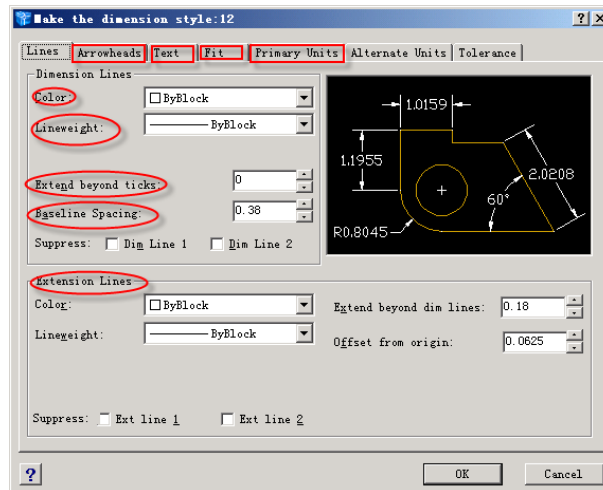
The Dimension Style command can be used to change the appearance of dimensions. The best method is to create a new style before you start creating dimensions so that you can leave the STANDARD style as a default option.

### Creating a new style

To create a new dimension style, make sure the STANDARD style is the current style. Click the “New” button. The new style is automatically set as the current style. Simply input a new style name in the “Create New Dimension Style” dialog box.



Click the “Continue” button, you can see the “Make the Dimension Style” dialogue box, you can set the color, lineweight, extend beyond ticks and baseline spacing of dimension lines and extension lines.



### Setting the Arrowhead

You can select the new arrow type in the dialogue box. If you require different arrowheads at each end of your dimensions you can set the other type using the "2nd" drop-down list. You can also set the arrow size and center marks for circles.

### Setting Text Style

You can select the new text in the dialogue box. You can set the text style, text color, text height and fraction height scale of text appearance. You can also set the text placement and text alignment.

### Setting Dimension Style and Fit

When you are working with drawings which will be plotted at different scales, you will need some way of changing the scale of the dimension lines relative to your drawing so that they always display the same size, irrespective of plotting scale. You can achieve this by using the Scale variable. The scaling applies to individual styles, so you could create different styles with different dimension scales to be used for different plotting scales.

When there isn't enough room to place arrows inside extension lines, you can select the way to move outside the extension. Also when text is not in the default position, you can select the way to place it.

### Setting Main Unit

You can select the unit format, precision of linear dimensions and angular dimensions in

the dialogue box.

You may have a drawing where different units are being used for different elements of the drawing. In such a case it is a good idea to include units to avoid confusion. Remember that the main idea behind dimensioning is to give the maximum amount of information in the clearest and most concise way.

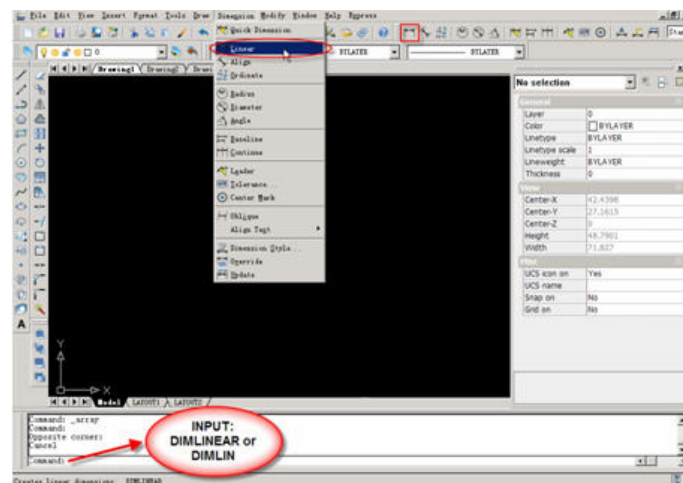
### 6.2 Linear

Input horizontal or vertical dimensions.

Pull-down menu: Dimensions>Linear

Toolbar: 

Commands: DIMLINEAR or DIMLIN

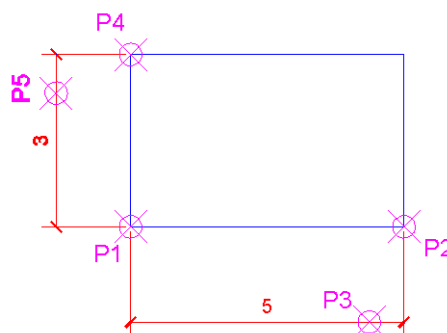


Specify first extension line origin or <select object>: (pick first point of a line)

Specify second extension line origin: (pick second point of a line)

Specify dimension line location or

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]:(pick location of dimension line)



Command: `_dimlinear`

Specify first extension line origin or <select object>: (pick P1)

Specify second extension line origin: (pick P2)

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:  
(pick P3)

Dimension text = 5

Command: `_dimlinear`

Specify first extension line origin or <select object>: (pick P1)

Specify second extension line origin: (pick P4)

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:  
(pick P5)

Dimension text = 3

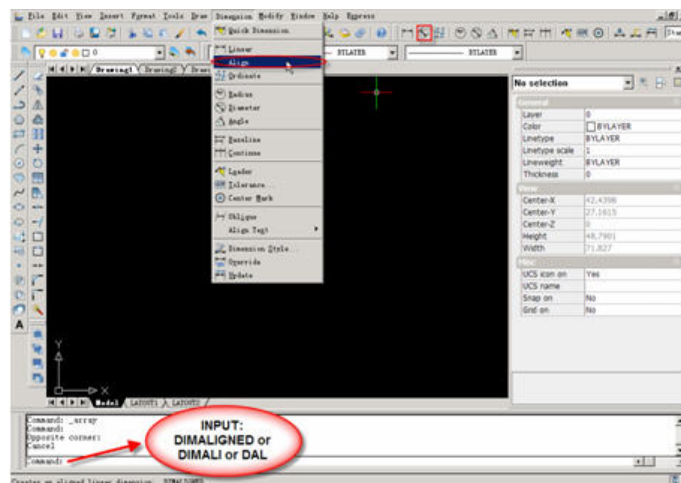
### 6.3 Aligned

Input dimensions parallel with sloped region.

Pull-down menu: Dimensions> Aligned

Toolbar: 

Commands: DIMALIGNED or DIMALI or DAL



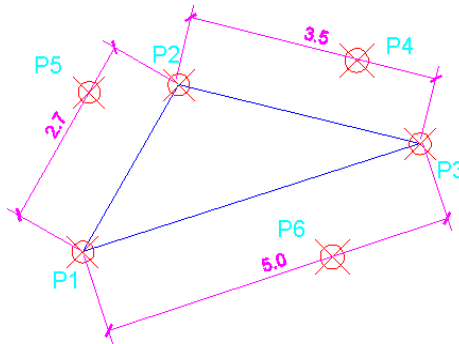
Specify first extension line origin or <select object>: (pick first point of a line)

Specify second extension line origin: (point second point of a line)

Specify dimension line location or [Mtext/Text/Angle]: (pick location of dimension)

line)

### The usage of aligned



Command: `_dimaligned`

Specify first extension line origin or <select object>: (pick P1)

Specify second extension line origin: (point P2)

Specify dimension line location or [Mtext/Text/Angle]: (pick P5)

Dimension text = 2.7

Command: `_dimaligned`

Specify first extension line origin or <select object>: (pick P2)

Specify second extension line origin: (point P3)

Specify dimension line location or [Mtext/Text/Angle]: (pick P4)

Dimension text = 3.5

Command: `_dimaligned`

Specify first extension line origin or <select object>: (pick P1)

Specify second extension line origin: (point P3)


Specify dimension line location or [Mtext/Text/Angle]: (pick P6)

Dimension text = 5.0

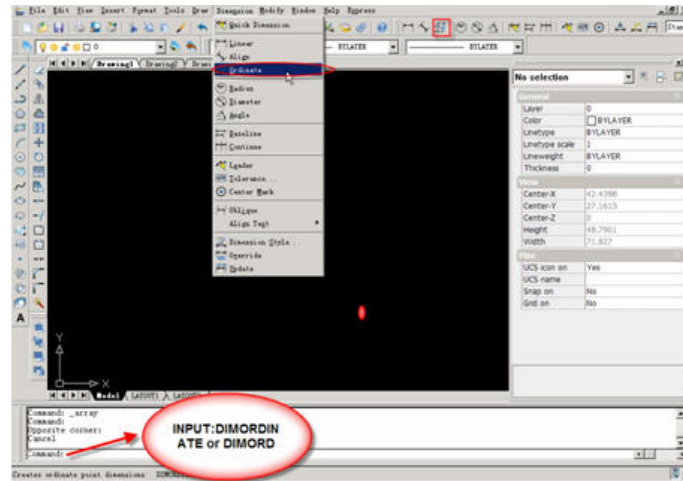
## 6.4 Ordinate

DIMORDINATE is used to make dimensions of relative distance between the origin and entities.

Pull-down menu: Dimensions> Ordinate

Toolbar: 

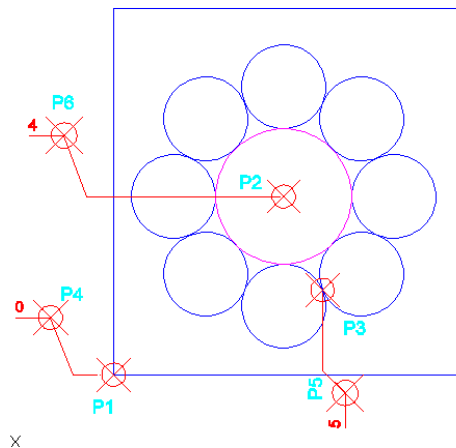
Commands: DIMORDINATE or DIMORD



Specify feature location: (pick dimension point)

Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/Angle]: (pick location of dimension)

## The usage of ordinate



Command: `_dimordinate`

Specify feature location: (pick P1)

Specify leader endpoint or [Xdatum/Ydatum/Mtext/Text/Angle]: (pick P4)

Dimension text = 0

Command: `_dimordinate`

Specify feature location: (pick P3)



Specify leader endpoint or [Xdatum/Y datum/Mtext/Text/Angle]: (pick P5)

Dimension text = 5

Command: `_dimordinate`

Specify feature location: (pick P2)

Specify leader endpoint or [Xdatum/Y datum/Mtext/Text/Angle]: (pick P6)

Dimension text = 4

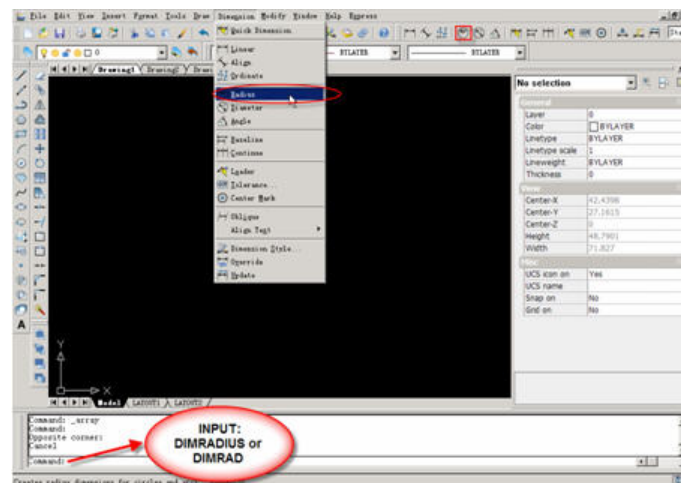
### 6.5 Radius

Input the radius of circles or arcs.

Pull-down menu: Dimensions>Radius

Toolbar: 

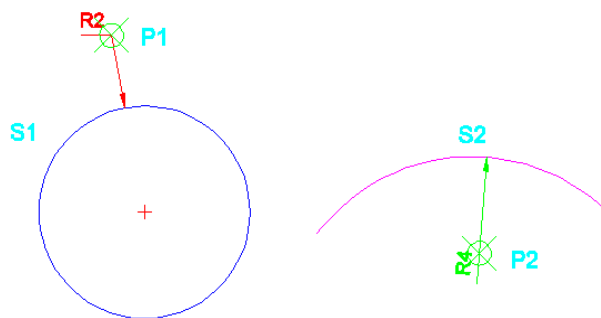
Commands: DIMRADIUS or DIMRAD



Select arc or circle: (select circles or arcs to input dimension)

Specify dimension line location or [Mtext/Text/Angle]: (pick location of dimension)

**The usage of radius**



Command: `_dimradius`

Select arc or circle: (select S1)

Specify dimension line location or [Mtext/Text/Angle]: (pick P1)

Dimension text = 2

Command: `_dimradius`

Select arc or circle: (select S2)

Specify dimension line location or [Mtext/Text/Angle]: (pick P2)

Dimension text = 4

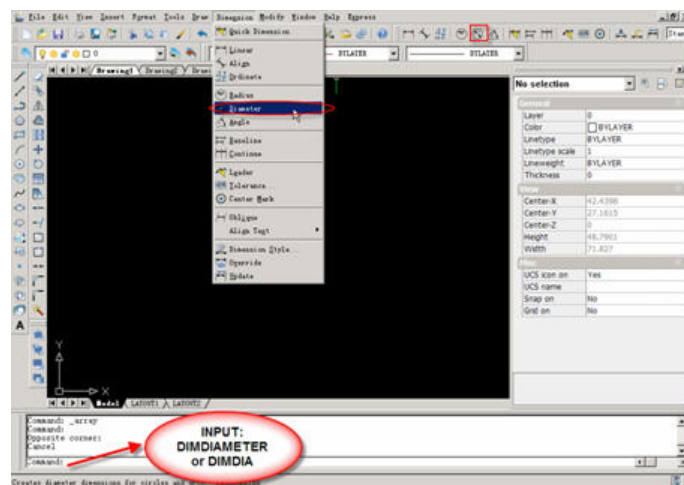
### 6.6 Diameter

Input the diameter of circles or arcs.

Pull-down menu: Dimensions>Diameter

Toolbar: 

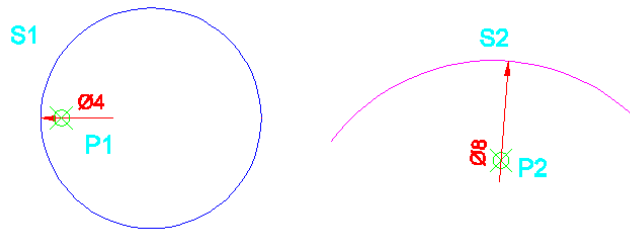
Commands: DIMDIAMETER or DIMDIA



Select arc or circle: (select circles or arcs to input dimension)

Specify dimension line location or [Mtext/Text/Angle]: (pick location of dimension)

**The usage of arc**



Command: `_dimradius`

Select arc or circle: (select S1)

Specify dimension line location or [Mtext/Text/Angle]: (pick P1)

Dimension text =4

Command: `_dimradius`

Select arc or circle: (select S2)

Specify dimension line location or [Mtext/Text/Angle]: (pick P2)

Dimension text =8

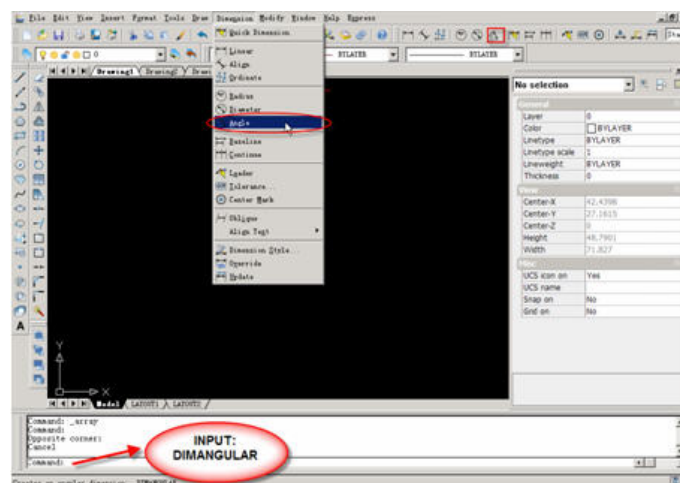
### 6.7 Angular

Input the angular of arc, circle or line.

Pull-down menu: Dimensions>Angle

Toolbar:

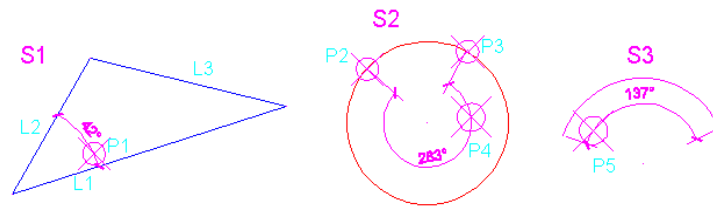
Commands: DIMANGULAR



Select arc, circle, line, or <specify vertex>: (select arc, circle, line to input dimension)

Specify angle vertex:

### The usage of angle



Command: `_dimangular`

Select arc, circle, line, or <specify vertex>: (pick L1)

Select second line: (pick L2)

Specify dimension arc line location or [Mtext/Text/Angle]: (pick P1)

Dimension text = 42

Command: `_dimangular`

Select arc, circle, line, or <specify vertex>: (pick P2)

Specify second angle endpoint: (pick P3)

Specify dimension arc line location or [Mtext/Text/Angle]: (pick P4)

Dimension text = 283

Command: `_dimangular`

Select arc, circle, line, or <specify vertex>: (pick S3)

Specify dimension arc line location or [Mtext/Text/Angle]: (pick P5)

Dimension text = 137

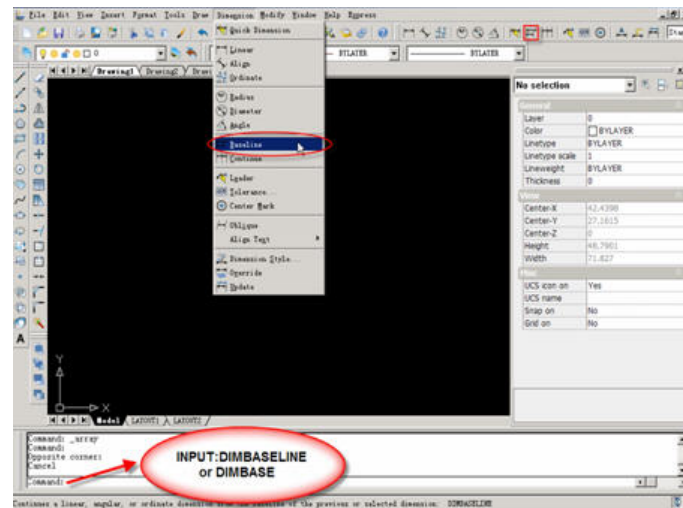
## 6.8 Baseline

BASELINE dimension is used to make dimensions base on one referenced line in order to understand the distances easily as following.

Pull-down menu: Dimensions>Baseline

Toolbar: 

Commands: DIMBASELINE or DIMBASE

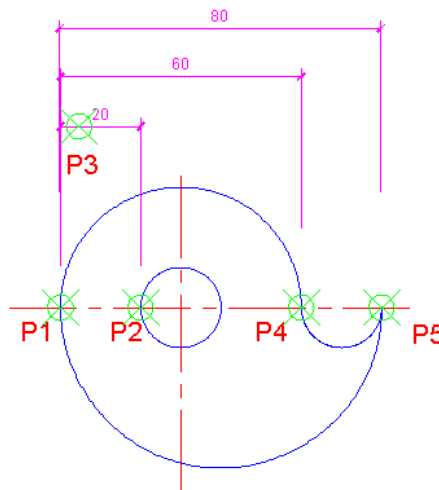


Specify a second extension line origin or [Select] <Select>: (pick second point)

Select base dimension:

### The usage of baseline

First of all, after you input a horizontal dimension, input BASELINE dimensions.



Command: `_dimbaseline`

Specify first extension line origin or <select object>: (pick P1)

Specify second extension line origin: (pick P2)

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:  
(pick P3)

Command: `_dimbaseline`

Specify a second extension line origin or [Select] <Select>: (pick P4)

Dimension text = 60

Specify a second extension line origin or [Select] <Select>: (pick P5)

Dimension text = 80

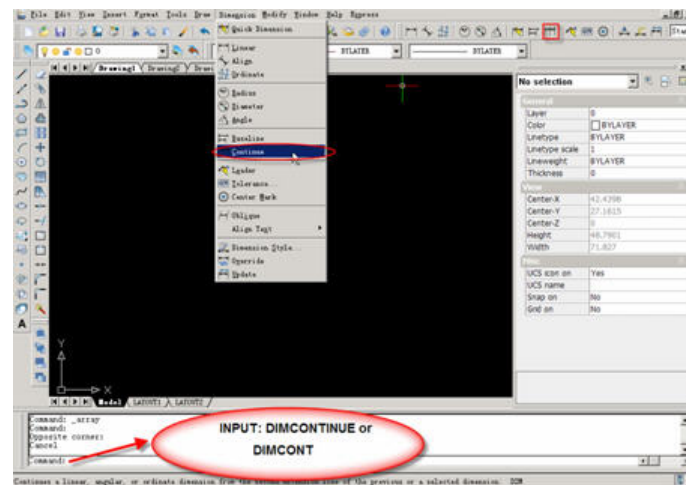
### 6.9 Continue

DIMCONTINUE is used to make dimensions continuously as following.

Pull-down menu: Dimensions>Continue

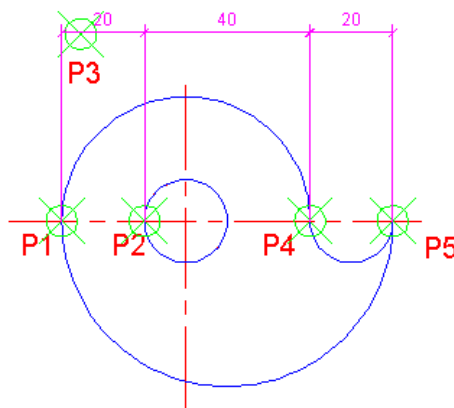
Toolbar: 

Commands: DIMCONTINUE or DIMCONT



### The usage of continue

First of all, after you input a reference dimension, input continuous dimensions.



Command: `_dimcontinue`

Specify first extension line origin or <select object>: (pick P1)

Specify second extension line origin: (pick P2)

Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]:  
(pick P3)

Command: `_dimcontinue`

Specify a second extension line origin or [Select] <Select>: (pick P4)

Dimension text = 40

Specify a second extension line origin or [Select] <Select>: (pick P5)


Dimension text = 20

Specify a second extension line origin or [Select] <Select>:

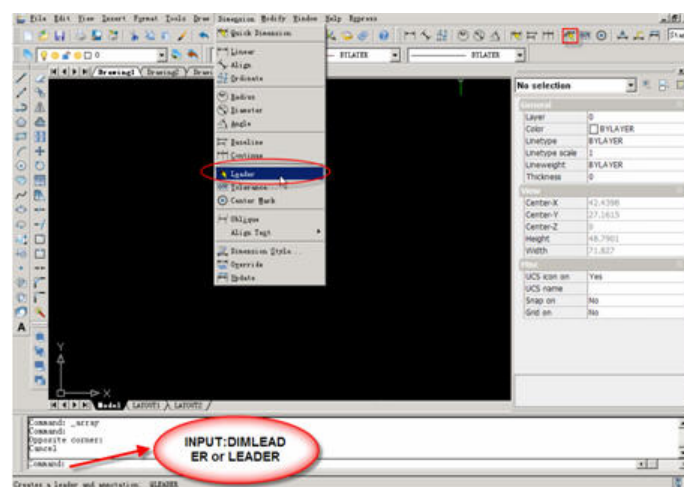
### 6.10 Leader

A leader object is a line or a spline with an arrowhead at one end and a multiline text object or block at the other. In some cases, a short horizontal line, called a landing, connects text or blocks and feature control frames to the leader line.

Pull-down menu: Dimensions>Leader

Toolbar: 

Commands: DIMLEADER or LEADER



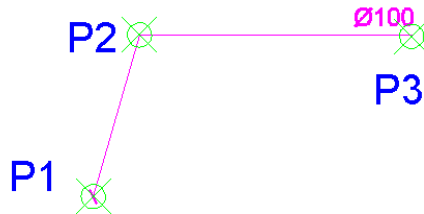
Specify first leader point: (pick start point)

Specify next point: (pick next point)

Specify next point, or [Annotation(A)/Format(F)/Undo(U)] <Annotation>:(pick next point)

Enter first line of annotation text, or <Setting>: (input dimension text)

### The usage of leader



Command: `_leader`

Specify first leader point, or [Settings] <Settings>: (pick P1)

Length/<Next point>: (pick P2)

Length/<Next point>: (pick P3)

Specify the text width <2.000000>:

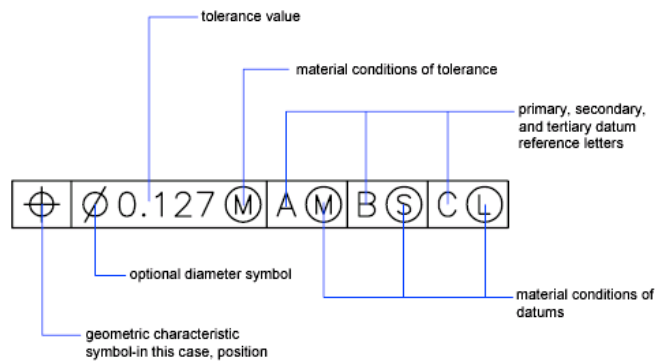
Enter first line of annotation text <Mtext>:%%C100

## 6.11 Tolerance

Geometric tolerances show acceptable deviations of form, profile, orientation, location, and runout of a feature. You add geometric tolerances in feature control frames. These frames contain all the tolerance information for a single dimension. Geometric tolerances can be created with or without leader lines, depending on whether you create them with TOLERANCE or LEADER.

A feature control frame consists of two or more components. The first feature control frame contains a symbol that represents the geometric characteristic to which a tolerance is being applied, for example, location, profile, form, orientation, or runout. Form tolerances control straightness, flatness, circularity and cylindricity; profiles control line and surface. In the illustration, the characteristic is position.

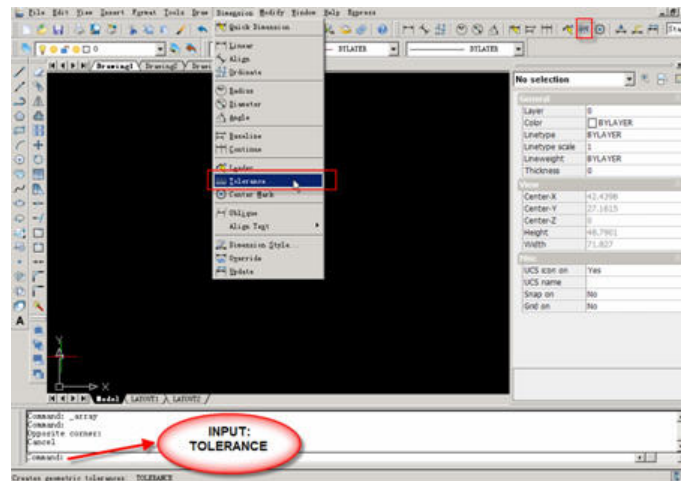




Pull-down menu: Dimensions>Tolerance

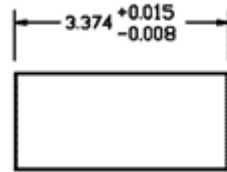
Toolbar: 

Command: TOLERANCE

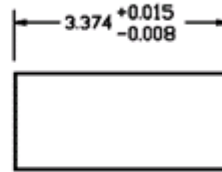


### To create a tolerance dimension

1. Command: TOLERANCE
2. Click the Tolerance tab.
3. Select the Append Dimension Tolerances check box.
4. Type or select the lower tolerance limit.
5. Type or select the upper tolerance limit.
6. Click OK.
7. Insert the dimension.



Tolerance dimension.




Limits dimension.

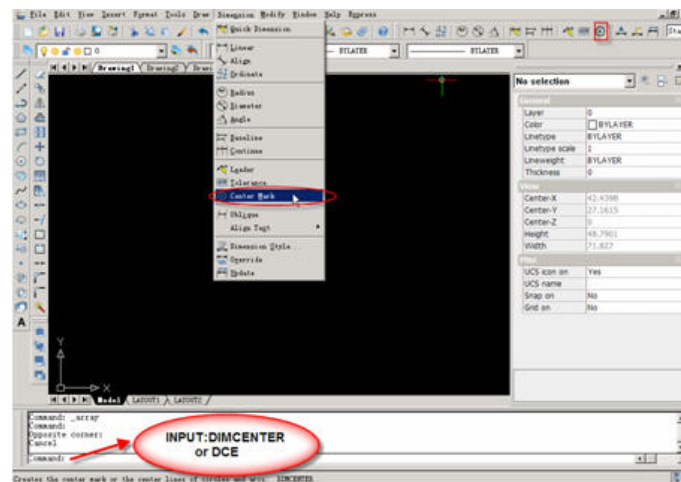
## 6.12 Center Mark

DIMCENTER is used to make dimensions of circles or arcs. If “DIMCENTER” is a negative (-) value, center line is drawn. If it is a positive (+) value, center mark is drawn.

Pull-down menu: Dimensions>Center Mark

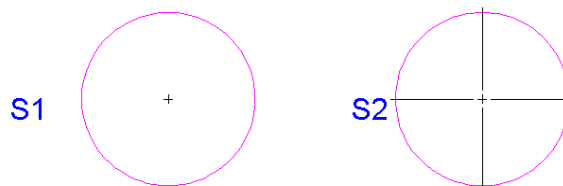
Commands: DIMCENTER or DCE

Toolbar: 



Select arc or circle: (select circles or arcs)

**The usage of center mark**



Command: DIMCEN

New current value for DIMCEN<0.0100>: 1

Command: \_DIMCENTER

Select arc or circle: (select S1)

Command: DIMCEN

New current value for DIMCEN<1.0000>: -1

Command: \_DIMCENTER

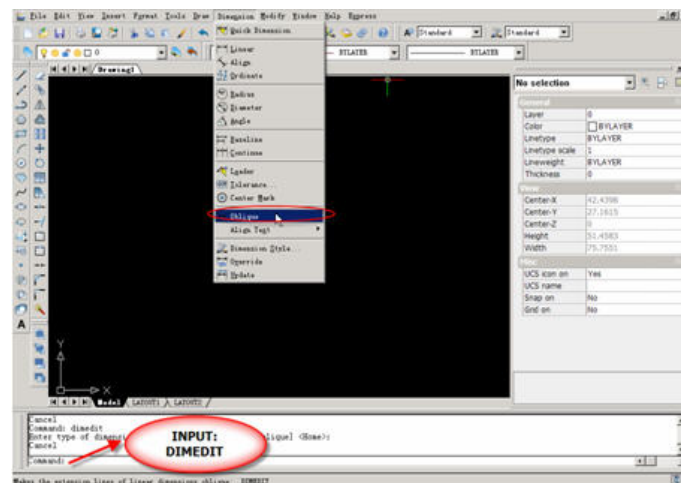
Select arc or circle: (select S2).

### 6.13 Making Dimensions Oblique

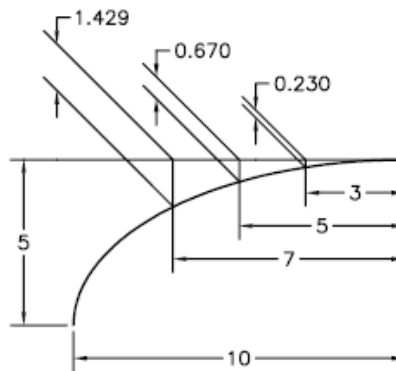
You can create dimensions with extension lines that are not perpendicular to their dimension lines. Extension lines are created perpendicular to the dimension line. However, if the extension lines conflict with other objects in a drawing, you can change their angle after the dimension has been drawn. New dimensions are not affected when you make an existing dimension oblique.

Pull-down menu: Dimensions>Oblique

Command: DIMEDIT then choose Oblique



**The example of dimension oblique**



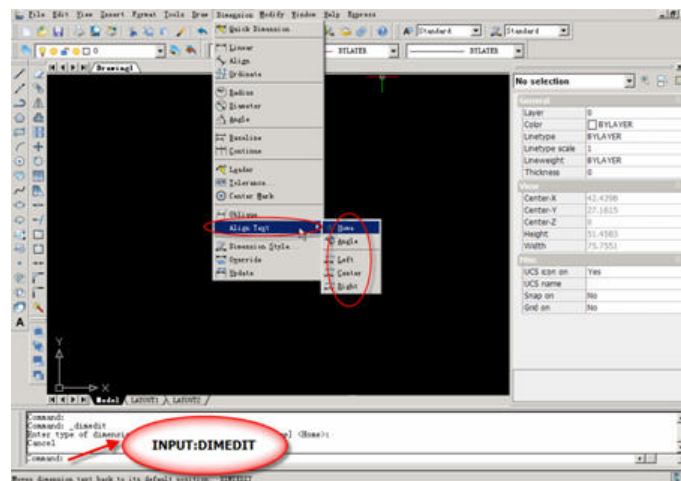
### 6.14 Editing Dimension Text

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 apply the change. All the selected dimensions are updated simultaneously.

Pull-down menu: Dimensions>Align Text

Command: DIMEDIT



#### Return dimension text to its home position

Click Dimension menu >Align Text>Home.

Select the dimension text you want to return to its home position.

#### Rotate dimension text

Click Dimension menu >Align Text>Angle.

Select the dimension to edit.

Enter the new angle for the text.

### **Move text to the left side of the dimension line**

Click Dimension menu>Align Text>Left.

Select the dimension.

The dimension text is left-justified along the dimension line inside the extension lines.

You can choose the Center or Right options to move the text to the center or right of the dimension line.

### **Move text to the center of the dimension line**

Click Dimension menu>Align Text >Center

Select the dimension.

The dimension text is center-justified along the dimension line inside the extension lines.

You can choose the right or Left options to move the text to the right or left of the dimension line.

### **Move text to the right side of the dimension line**

Click Dimension menu>Align Text>Right

Select the dimension.

The dimension text is right-justified along the dimension line inside the extension lines.

You can choose the Center or Left options to move the text to the center or left of the dimension line.

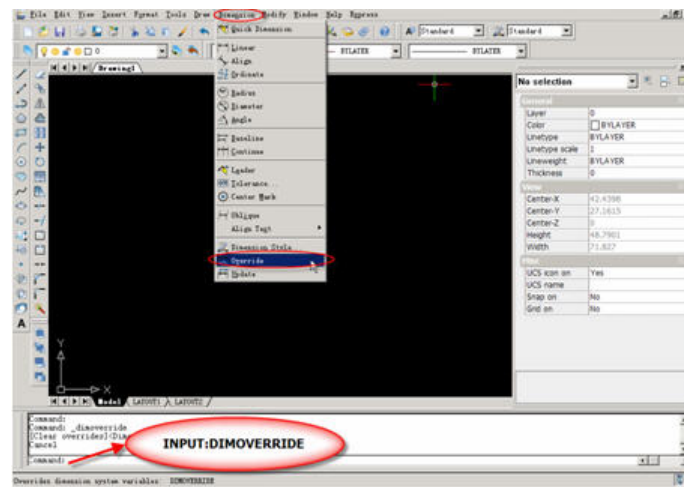
## **6.15 Override**

With dimension style overrides, you can temporarily change a dimensioning system variable without changing the current dimension style.

A dimension style override is a change made to specific settings in the current dimension style. It is equivalent to changing a dimensioning system variable without changing the current dimension style.

Pull-down menu: Dimension>Override

Command: DIMOVERRIDE



### Set up dimension style overrides

Click Dimension menu>Dimension style.

In the Dimension Style Manager, under Styles, select the dimension style for which you want to create an override. Click Override.

In the Override Current Style dialog box, make changes to the dimension style by clicking the appropriate tab.

Click OK to return to the Dimension Style Manager.

The dimension style overrides are listed below the style they modify in the dimension style name list.

Click Close.

### To apply dimension style overrides

Click Dimension menu>Dimension style.

In the Dimension Style Manager, click Override.

In the Override Current Style dialog box, enter the style overrides. Click OK.

The program displays <style overrides> below the dimension style name in the Dimension Style Manager dialog box. After you create dimension style overrides, you can continue to modify dimension styles, compare them with other dimension styles, or delete or rename the overrides.

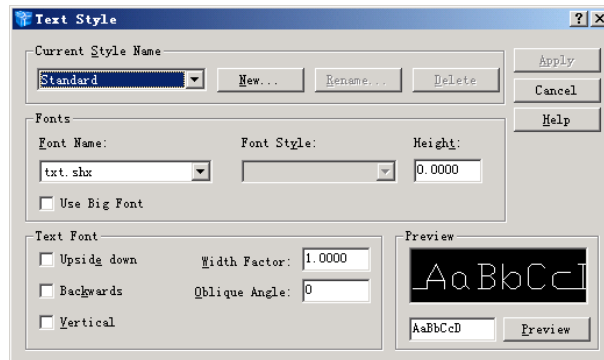
## Chapter 7 Text

### 7.1 Text Style

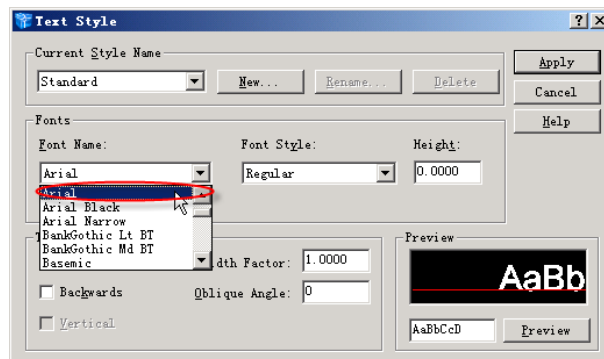
STYLE command is used to set up text font, scale and angle ratio of the text.

Pull-down menu: Format>Text Style

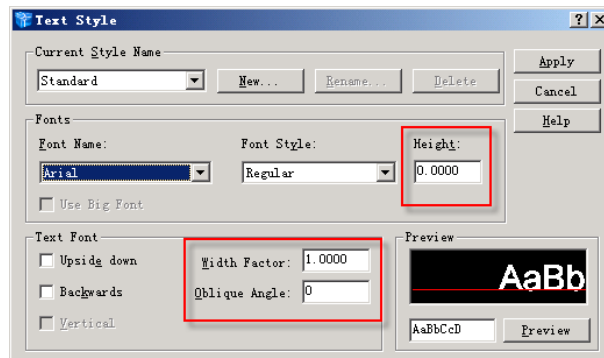
Command : STYLE



- Select text font when dialogue box is displayed.



- Change text height, width factor and oblique angle. Width factor is a ratio of width-length of the text.



- After selecting text font then click “apply”

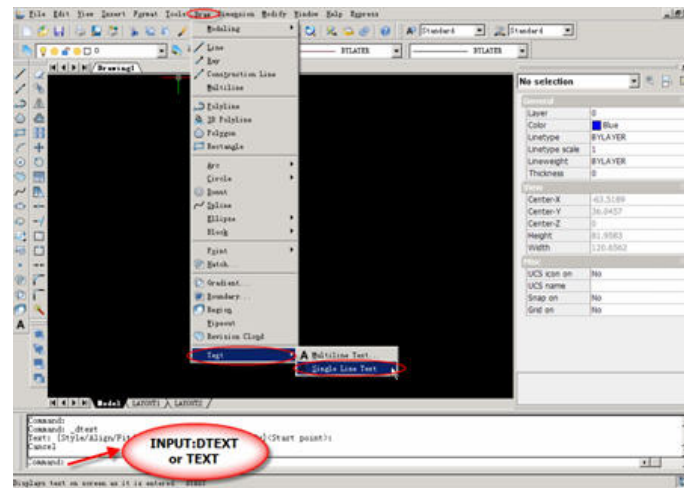
## 7.2 Single Line Text

DTEXT is used to input the text in order to mark the title and part table in the drawing.

(1) Execution

Pull-down menu: Draw>Text>Single Line Text

Command: DTEXT or TEXT



Text: Style/Align/Fit/Center/Middle/Right/Justify/<Start point>:(pick start point)

Height of text <3.0000>: (input text height)

Rotation angle of text <0>: (input text slope)

Text: (input text)

(2) Usage

- Input text



Command: `_dtext`

Text Style/Align/Fit/Center/Middle/Right/Justify/< Start point>: (pick P1)

Height of text<3.0000>:3

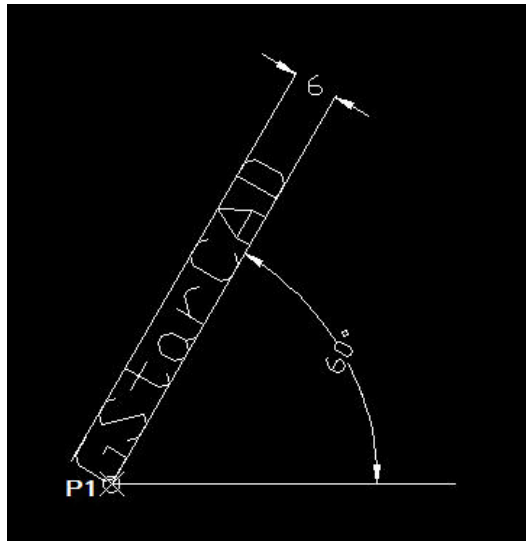
Rotation angle of text<0>:

Text: GStarCAD

Text:



- Change the height and slope of the text



Command: `_dtext`

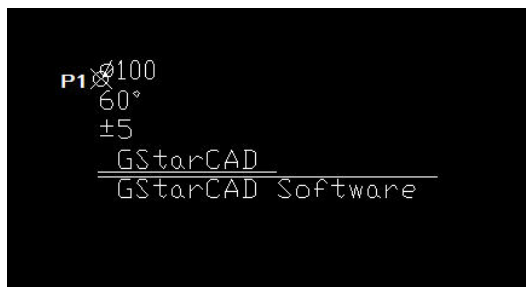
Text Style/Align/Fit/Center/Middle/Right/Justify/< Start point>: (pick P1)

Height of text<3.0000>: 6

Rotation angle of text<0>: 60

Text: GStarCAD

- Use special text



Command: `_dtext`

Text Style/Align/Fit/Center/Middle/Right/Justify/< Start point: (pick P1)

Height of text<6.0000>: 3

Rotation angle of text<0>:

Text: `%%C100` (mark diameter)

Text: `60%` (mark angle)

Text: `%%P5` (mark tolerance)

Text: `%%U GStarCAD` (draw underline)

Text: %%O GStarCAD Software (draw underline)

Text:

### 7.3 Multiline Text

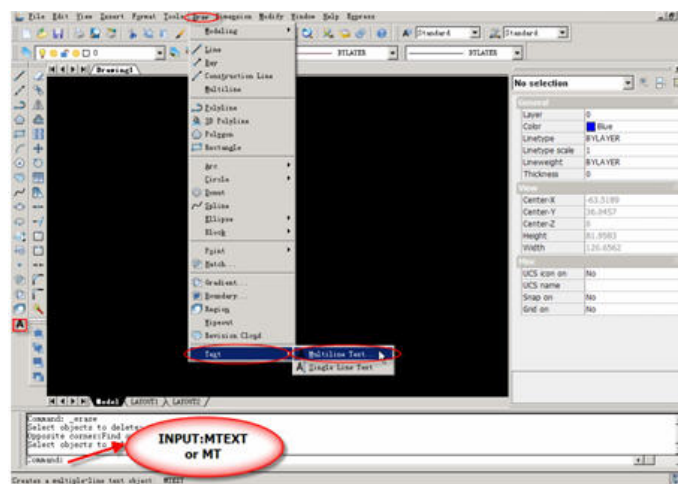
While DTEXT is used to input the text in a line, MTEXT is used to input multiple lines in indicated area using the text editor.

(1) Execution

Pull-down menu: Draw>Text>Multiline Text

Toolbar: 

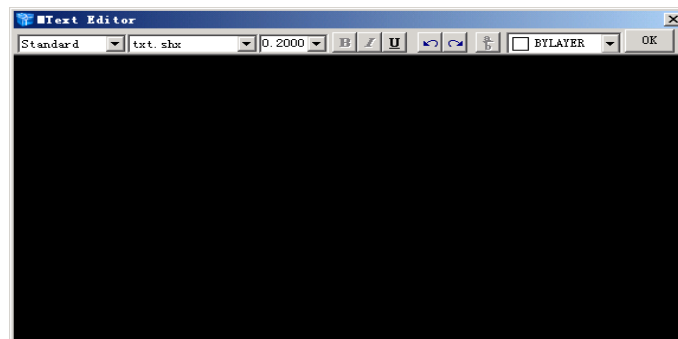
Commands: MTEXT or MT



Multiline Text: First corner for block of text: (pick upper-left point of text input area)

Justification/Rotation/Style/Height/Direction/Width/<Opposite corner for block of text>:

(pick lower-right point of text input area)



(2) Usage

- Input the text in multiple lines on the screen.

P2

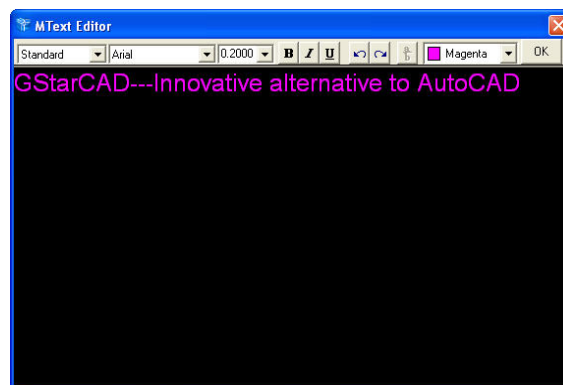
P1

Command: `_mtext`

Multiline Text: First corner for block of text

Justification/Rotation/Style/Height/Direction/Width/ <Opposite corner for block of text>:

- Select a font and size of the text to use. Then input text.



- Select "OK" button.



### 7.4 Ddedit

DDEDIT is used to modify colors, layers, size, font and angle of the text by selecting texts that are already inputted.

(1) Execution

Commands: DDEDIT or ENTPROP

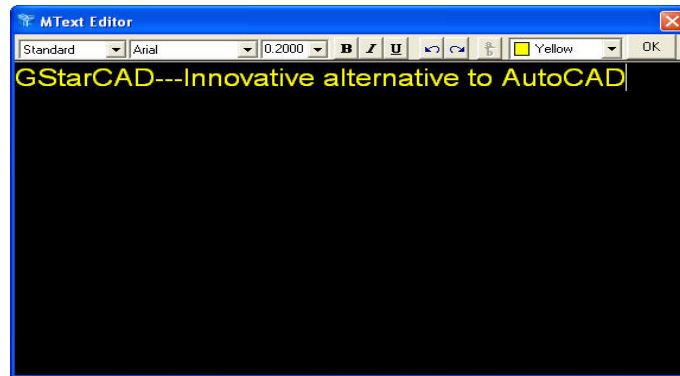
Select entities to modify: (select text to modify)

(2) Usage

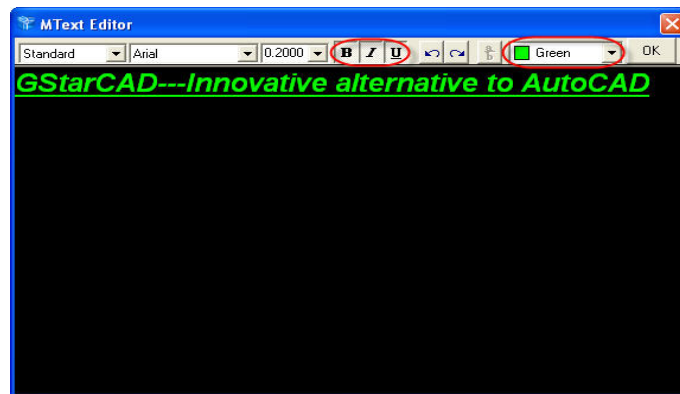
- Modify the text.

Command: Ddedit

Select entities to modify



- Modify selected texts.



## **Chapter 8 Drawing Aids**

### **8.1 Ortho Mode**

Status Bar: 

Pull-down menu: None

Command: ORTHO or F8

Ortho is short for orthogonal, which means either vertical or horizontal. Like the other options on the status bar, Ortho is not really a command, it is a drawing mode which can either be turned on or off. Ortho mode can be toggled on or off in one of three ways. The quickest way is just to click on the ORTHO button on the status bar.

Ortho can also be toggled on and off using the F8 Function key .Finally, you can also type ORTHO at the command line.

#### **Command Sequence**

Command: ORTHO

Enter mode [ON/OFF] <OFF>: (type ON or OFF)

### **8.2 The Drawing Grid**

Status Bar: 

Pull-down menu: None

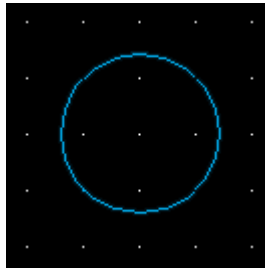
Command: GRID or F7

The drawing grid is a regular pattern of dots displayed on the screen which acts as a visual aid, it is the equivalent of having a sheet of graph paper behind your drawing on a drawing board. You can control the grid spacing, so it can give you a general idea about the size of drawn objects. It can also be used to define the extent of your drawing.

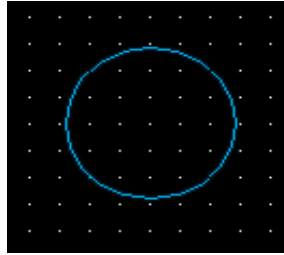
#### **Command Sequence**

Command: GRID

Specify grid spacing(X) or [ON/OFF/Snap/Aspect] <10.000>: (enter grid spacing)



Grid spacing set to 10 drawing units



Grid spacing set to 5 drawing units

Although you can use the command line to control the visibility of the grid by using the "ON" and "OFF" options this is more easily achieved using the F7 key or, better still, by clicking the GRID button on the status bar. However, the command line does offer some additional options. The Snap option allows you to automatically set the grid spacing to the current snap spacing. You can also change the aspect ration of the grid. By default, the X and Y spacing of the Grid are the same, resulting in a regular square matrix of grid points. But you can display a grid with different X and Y spacing by using the "Aspect" option.

Grid mode and X/Y spacing can also be set using the Drafting Settings dialogue box. You can access grid settings by right-clicking the Grid button on the status bar and selecting “Snap and Grid” from the menu.

You may have noticed that the grid does not extend infinitely in all directions. In fact, it is only displayed within a finite rectangle. You can control the extent of the visible grid using Drawing Limits.

### 8.3 Snap Mode

Status Bar: 

Pull-down menu: None

Command: SNAP or F9

With Snap mode turned on GStarCAD only allows you to pick points which lie on a regular grid. The Snap grid is completely independent of the display grid. However, the Grid spacing and Snap spacing are usually set to the same value to avoid confusion. You can force the display grid to conform with the snap grid by setting the display grid spacing to zero. The display grid will automatically change each time when the snap grid is changed. When Snap mode is turned on and the Grid is displayed, the Snap and Grid spacings are the same and the crosshairs will jump from one grid point to another as you move across the screen. This makes it very easy to draw objects which have a regular shape. The Snap command is used to set the snap spacing and to toggle Snap mode.

### **Command Sequence**

Command: SNAP

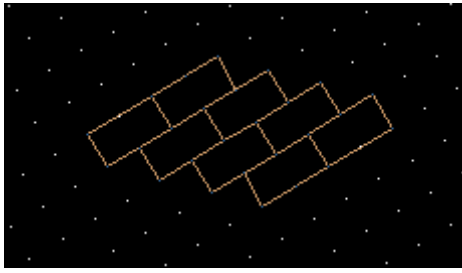
Snap is on, x and y = 5.0000: [OFF/Rotate/Style/Aspect] <Snap spacing>:

The "Aspect" option can be used to vary the horizontal and vertical snap spacings independently.

"Rotate" is used to set the snap grid to any angle.

You can also set the snap style to either Isometric or Standard (the default) using the "Style" option. The Standard style is used for almost all drawing situations including detail drawings in Orthographic Projection. The Isometric style is specifically to aid the creation of drawings in Isometric Projection (see the illustrations on the right).

The grid snap is particularly useful if you need lots of modular objects such as bricks or paviors. In the illustration on the below, the Aspect option is used to set the X and Y snap spacings to the brick dimensions and the Rotate option is used to set the orientation of the bond. Once these settings are made, the bricks can be accurately drawn without any other drawing aids.

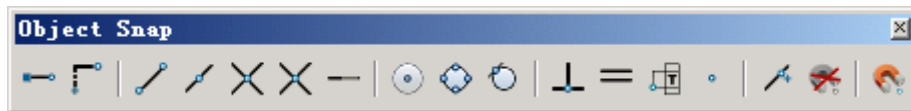


All of the Snap variables can also be set using the Drafting Settings dialogue box. Right-click on the SNAP button and choose “Snap and Grid” from the menu.

### 8.4 Object Snaps

The main reason for using the Object Snaps is that they are used very frequently. Experienced users will use Object Snaps all of the time, because they are the only way to make sure that the objects you are drawing are drawn accurately.

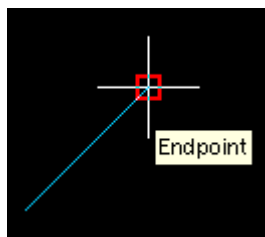
Object Snap Toolbar:




#### 1. Endpoint

Toolbar: Object Snap 

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

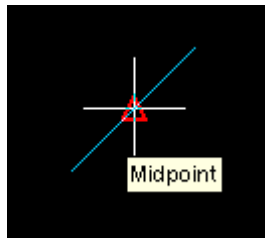


#### 2. Midpoint

Toolbar: Object Snap 

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

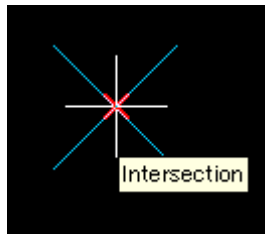




### 3. Intersection

Toolbar: Object Snap 

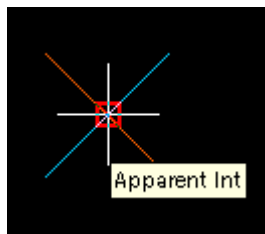
The Intersection Osnap snaps to the physical intersection of any two drawing objects (i.e. where lines, arcs or circles etc. cross each other) and to Polyline vertices.



### 4. Apparent Intersect

Toolbar: Object Snap 

Apparent Intersection snaps to the point where objects appear to intersect in the current view. For example, you may be looking at a drawing in plan view where two lines cross, as in the illustration. However, since GStarCAD is a 3 dimensional drawing environment, the two lines may not physically intersect. One line may be at ground level and the other may be 10 meters or more above or below ground level. As with the Intersection Osnap, Apparent Intersection also has an "Extended" mode.

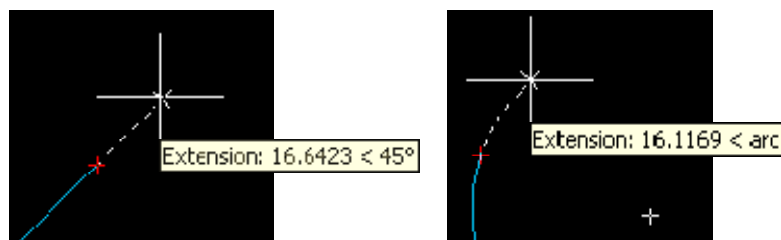


### 5. Extension

Toolbar: Object Snap 

The Extension Osnap enables you to snap to some point along the imaginary extension

of a line, arc or polyline segment. To use this osnap, you must hover the cursor over the end of the line you want to extend. When the line end is found, a small cross appears at the endpoint and a dashed extension line is displayed from the endpoint to the cursor, providing the cursor remains close to the extension. The Snap Tip for Extension also includes the relative polar co-ordinate of the current cursor position. This can be a useful guide for positioning your next pick point. The co-ordinate includes a distance from the endpoint and the angle of the extension. In the case of the arc extension, the Snap Tip displays the distance along the arc.

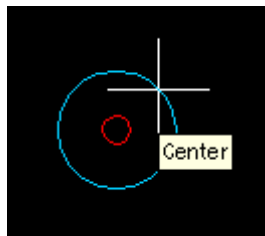


### 6. Center

Toolbar: Object Snap



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

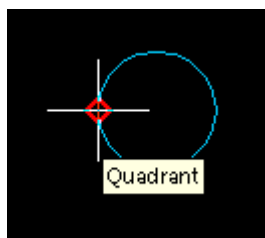


### 7. Quadrant

Toolbar: Object Snap



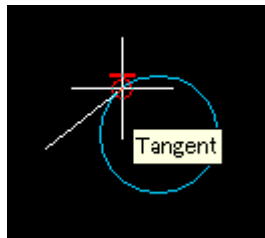
The Quadrant Osnap snaps to one of the four circle quadrant points located at north, south, east and west or 90, 270, 0 and 180 degrees respectively.




### 8. Tangent

Toolbar: Object Snap 

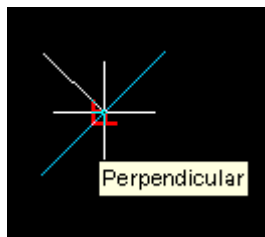
The Tangent Osnap snaps to a tangent point on a circle. This osnap works in two ways. You can either draw a line from a point to the tangent point (see illustration) or you can draw a line from a tangent point, the latter is referred to as the "Deferred Tangent" snap mode.



### 9. Perpendicular

Toolbar: Object Snap 

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.

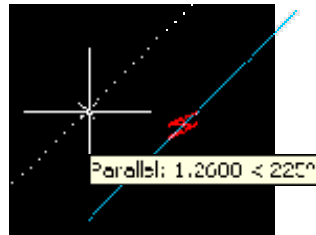


### 10. Parallel

Toolbar: Object Snap 

The Parallel Osnap is used to draw a line parallel to any other line in your drawing. In operation, this osnap is slightly less intuitive than some of the others. To draw a parallel line, first start the Line command, specify the first point when prompted and then start the Parallel Osnap. Hover the cursor over an existing line until you see the Parallel snap marker. Now, move the cursor close to a parallel position and a dotted line will appear,

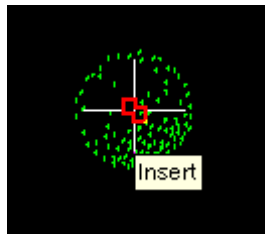
indicating the parallel. You can now pick the second point of your line.




### 11. Insert

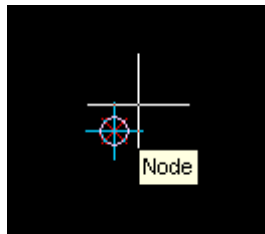
Toolbar: Object Snap 

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

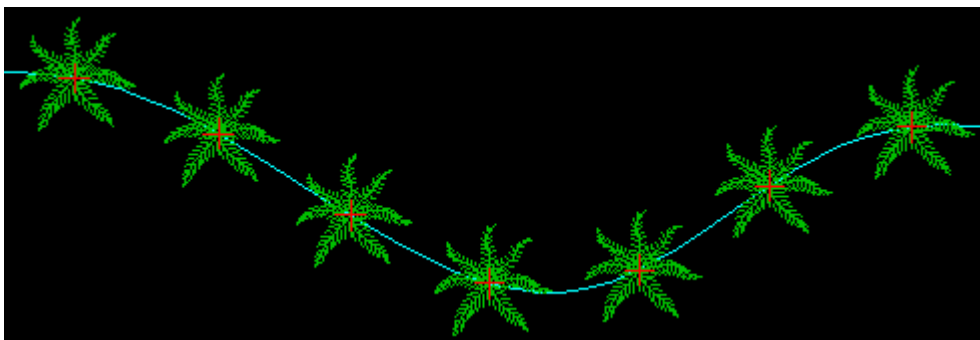


### 12. Node

Toolbar: Object Snap 



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.

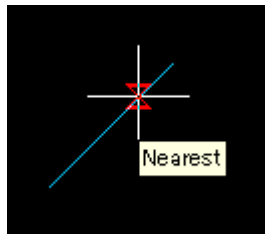


The image as a Spline (cyan). The Measure command has been used to set Points (red) at regular intervals and a tree symbol has been inserted with the Insert command at each Point location using the Node Osnap.

### 13. Nearest

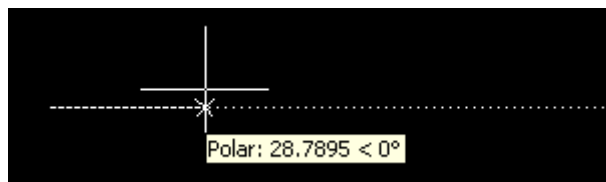
Toolbar: Object Snap 

The Nearest Osnap snaps to the nearest point on a drawing object. This Osnap is useful if you want to make sure that a pick point lies on a drawing object but you don't necessarily mind exactly where it is located.



## 8.5 Polar Tracking

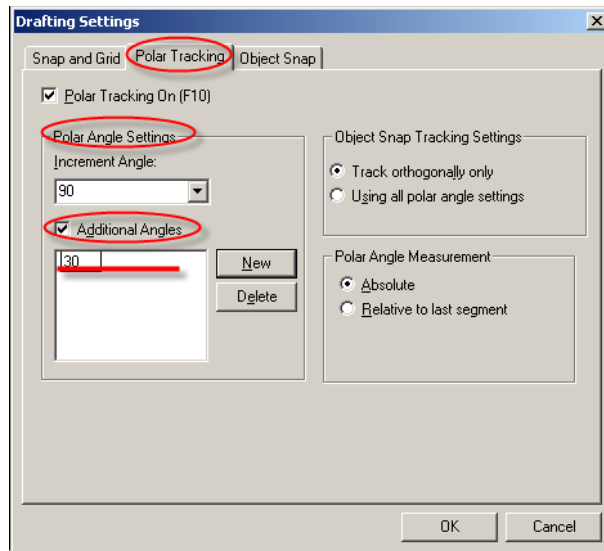
Polar Tracking is a bit like Ortho mode on steroids. Whereas Ortho constrains your lines to either the horizontal or the vertical, Polar Tracking allows you to snap into whatever angles you choose to configure.



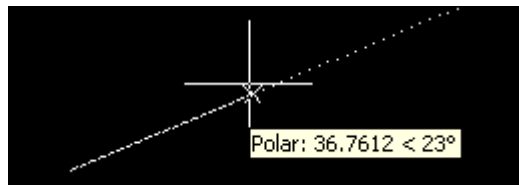
By default, Polar Tracking snaps only to the horizontal and the vertical. When you are drawing a line, pick the first point and then move the cursor close to a vertical or horizontal location and your cursor will snap into place. You will see a polar tracking vector and a tool tip which displays the angle and the distance from the pick point; in other words, a relative polar co-ordinate.

In this default setting, Polar Tracking works like a more flexible version of Ortho but if you look at the Polar Tracking tab on the Drafting Settings dialogue box, you will see just how versatile it can be. Right-click on the POLAR button on the status bar and

choose “Polar Tracking” from the menu.



You can use the Increment angle drop-down list to select one of the preset angle increments. For example, if the increment angle is set to 22.5 degrees, Polar Tracking will snap at 22.5 degree increments starting with zero degrees.



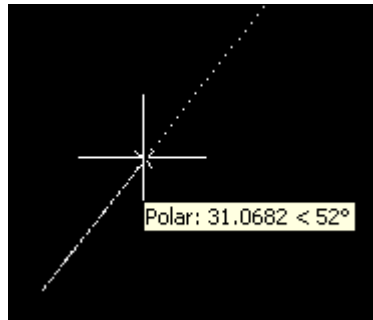
Incidentally, you will notice that the reported angle on the tool tip shown in the illustration on the right is "23", whereas the actual snap angle is 22.5 degrees. This is because angular units are set to display only whole degrees and so, 22.5 is rounded up to 23.

You may sometimes need to snap to specific angles. Say you are working on a drawing of a site and the buildings are orientated in a particular way. If you know the angle, you can use the Additional angles option to add this specific angle so that Polar Tracking will snap to it.

To set additional angles, all you need to do is click on the New button and enter the value in the list. The Delete button can be used to remove unwanted angles. It is also possible to temporarily suspend Additional angles. You can do this by deselecting the Additional angles check box. When you do this, the angles list is greyed out and those

angles won't be used for Polar Tracking until you check the box again.

One of the great benefits of Polar Tracking is that, when used in combination with direct distance entry, you can draw lines of a given length and at a preset angle without using any construction lines and without the need for entering relative co-ordinates. Drawing using this technique can be extremely efficient



## Chapter 9 Entity Attributes

Every object, such as a line or a circle has properties. Some properties such as Color, Linetype and Layer are common to all objects. Some objects have properties which are specific to themselves.

### 9.1 Layers

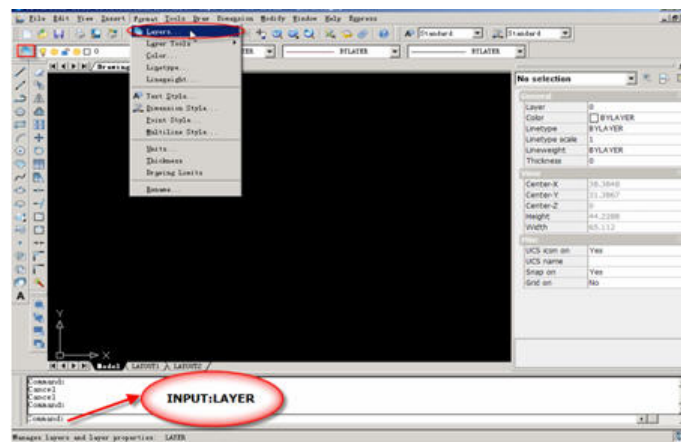
The layer is very important and the correct use of layers can make your drawing much easier to work with. Layers are very powerful because you can have many layers in a single drawing and you can control the visibility, color and linetype of layers independently. This makes you become efficiently to work with complicated drawings. Layers are a effective way to order your drawing.

#### 9.1.1 Creating a New Layer

Pull-down: format>Layer

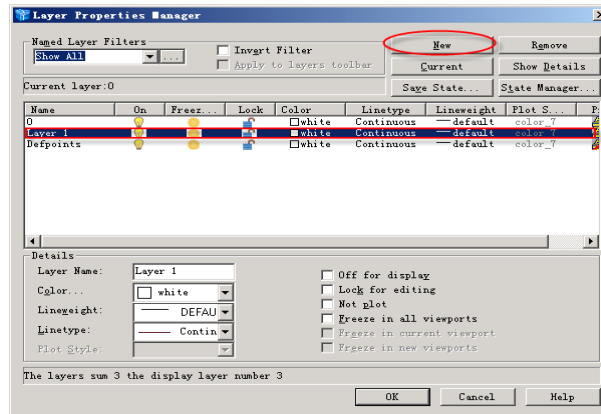
Toolbar 

Command: LAYER



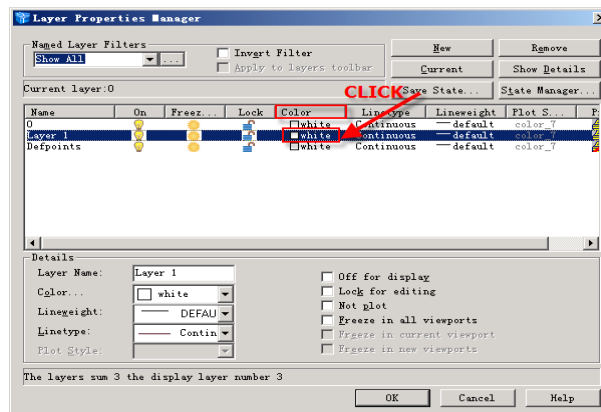
Click on the "New" button. A new layer called "Layer1" is automatically created in the layer list below layer 0. As you can see from the illustration, the layer name is automatically highlighted for you so that you can give the layer a more meaningful name. When you have entered an appropriate name, press the OK to complete the operation. You have now created a new layer and given it a name. Notice that by default it has been assigned the color white and the linetype "Continuous".





### 9.1.2 Set the Color of a Layer

To set a layer color, you can click on the color icon in the layer properties manager with the layer you want to set color



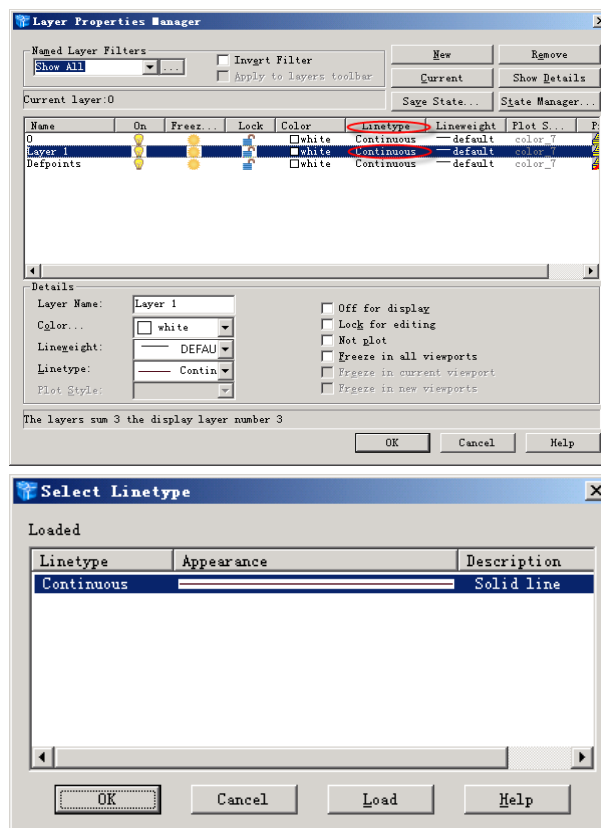
Notice that all layers have their own color icon. Click on the icon and then you will open the Select Color dialogue box. You can select any of the 255 standard colors by picking on the color palette or by entering the color name or number in the text edit box. When you have selected the color you want, click on the "OK" button to set the color.



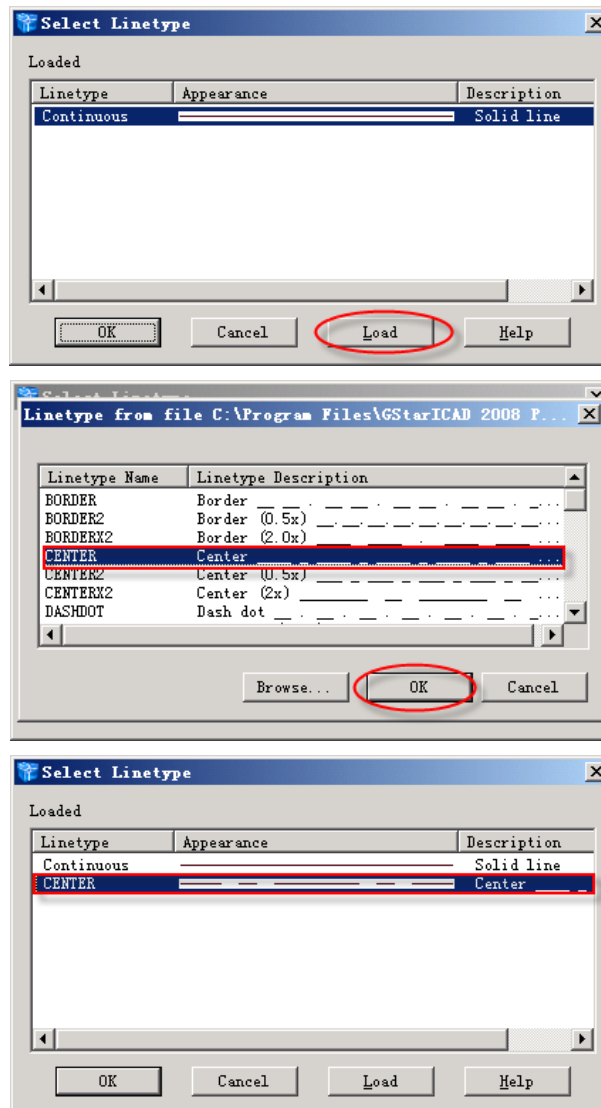
Assigning different colors to your layers will make your work much easier.

### 9.1.3 Set and Load the Linetype of a Layer

In the same way that you can assign a color to a layer you can also assign a linetype to a layer. To set a linetype to a layer, in the layer properties manager, click on the current linetype name associated with your layer in the layer list. By default, layers have the "Continuous" linetype. Click on the linetype name and you will open the Select Linetype dialogue box. You will notice that the "Continuous" linetype is the only one listed. That's because all linetypes, except "Continuous", are stored in an external file and have to be loaded before they can be used.




To load a linetype, click on the "Load..." button in the "Select Linetype" dialogue box. The Load or Reload Linetypes dialogue box appears and displays a list of the available linetypes. Select as many of the listed linetypes as you wish and then click the OK button to return to the Select Linetype dialogue box.








When you return to the Select Linetype dialogue box the loaded linetypes are displayed in the list.

## 9.1.4 Control Layer States

There are a number of other properties all of which relate to whether or not objects on a layer can be seen and/or modified. The current states of these properties are all indicated by icons in the various layer lists. The meaning of these icons is shown in the table below:

ICON	NAME	DESCRIPTION
	On	The layer is visible unless it is also frozen

	Off	The layer is invisible but objects are still regenerated unless it is also frozen
	Thaw in all viewports	The layer is not frozen
	Freeze in all viewports	The layer is invisible and objects are suspended from regeneration
	Unlock	The layer is unlocked
	Lock	The layer is locked and objects cannot be selected or modified

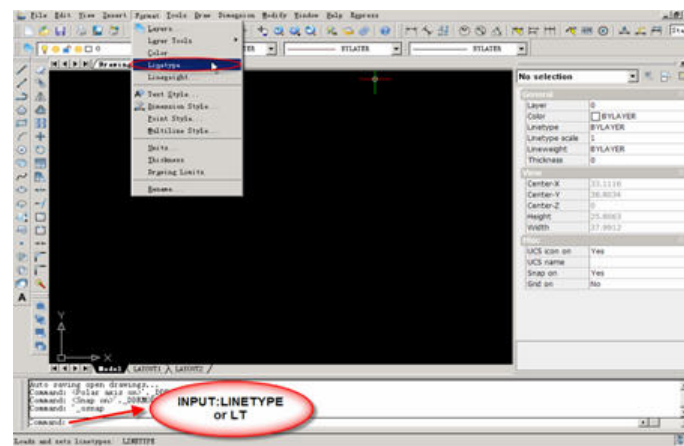
## 9.2 LINETYPE

A linetype is a repeating pattern of dashes, dots, and blank spaces displayed in a line or a curve. You assign linetypes to objects either by layer or by specifying the linetype explicitly, independent of layers. In addition to choosing a linetype, you can set its scale to control the size of the dashes and spaces, and you can create your own custom linetypes.

### 9.2.1 The Linetype Command

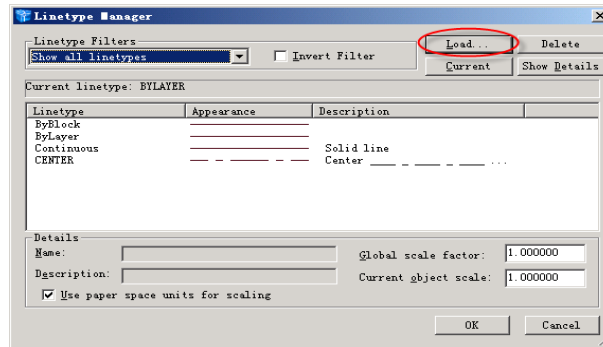
Pull-down menu: Format>Linetype

Commands: LINETYPE or LT

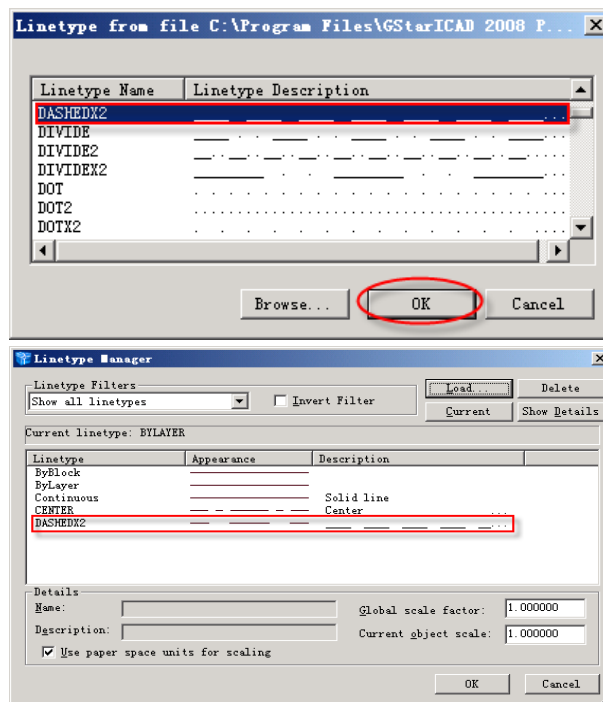


If you have just started a new drawing the only true linetype available will be the "Continuous" linetype. Before you are able to assign any other linetype, you must first load the linetypes you may need. To load linetypes you must use the "Linetype"

command. See illustration below:



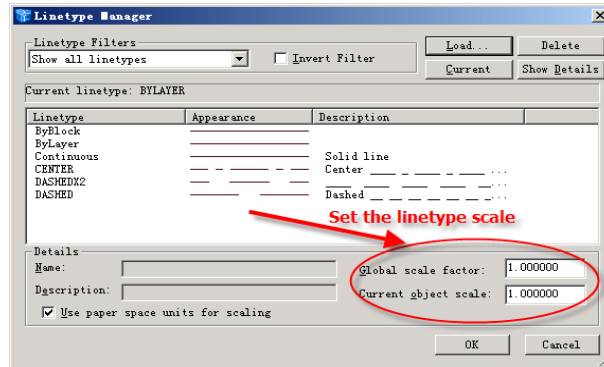
In the “linetype manager” dialogue box, click on the "Load" button, this brings up the Load or Reload Linetypes dialogue box .Select the required linetypes from this dialogue box and then click the "OK" button to return to the “linetype manager” dialogue box, where you will see the newly loaded linetypes in the Linetype list.



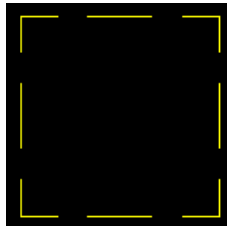
### 9.2.2 Set the Linetype Scale

In many cases your linetypes will display just as you want them. However, it is inevitable that at some time you will need to change the scale at which your linetypes are displayed. By default the linetype scale is set to 1.0, this means that each linetype pattern will repeat every 1.0 drawing units. To make the pattern appear larger, change the scale to a larger number. Setting the linetype scale to 5.0 for example causes the

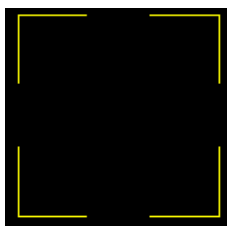
linetype pattern to repeat every 5.0 drawing units so that the pattern will appear 5 times larger. Conversely, setting the scale to 0.2 causes a repetition every 0.2 drawing units which will make the pattern appear 5 times smaller.



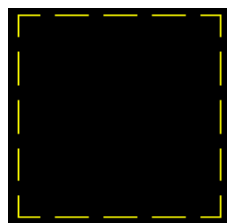
To change the linetype scale, you can set the linetype scale by changing the value in the "Global scale factor" edit box or "current object scale" edit box. Once you have changed the scale factor, click on the OK button to return to your drawing. GStarCAD automatically regenerates the drawing to display all linetypes with the new scale factor.



Linetype scale = 1.0



Linetype scale = 2.



Linetype scale = 0.5

## Chapter 10 Blocks and External reference

Blocks, attributes, and external references provide mechanisms for managing entities in your drawings and for including additional information with the standard drawing entities.

With blocks, you can combine numerous entities into a single entity, and then reuse it, inserting multiple copies.

With attributes, you can associate text, such as part numbers or prices, with blocks, and then extract the text-attribute information to a separate file, such as a database, for further analysis.

With external references, you can link separate reference drawing files to the current drawing to combine information without adding the contents of the reference drawings to a drawing. If you make changes to the referenced file, all references are updated automatically.

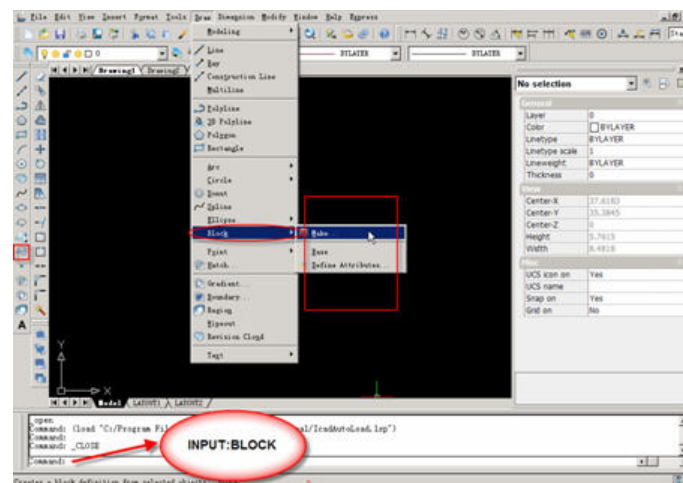
### 10.1 Working with Blocks

#### 10.1.1 Creating Blocks

Pull-down menu: Draw >Block> Make.

Toolbar: 

Command: BLOCK



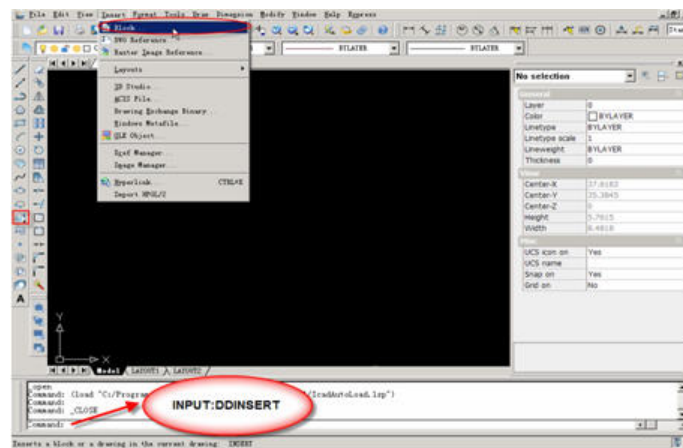
### 10.1.2 Inserting Blocks

When you insert a block, you must specify the insertion point, scale, and rotation angle. The block's insertion point is the reference point specified when you created the block. When you insert a drawing as a block, the program takes the specified insertion point as the block insertion point. You can change the insertion point, however, by first opening the original drawing and redefining the block.

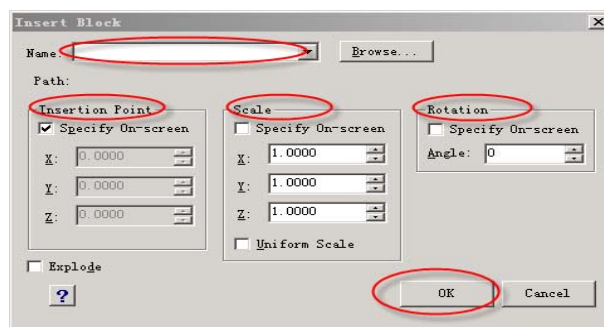
Pull-down menu: Insert > Block.

Toolbar: 

Command: Ddinsert



In the Insert Block dialog box, click Block Name, specify the insertion point for the block, specify the x, y, and z scale factors and the rotation angle, or press Enter to accept the default values.



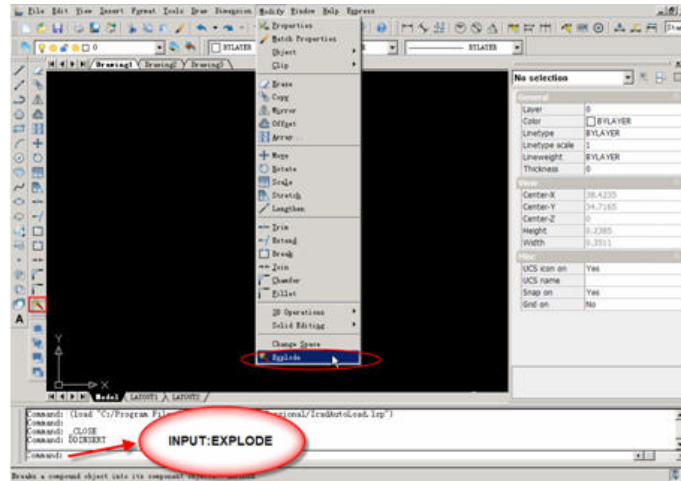
### 10.1.3 Explode Blocks

Pull-down menu: Modify > Explode.



Toolbar: 

Command: EXPLODE



## 10.2 Working with Attributes

An attribute is a particular entity that you can save as part of a block definition. Attributes consist of text-based data. Attributes have either fixed or variable values. When you insert a block containing attributes, the program adds the fixed values to the drawing along with the block, and you are prompted to supply any variable values.

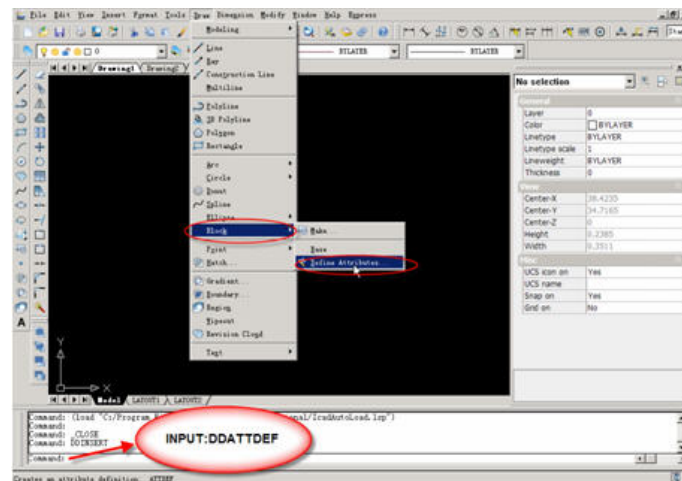
After you insert blocks containing attributes, you can extract the attribute information to a separate file and then use that information in a spreadsheet or database to produce a parts list or bill of materials. You can also use attribute information to track the number of times a particular block is inserted into a drawing.

Attributes can be visible or hidden. Hidden attributes are neither displayed nor printed, but the information is still stored in the drawing and written to a file when you extract it.

### 10.2.1 Defining Attributes

Pull-down menu: Draw >Block>Define attributes

Command: Ddattdef



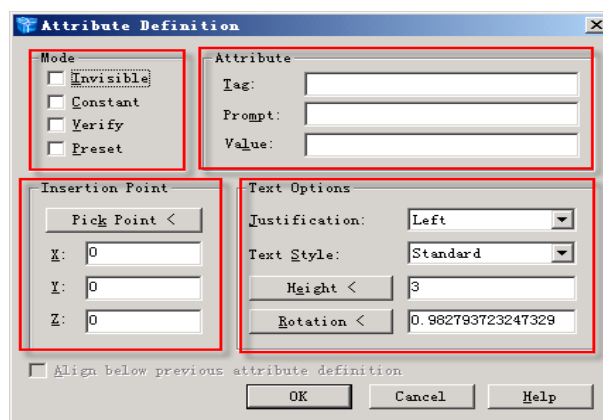
In the “Attribute definition” dialog box,

Under “Attribute” type the tag, prompt, and value.

Under “Insert Point”, specify the location of the attribute, or click Pick Point to select a point in the drawing.

Under “Mode” select the optional attribute modes.

Under “Text Options” specify the text characteristics.



### 10.2.2 Editing Attributes Attached to Blocks

You can edit the attribute values of a block that has been inserted into a drawing.

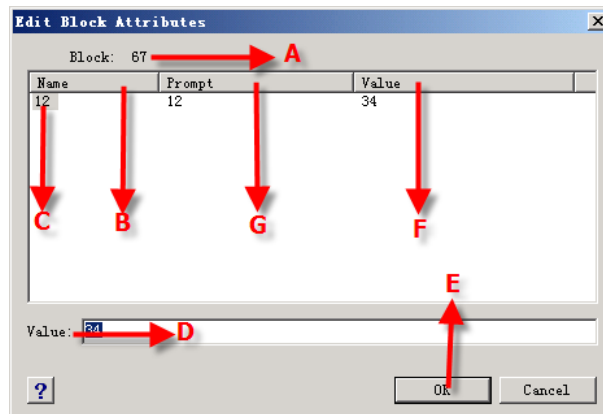
1 Command : Ddatte

2. Select the block to edit.

The Edit Block Attributes dialog box displays all the attributes attached to the block you select.

3. Edit the attribute values as necessary.

4. Click OK.

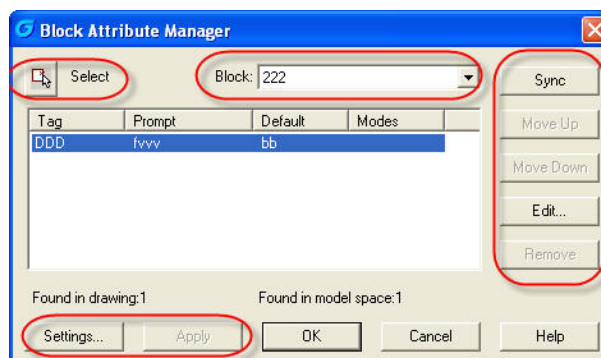


- A. Identifies the name of the block.
- B. Displays the names of all attributes attached to the block.
- C. Click to select attribute.
- D. Type the new value for the selected attribute.
- E. Click to update the attribute values and exit.
- F. Displays the value of each attribute attached to the block.
- G. Displays the prompt for each attribute attached to the block.

### 10.2.3. Block Attribute Manager

Manages the attributes for a selected block definition, preview modification automatically, it is convenient to modify attribute block in quantity.

Command: Battman



#### Select

Allow you to use your pointing device to select a block from the drawing area.

When you choose Select Block, the dialog box closes until you select a block from the drawing or cancel by pressing ESC.

If you modify attributes of a block and then select a new block before you save the attribute changes you made, you are prompted to save the changes before selecting another block.

### **Block**

List all block definitions in the current drawing that have attributes. Select the block whose attributes you want to modify.

### **Sync**

Update all instances of the selected block with the attribute properties currently defined. This does not affect any values assigned to attributes in each block.

### **Move Up**

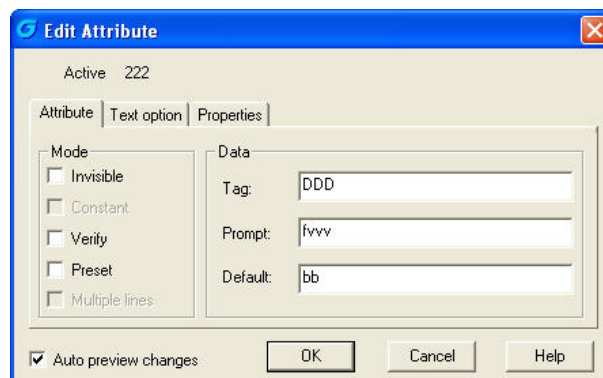
Move the selected attribute tag earlier in the prompt sequence. The Move Up button is not available when a constant attribute is selected.

### **Move Down**

Move the selected attribute tag later in the prompt sequence. The Move Down button is not available when a constant attribute is selected.

### **Edit**

Opens the “Edit Attribute” dialog box, where you can modify attribute properties.



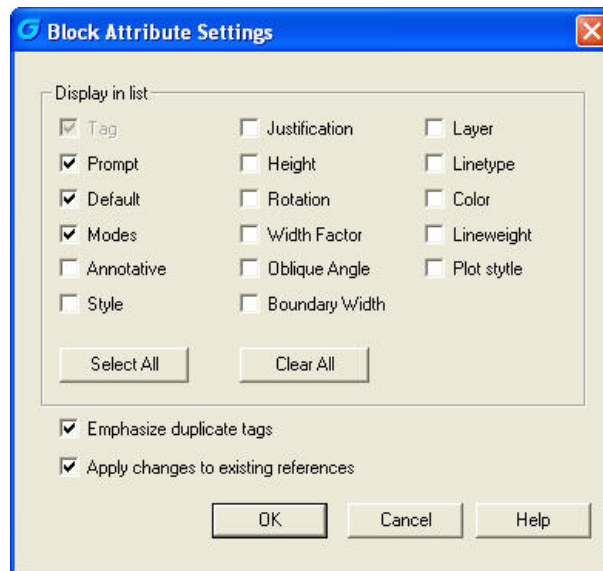
### **Remove**

Remove the selected attribute from the block definition. If Apply Changes to Existing References is selected in the Settings dialog box before you choose Remove,

the attribute is removed from all instances of the block in the current drawing. The Remove button is not available for blocks with only one attribute.

### Settings

Opens the “Block Attribute Settings” dialog box, where you can customize how attribute information is listed in the Block Attribute Manager.



### Apply

Apply the changes you made, but leave the dialog box open.

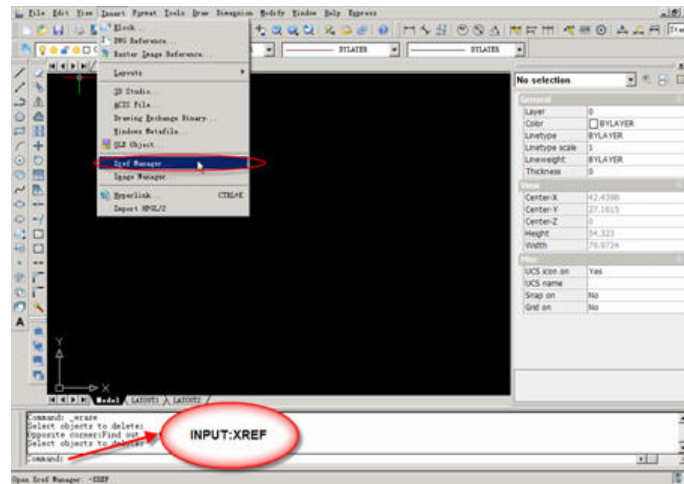
## 10.3 Working with External References

You can link entire drawings to the current drawing as external references. Unlike inserting a drawing as a block, in which you add all the entities from the separate drawing into the current drawing, external references attach a pointer to the external file. The entities in the external reference appear in the current drawing, but the entities themselves are not added to the drawing. Thus, attaching an external reference does not significantly increase the size of the current drawing file.

### 10.3.1 Opening External References

1. Pull-down menu: Insert > Xref Manager.

Command: XREF



2. Select the external reference to open.
3. Click Open.

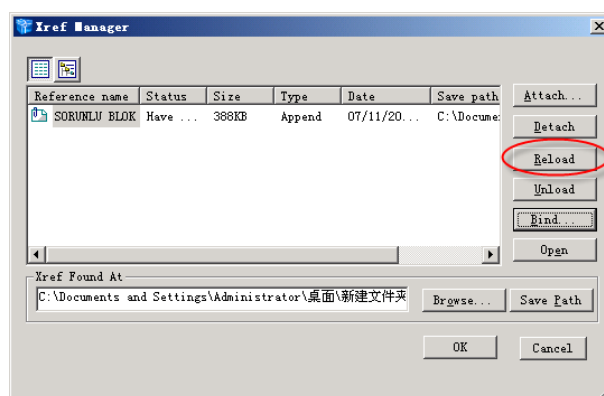
### 10.3.2 Reloading External References

When you open or print a drawing, any external references in the drawing are updated automatically. If a drawing is already open and a referenced drawing is modified, you can update the current drawing manually to display the latest version of the referenced drawing.

You may also want to reload an external reference that has been unloaded temporarily.

#### To reload an external reference

1. Pull-down menu: Insert > Xref Manager.



2. Select the external reference to reload.
3. Click Reload.

### 10.3.3 Removing External References

Removing external references from the current drawing is easy with the Xref Manager. You can unload an external reference, which keeps some information about the external reference in the current drawing for easy reloading later, or you can detach the external reference entirely.

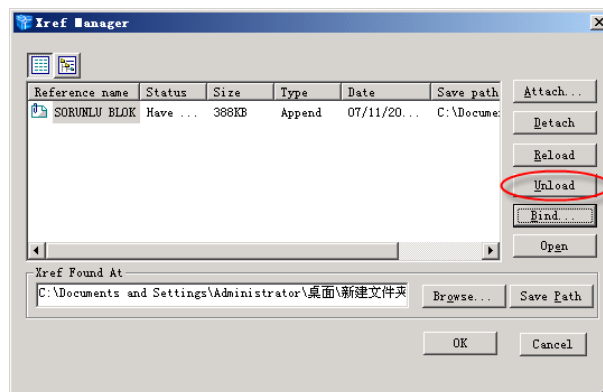
When you unload an external reference, you remove it from the current drawing. However, its elements, such as layers and linetypes, remain in the drawing and it is still listed in the Xref Manager. By detaching an external reference you remove it and all of its elements from the current drawing, and it is no longer listed in the Xref Manager.

#### To unload an external reference

1. Do one of the following:

Pull-down menu: Insert > Xref Manager.

Command: XRM



2. Select the external reference to unload.

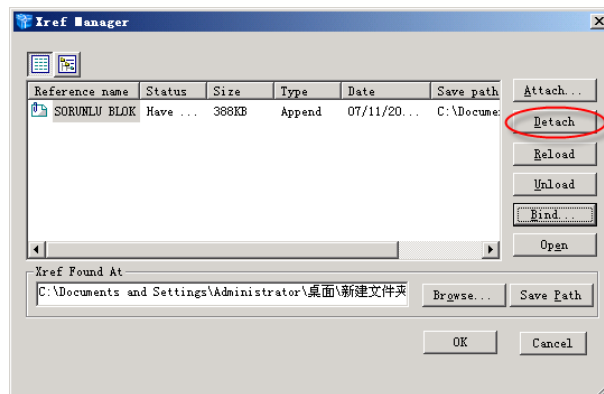
3. Click Unload.

#### To detach an external reference

1. Do one of the following:

Pull-down menu: Insert > Xref Manager.

Command: XRM



2. Select the external reference to detach.
3. Click Detach.

### 10.3.4 Attaching External References

Attaching a separate drawing to the current one creates an external reference. The external reference appears in the drawing as a block definition, but the drawing entities are linked rather than added to the current drawing. If you modify the linked drawing, the current drawing that contains the external reference is updated automatically when you open it, or you can reload the external reference manually so it reflects the latest version of the external reference.

When you attach an external reference, its layers, linetypes, text styles, and other elements are not added to the current drawing. Rather, these elements are also linked from the referenced file.

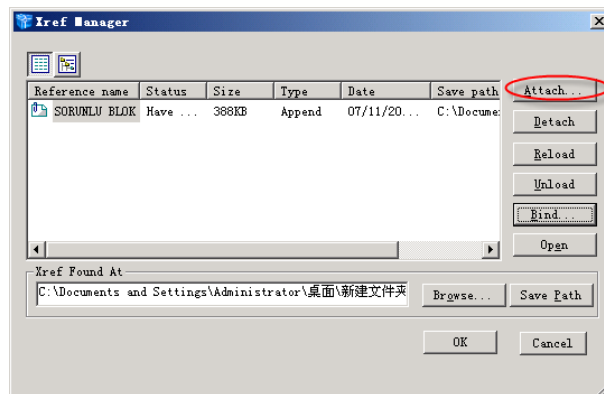
#### To attach external references

1. Do one of the following:

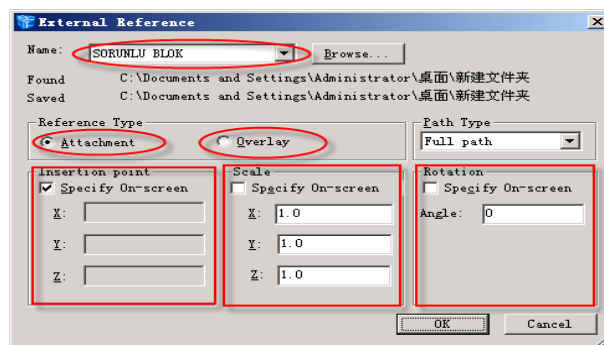
Pull-down menu: Insert > Xref Manager.

Command: XRM





2. Click Attach.



3. Specify the drawing file to attach as an external reference, and then click Open.

4. In Reference Type, choose how you want to insert the drawing:

- Attachment - inserts a copy of the drawing and includes any other drawings that are externally referenced within the referenced drawing.
- Overlay - lays a copy of a drawing over your original drawing; it does not include any nested external references in the externally referenced drawing.

5. Make any additional selections.

6. Click OK.

7. If you marked Specify On-Screen for any items, follow the prompts to attach the external reference.

### 10.3.5 Binding External References to Drawings

External references are not part of the drawing. Rather, they are links to an externally referenced file. To provide a copy of a drawing containing external references to someone else, you must also provide all the external reference files. In addition, the

person receiving the drawings must either re-create the same paths you used when linking the external references or change the paths for the external references.

To provide a copy of a drawing that contains external references, it is often easier to first bind the external references to the drawing. Binding the external references makes them a permanent part of the drawing, which is similar to inserting a separate drawing as a block.

You can bind external references that are attached directly to the current drawing; you cannot bind nested external references.

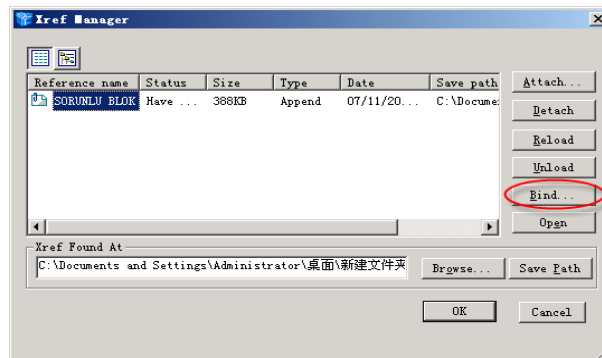
### To bind an existing external reference to a drawing

#### 1. Do one of the following:

Pull-down menu: Insert > Xref Manager.

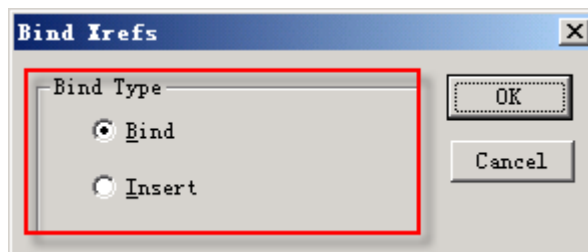
Command: XRM

#### 2. Select the external reference to bind.



#### 3. Click Bind.

#### 4. Choose one of the following:



### The type of binds

- Bind Binds: the external reference, and creates a unique name for each named entity, such as a layer or block, that is located in the external reference.

- Insert Binds: the external reference, but does not change the names of any named entities in the external reference.

### 10.3.6 Clipping External References

When you attach a drawing as an external reference, all of the referenced drawing displays in the current drawing. However, after you attach an external reference, you can define a clipping boundary that determines which portion of the referenced drawing is visible or hidden.

You can edit, move, or copy clipped external references as the same way you modify unclipped external references. The boundary moves with the reference. If an external reference contains nested clipped external references, they also appear clipped in the drawing.

#### 10.3.6.1 Adding Clipping Boundaries

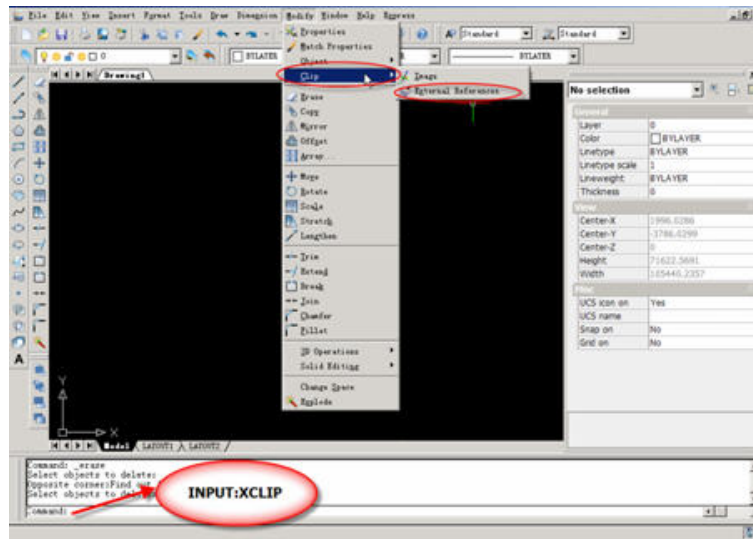
When you create a clipping boundary, it affects only the display of the referenced drawing; it does not affect the original referenced drawing or any referenced geometry. The portion of the external reference within the clipping boundary is visible and the remainder of the external reference becomes hidden.

#### To define a rectangular clipping boundary

1. Do one of the following:

Pull-down menu: Modify > Clips.

Command: Xclip



2. Select the external references to clip. If desired, you can also select blocks.
3. Press Enter.
4. Press Enter to create a new clipping boundary.
5. If prompted, press Enter to delete any existing boundaries.
6. Choose Rectangular.
7. Define the first corner of the clipping rectangle.
8. Define the second corner of the clipping rectangle.

The selected external references are clipped by the rectangle.

### 10.3.6.2 Turning Clipping Boundaries On and Off

You can turn xref clipping on or off. When a clipping boundary is turned off, the boundary does not display and the entire external reference is visible, provided that the geometry is on a layer that is on and thawed. When a clipping boundary is turned off, it still exists and can be turned on. However, deleting a clipping boundary is permanent.

#### To turn clipping boundaries on and off

1. Do one of the following:

Pull-down menu: Modify > Clips.

Command: Xclip

2. Select the desired external references.
3. Press Enter.
4. To turn off clipping boundaries, choose Off. To turn on existing clipping boundaries,

choose On.

5. Press Enter.

If you are turning off a clipping boundary, click the clipped portion of the external reference to view the previously hidden portion of the referenced drawing.

### **10.3.6.3 Deleting Clipping Boundaries**

If you no longer need a clipping boundary for an external reference, you can delete it.

#### **To delete a clipping boundary**

1. Do one of the following:

Pull-down menu: Modify > Clips.

Command: Xclip

2. Select the desired external references.

3. Press Enter.

4. Choose Delete, and then press Enter.

5. Click the clipped portion of the external reference.

The previously hidden portion of the referenced drawing displays.

## Chapter 11 Drawing in Three Dimensions

GStarCAD2009 supports the following types of three-dimensional models:

- Wire-frame models, which consist of lines and curves that define the edges of a three-dimensional entity.
- Surface models, which consist of both edges and the surfaces between those edges.
- 3D solids, which are three-dimensional ACIS entities that consist of faces and edges.

### 11.1 Creating Ruled Surfaces

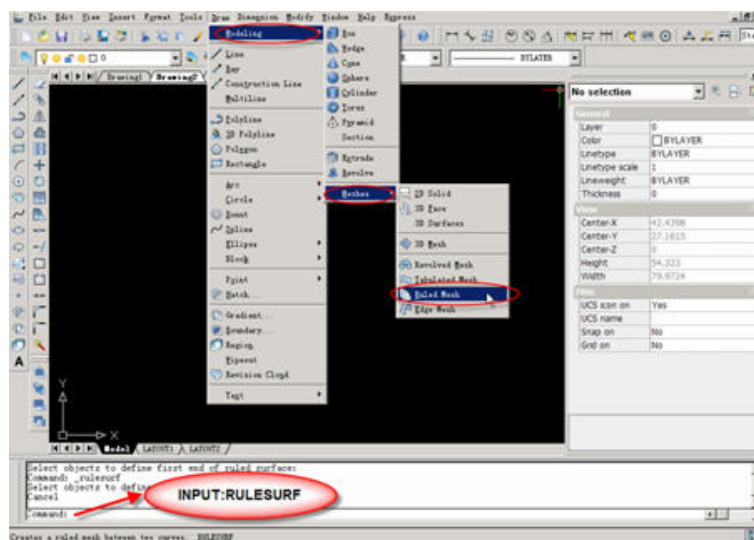
You can create a ruled surface, which is a three-dimensional body that approximates the surface between two existing entities. You select the two entities that define the ruled surface. These entities can be arcs, circles, lines, points, or polylines.

**To create a ruled surface**

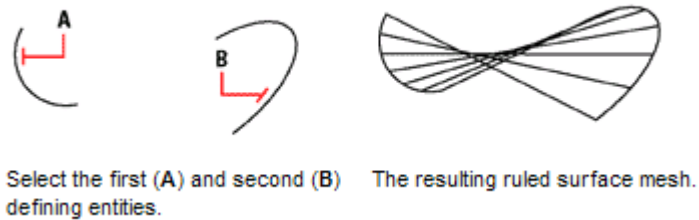
1. Advanced experience level
2. Do one of the following:

Pull-down menu: DRAW> MODELING> MESH> Ruled Mesh.

Command: RULESURF



3. Select the first defining entity.
4. Select the second defining entity.



### 11.2 Creating Edge-defined Coons Surfaces

You can create a surface called a Coons surface, a region 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. You can select the edges in any order.

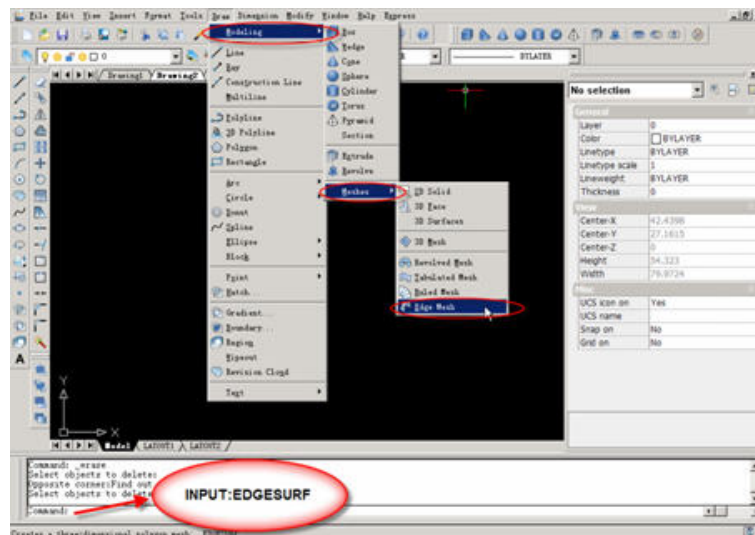
**To create an edge-defined Coons surface**

1. Advanced experience level

2. Do one of the following:

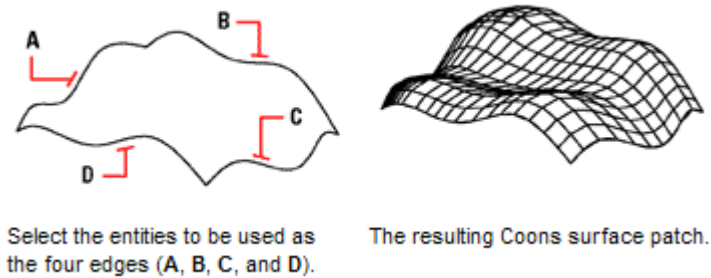
Pull-down menu: DRAW> MODELING> MESH>Edge Mesh

Command: EDGESURF



3. Select the first edge.

4. Select the second, third, and fourth edges.

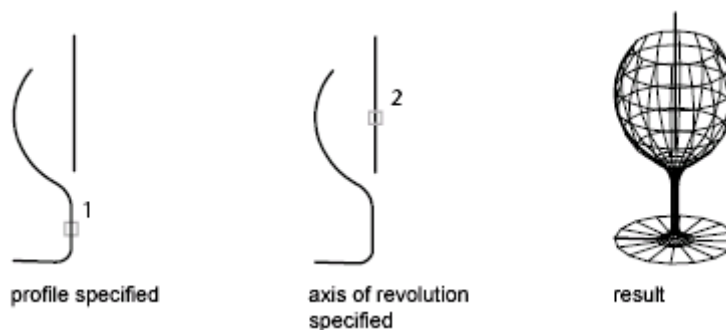


### 11.3. Create a Revolved Mesh

Use the REVSURF command to create a revolved mesh by rotating a profile of the object about an axis. REVSURF is useful for mesh forms with rotational symmetry.

Pull-down menu: DRAW> MODELING> MESH> Revolved Mesh.

Command: REVSURF



The profile is called a path curve, which can be any combination of lines, circles, arcs, ellipses, elliptical arcs, polylines, or splines, closed polylines, polygons, closed splines, or donuts.

### 11.4. Create a Tabulated Mesh

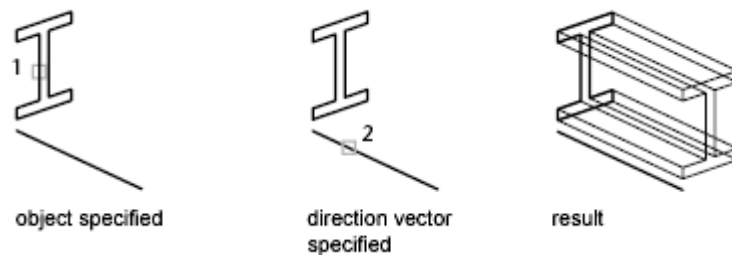
With the TABSURF command, you can create a mesh representing a general tabulated surface defined by a path curve and a direction vector. The path curve can be a line, arc, circle, ellipse, elliptical arc, 2D polyline, 3D polyline, or spline. The direction vector can be a line or an open 2D or 3D polyline. TABSURF creates the mesh as a series of



parallel polygons running along a specified path. You must have the original object and the direction vector already drawn, as shown in the following illustrations.

Pull-down menu: DRAW> MODELING> MESH> Tabulated Mesh.

Command: TABSURF




### 11.5 Creating Extruded Surfaces

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.

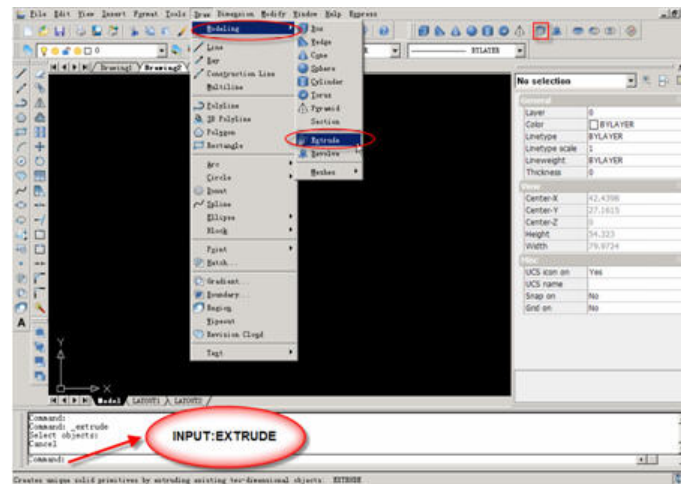
#### To create an extruded surface

1. Advanced experience level
2. Do one of the following:

Pull-down menu: DRAW> MODELING> Extrude

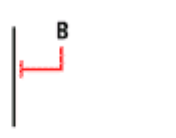
Toolbar: 

Command: EXTRUDE.



3. Select the entity to extrude.

4. Select the extrusion path.



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



The resulting extruded surface mesh.

## 11.6 Creating Revolved Surfaces

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.

Revolving the profile 360 degrees creates a closed body.

### To create a revolved surface

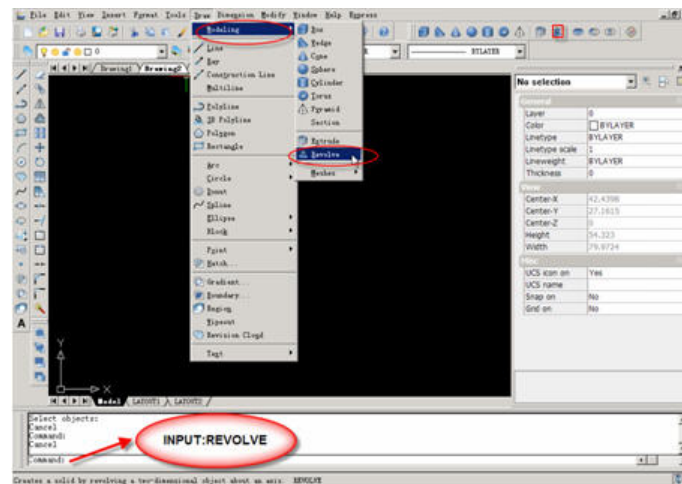
1. Advanced experience level

2. Do one of the following:

Pull-down menu: DRAW > MODELING> Revolve

● Toolbar: 

● Command: REVOLVE



3. Select the entity to revolve.
4. Select the entity to be used as the axis of revolution.
5. Specify the starting angle.
6. Specify the number of degrees to revolve the entity.

### 11.7 Creating Boxes

You can create a rectangular box, or cube. A box consists of six rectangular surface planes. The base of the box is always parallel to 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.

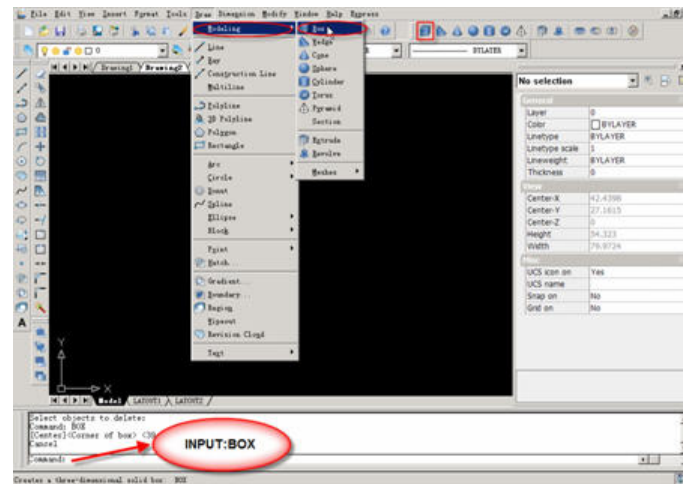
#### To create a box

1. Do one of the following:

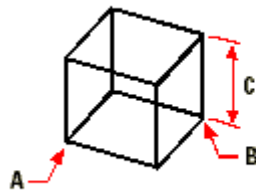
Pull-down menu: DRAW > MODELING > Box.

Toolbar: 

Command: BOX



2. Specify the first corner of the base.
3. Specify the opposite corner of the base.
4. Specify the height.



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

### 11.8 Creating Wedges

You can create a three-dimensional wedge consisting of five surface planes. The base of the wedge is always parallel to the xy plane of the current UCS with the sloped face opposite the first corner. The height is always parallel to the z-axis. You position the wedge by specifying either a corner or the center of the wedge. You determine the size of the wedge by either specifying a second corner and the height; defining the wedge based on a cube having a given length; or specifying the length, width, and height.

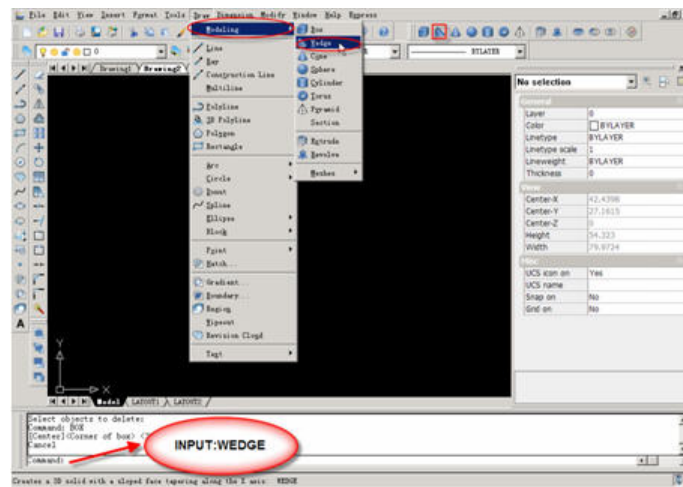
#### To create a wedge

1. Do one of the following:

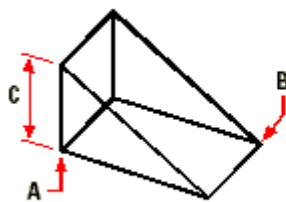
Pull-down menu: DRAW > MODELING > Wedge.

Toolbar: 

Command: WEDGE



2. Specify the first corner of the base.
3. Specify the opposite corner of the base.
4. Specify the height.



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

## 11.9 Creating Cones

You can create a three-dimensional cone defined by a circular base and tapering to a point perpendicular to the base. The base of the cone is always parallel to the xy plane of the current UCS; the height of the cone is always parallel to the z-axis. You position the cone by specifying the center of the base. You determine the size of the cone by specifying either the radius or the diameter of the base and the height.

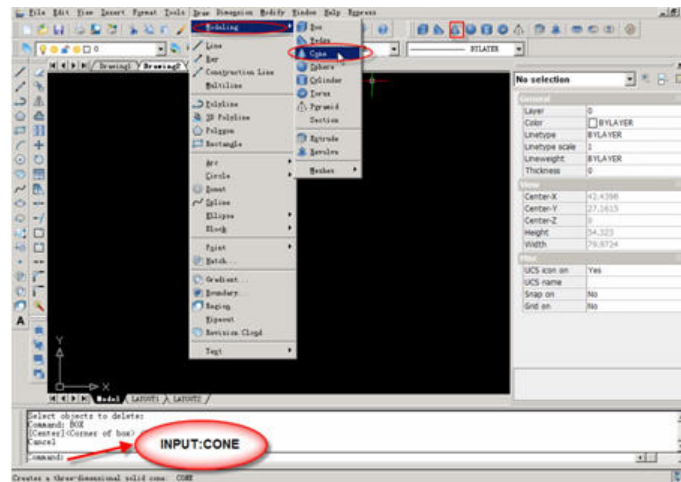
### To create a cone

1. Do one of the following:

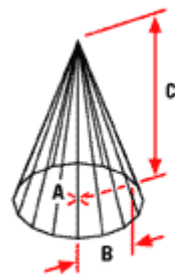
Pull-down menu: DRAW > MODELING > Cone.

Toolbar: 

Command: CONE



2. Specify the center of the base of the cone.
3. Specify the radius or diameter.
4. Specify the height.




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

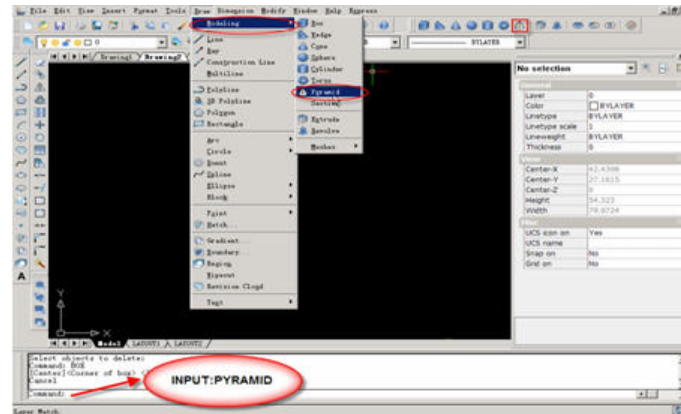
### 11.10 Creating Pyramids

You can create a tetrahedron (three-sided pyramid) or a four-sided pyramid. 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 to the xy plane of the current UCS. You position the pyramid by specifying a corner of the base. You determine the size of the pyramid by specifying the base points and either the apex, the corners of the top surface, or the endpoints of the ridge.

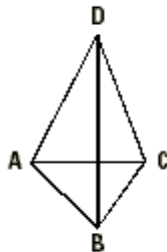
**To create a tetrahedron**

1. Do one of the following:

- Pull-down menu: DRAW > MODELING > Pyramid.
- Toolbar: 
- Command: PYRAMID



2. Specify the first point for the base of the pyramid.
3. Specify the second and third points.
4. In the prompt box, choose Tetrahedron.
5. Specify the apex of the tetrahedron.



The first point (A), second point (B), and third point (C) of the base, and the apex (D).

### 11.11 Creating Cylinders

You can create a cylinder defined by a circular base. The base of a cylinder is always parallel to the xy plane of the current UCS; the height of a cylinder is always parallel to the z-axis. You position a cylinder by specifying the center of the base. You determine the size of a cylinder by specifying either the radius or diameter of the base and the height.

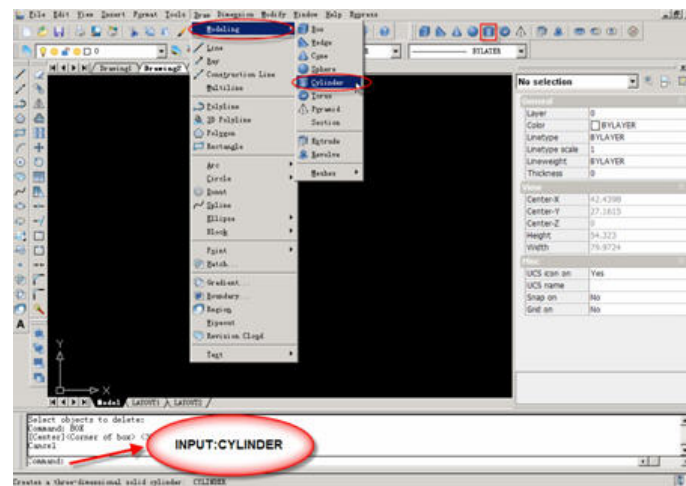
### Create a cylinder

1. Do one of the following:

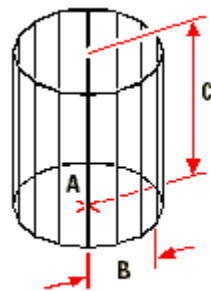
Pull-down menu: DRAW > MODELING >Cylinder.

Toolbar: .

Command: CYLINDER



2. Specify the center of the base of the cylinder.
3. Specify the radius or diameter.
4. Specify the height.



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

### 11.12 Creating Spheres

You can create a sphere. The latitude lines of a sphere are always parallel to the xy plane of the current UCS; the central axis is always parallel to the z-axis. You position a sphere by specifying its center point. You determine the size of a sphere by specifying either its radius or its diameter.



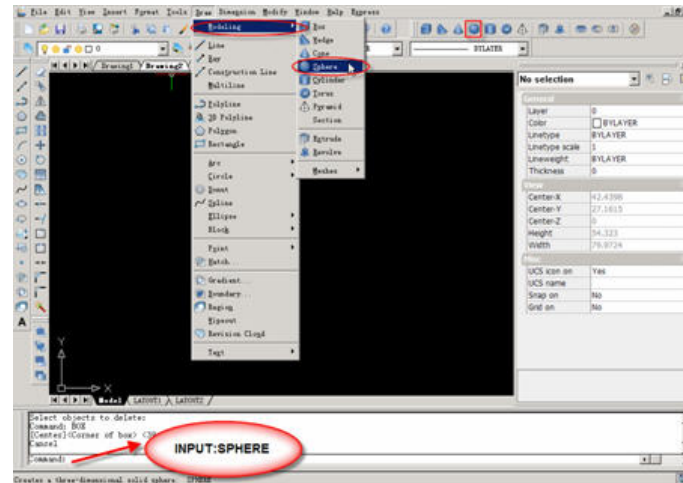
### To create a sphere

1. Do one of the following:

Pull-down menu: DRAW > MODELING > Sphere.

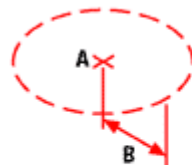
Toolbar: 

Command: SPHERE

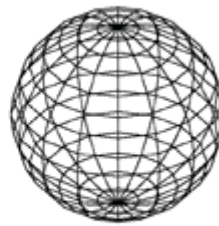


2. Specify the center of the sphere.

3. Specify the radius or diameter.



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



The resulting sphere.

### 11.13 Creating Torus


You can create a three-dimensional donut or ring-shaped entity known as a torus. The diameter of a ring is always parallel to 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 to the z-axis of the current UCS. You position a torus by specifying its center

point. You determine the size of a torus by specifying its overall diameter or radius and the diameter or radius of the tube (the circle being revolved).

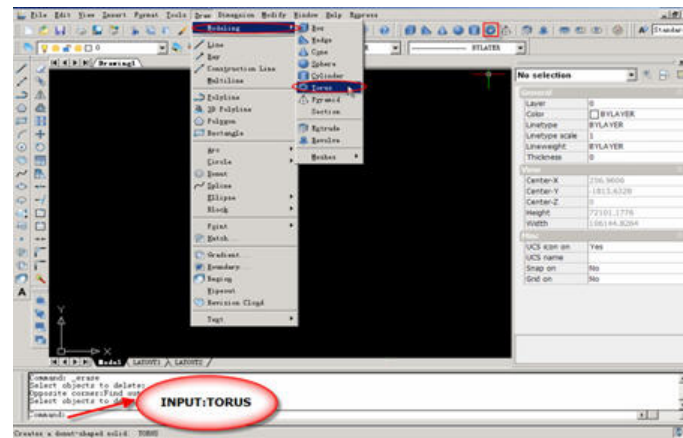
### To create a torus

1. Do one of the following:

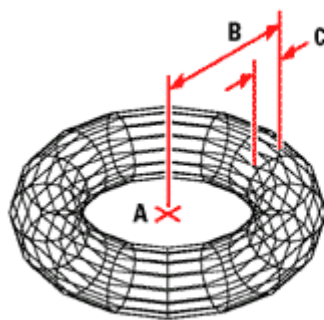
Pull-down menu: DRAW > MODELING > Torus.

Toolbar: 

Command: TORUS



2. Specify the center of the whole torus.
3. Specify the radius or diameter of the whole torus.
4. Specify the radius or diameter of the body of the torus.



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

## 11.12 Editing in Three Dimensions

You can copy, move, rotate, array, mirror, and align two-dimensional and three-dimensional entities in both two-dimensional and three-dimensional space. You

can also change and edit properties of three-dimensional entities much like you change and edit properties for two-dimensional entities. When you modify three-dimensional entities in two-dimensional space, you modify the entity in relation to the current UCS.

### 11.12.1 Rotating in Three Dimensions

You can rotate selected entities about a specified axis in three-dimensional space. You select the entities to rotate and then define the axis of rotation either by specifying two points; selecting an existing entity; aligning the axis with the x-, y-, or z-axis of the current UCS; or aligning the axis with the current view.

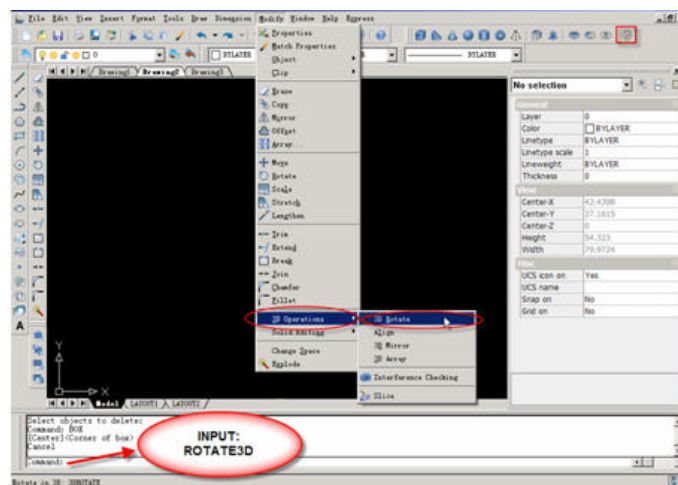
#### To rotate an entity about an axis in three dimensions

1. Do one of the following:

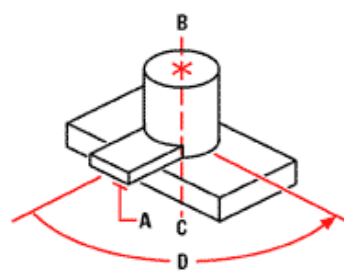
Pull-down menu: Modify >3D Operation >3D Rotate.

Toolbar: 

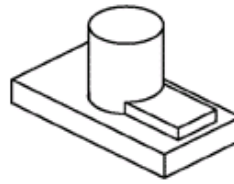
Command: ROTATE3D



2. Select the entities to rotate, and then press Enter.
3. Choose from one of the following options: Entity, Last, View, Xaxis, Yaxis, Zaxis.
4. Specify the rotation angle.
5. Specify the reference angle.



Select the entities to rotate (A), specify the endpoints of the axis of rotation (B and C), and then specify the rotation angle (D).



Result after rotating the entities.

### 11.12.2 Arraying in Three Dimensions

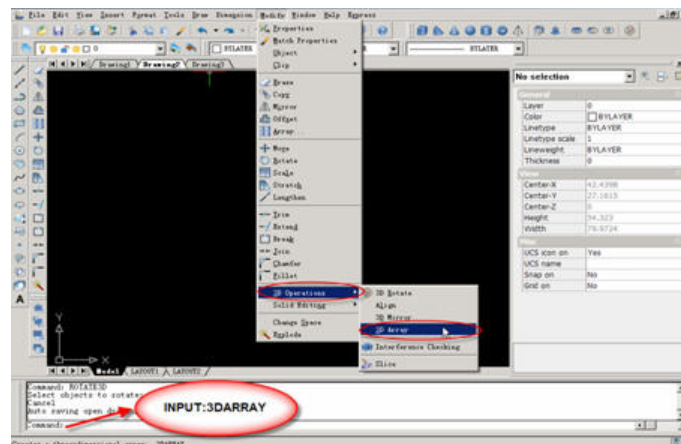
You can copy selected entities and arrange them in a three-dimensional rectangular or polar (circular) pattern. For a rectangular array, you control the number of copies in the array by specifying the number of rows and columns and the number of levels. You also specify the distance between each. For a polar array, you specify the axis around which to array the entities, the number of copies of the entities to create, and the angle subtended by the resulting array.

#### To create a three-dimensional rectangular array

1. Do one of the following:

Pull-down menu: Modify>3D Operation > 3D Array.

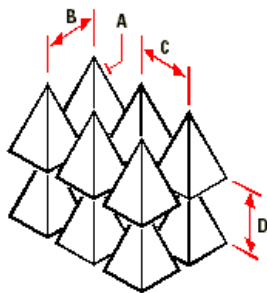
Command: *3DARRAY*



2. Select the entities, and then press Enter.

3. In the prompt box, choose Rectangular.

4. Type the number of rows in the array.
5. Type the number of columns.
6. Type the number of levels.
7. Specify the vertical distance between the rows.
8. Specify the horizontal distance between the columns.
9. Specify the depth between the levels.



To create a three-dimensional rectangular array, select the entity to copy (A), type the number of rows, columns, and levels, and then specify the distance between each row (B), column (C), and level (D).

### 11.12.3 Mirroring in Three Dimensions

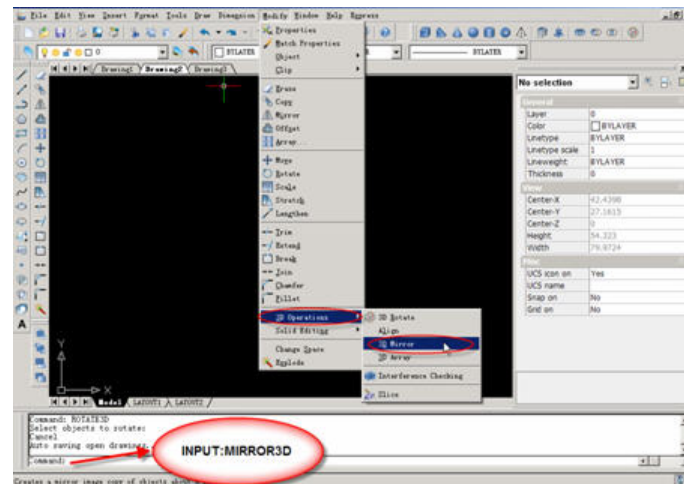
You can create a mirror image of selected entities in three-dimensional space. You mirror the entities about a mirror plane that you define by either specifying three points; selecting an existing two-dimensional planar entity; aligning the plane parallel to the xy, yz, or xz plane of the current UCS; or aligning the plane with the current view. You can delete or retain the original entities.

#### To mirror an entity about a three-dimensional plane

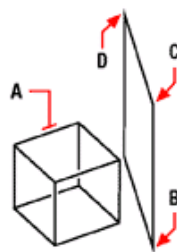
1. Do one of the following:

Pull-down menu: Modify>3D Operation > 3D Mirror.

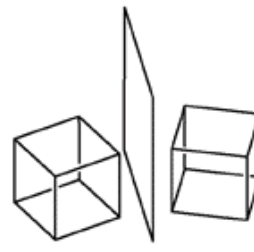
Command: MIRROR3D



1. Select the entities, and then press Enter.
2. In the prompt box, choose 3 Points, or press Enter to select the default.
3. Specify the first point on the mirror plane.
4. Specify the second and third points on the plane.
5. In the prompt box, choose one of the following:
  - Yes-Delete Entities to delete the original entities.
  - No-Keep Entities to retain the original entities.



Select the entity to mirror (A), and then specify the first point (B), second point (C), and third point (D) defining the mirror plane.



The resulting mirrored entity.

### 11.12.4 Aligning in Three Dimensions

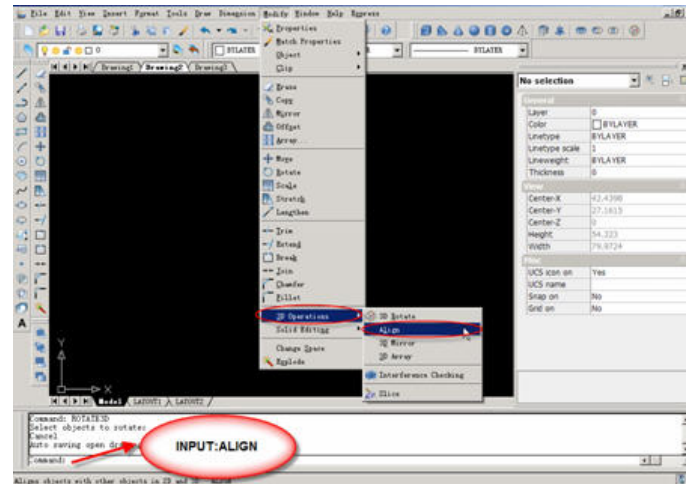
You can align selected entities with other entities in three-dimensional space. You select the entities you wish to align, and specify one, two, or three pairs of points to align the selected entities.

**To align an entity an entity with another**

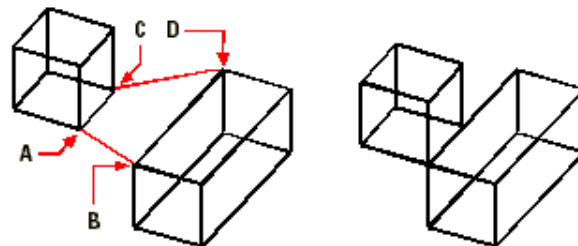
1. Do one of the following:

Pull-down menu: Modify>3D Operation >Align.

Command: ALIGN.



1. Select the entities, and then press Enter.
2. Specify the first source point.
3. Specify the first destination point.
4. Specify additional source and destination points if desired (up to three pairs).



Select the entities to align, and then specify the first source point (A), the first destination point (B), the second destination point (C), and the second destination point (D). You can specify up to three pairs of source/destination points.

The resulting aligned entity.

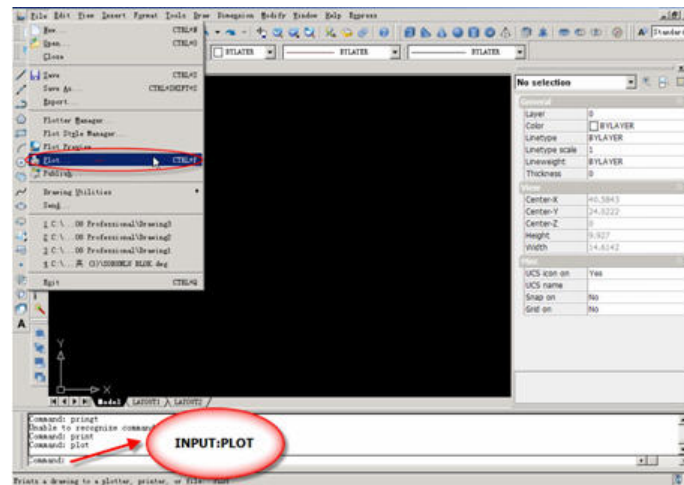
## Chapter 12 Print Drawings

### 12.1 Print Introduction and Interface

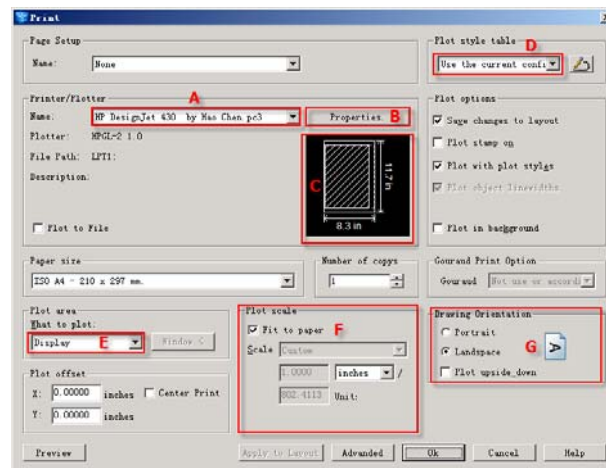
Toolbar: 

Pull-down menu: File>Plot

Command: Print



The print interface of GStarCAD2009 is similar to AutoCAD basically.



A: select a printer or plotter

B: Set print driver

C: Partial preview

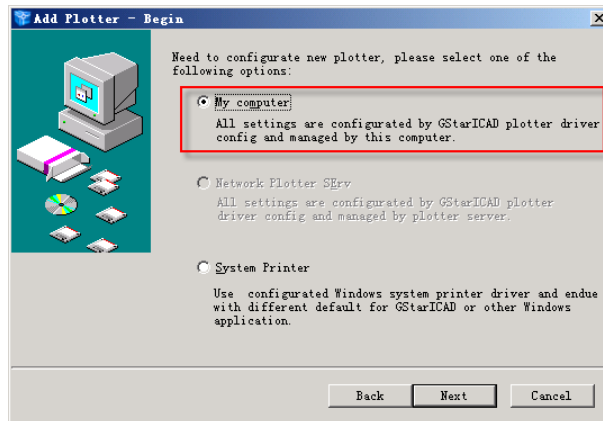
D: Choose and set plot style

E: Specify the area to plot

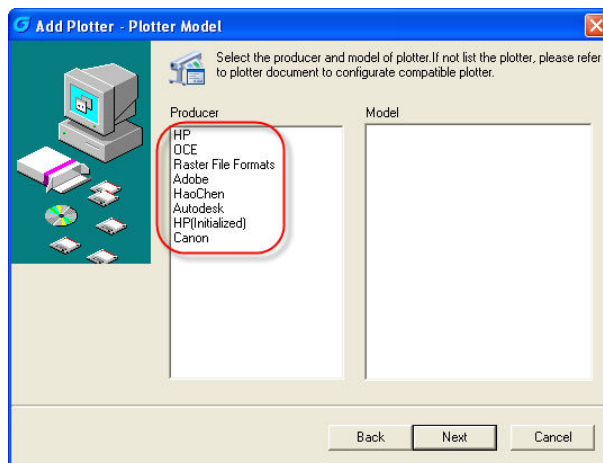




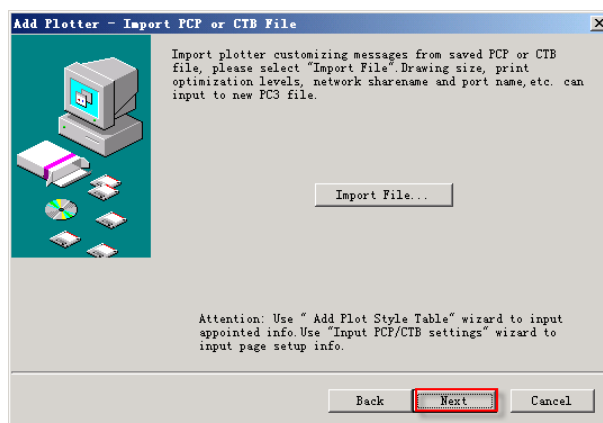
Click “next” then choose “my computer”



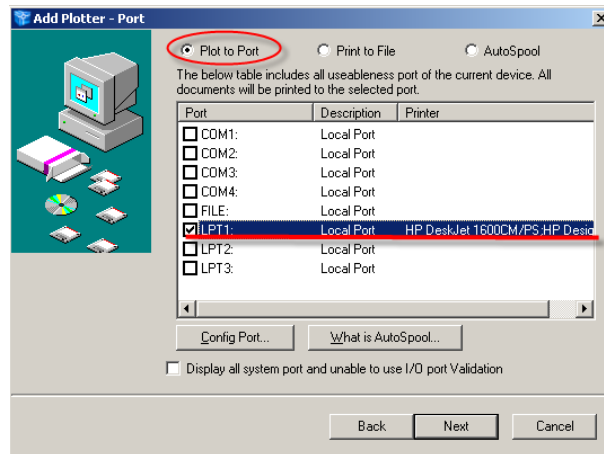
Click “next”, choose a producer from the left table then choose a model from the right table.



Click “next” you will see:



Click “next” then choose “plot to port” or “print to file” according to your need. Many plotter choose to use LPT1 port currently.



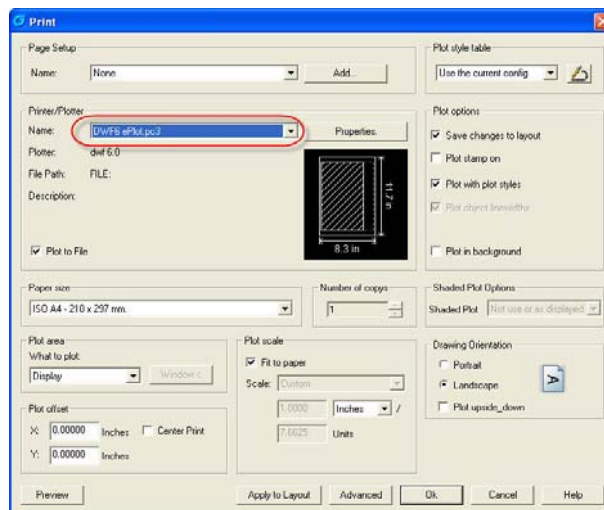
You can finish the install work according to the prompts.

### 12.3. Add DWF virtual print driver

It can realize the web publish function, different from the direct DWF export before, this published file contains all the DWF information and users can export DWF file according to selected area

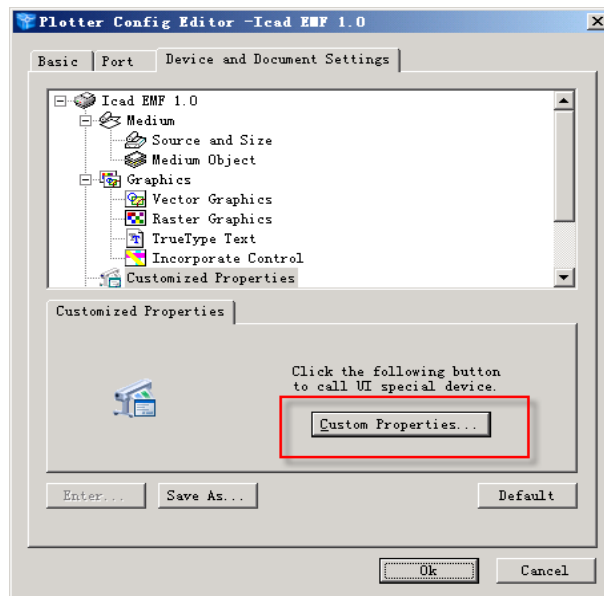
#### The process of creating DWF file

Select “DWF6 ePlot.pc3” printer, other steps are same to the normal print.

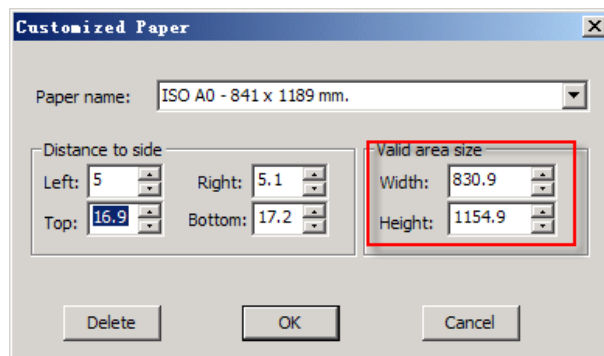
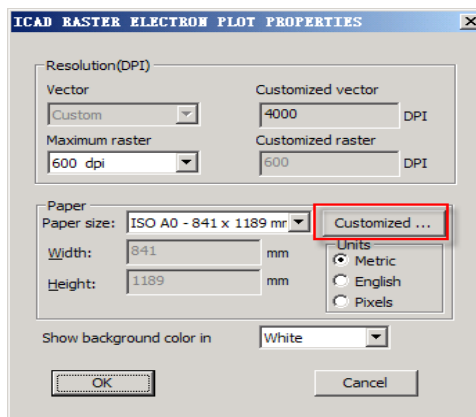


### 12.4 Set the Print Driver

Click “properties” then you can enter into the interface of print driver setup.



Click “Custom Properties” then you can set print driver and paper.



If you use GStarCAD2009 built-in print driver you can set the valid area size and the margin; but if you use the system print driver you can not do this.

### 12.5 Specify the Area to Plot

When plotting a drawing, you must specify the area of the drawing to plot. The Plot

dialog box provides the following options under Plot Area.

- **Limits:** When plotting a layout, plot everything within the printable area of the specified paper size, with the origin calculated from 0, 0 in the layout. When plotting the Model tab, plot the entire drawing area defined by the grid limits. If the current viewport does not display a plan view, this option has the same effect as the Extents option.
- **Display:** Plot the view in the current viewport in the Model tab or the current paper space view in a layout tab.
- **Window:** Plot any portion of the drawing you specify. Click the Window button to use a pointing device to specify opposite corners of the area to be plotted, or enter coordinate values.

The settings in the “Plot Offset” area of the Plot dialog box specify an offset of the plot area relative to the lower-left corner (the origin) of the printable area or the edge of the paper, depending on the setting made in the Specify Plot Offset Relative To option (Options dialog box, Plot and Publish tab). The Plot Offset area of the Plot dialog box displays the specified plot offset option in parentheses.

You can offset the drawing on the paper by entering positive or negative values in the X and Y offset boxes. However, this may result in the plot area being clipped.

## 12.6 Set Plot Scale and Set Drawing Orientation

### Set Plot Scale

When you specify a scale to output your drawing, you can choose from a list of real-world scales, enter your own scale, or select Fit to Paper to scale the drawing to fit onto the selected paper size.

Usually, you draw objects at their actual size. That is, you decide how to interpret the size of a unit (an inch, a millimeter, a meter) and draw on a 1:1 scale. For example, if your unit of measurement is millimeters, then every unit in your drawing represents a millimeter. When you plot the drawing, you either specify a precise scale or fit the image to the paper.

Most final drawings are plotted at a precise scale. The method used to set the plot scale depends on whether you plot the Model tab or a layout:

- On the Model tab, you can establish the scale in the Plot dialog box. This scale represents a ratio of plotted units to the world-size units you used to draw the model.
- In a layout, you work with two scales. The first affects the overall layout of the drawing, which usually is scaled 1:1, based on the paper size. The second is the scale of the model itself, which is displayed in layout viewports. The scale in each of these viewports represents a ratio of the paper size to the size of the model in the viewport.

### Set Drawing Orientation

The drawing orientation determines whether the position of the plotted drawing is landscape (the longer edge of the drawing is horizontal) or portrait (the longer edge of the drawing is vertical). This is based on the size of paper selected. You can also choose to plot upside down.

## 12.7 Choose Plot Style

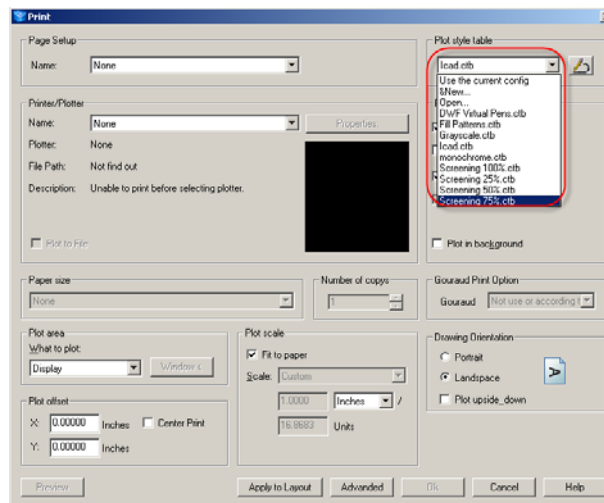
A plot style controls how an object or layer is plotted by determining plotted properties such as lineweight, color, and fill style. Plot style tables collect groups of plot styles. The Plot Style Manager is a window that shows all the plot style tables available.

There are two plot style types:


**Color-dependent and named:** A drawing can use only one type of plot style table. You can convert a plot style table from one type to the other. You can also change the type of plot style table which has been set before. For color-dependent plot style tables, an object's color determines how it is plotted. These plot style table files have .ctb extensions. You cannot assign color-dependent plot styles directly to objects. Instead, to control how an object is plotted, you change its color. For example, all objects assigned the color red in a drawing are plotted in the same way.

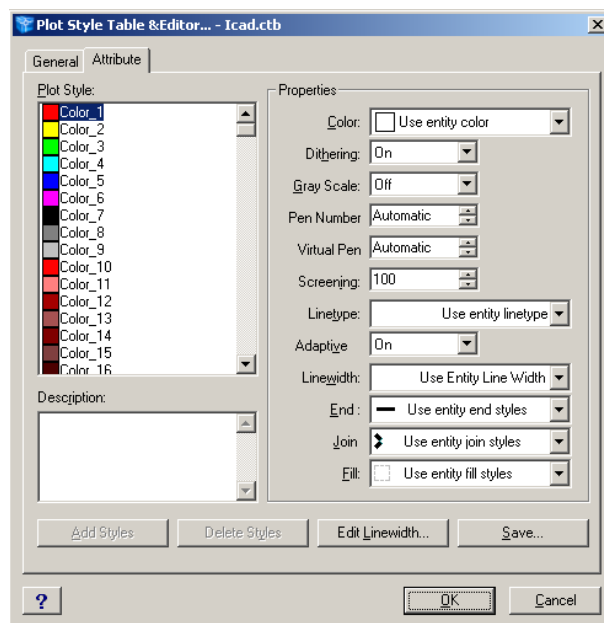
**Named plot style tables:** use plot styles that are assigned directly to objects and layers. These plot style table files have .stb extensions. Using them enables each object in a drawing to be plotted differently, independent of its color.

ICAD2008 support pcp file , stb file and ctb file . You can choose the plot style files that have been designed, Users can also customize related setup.



You can choose a stb or ctb plot style in this pull-down list. If you want to use other ctb or stb plot style, you can copy the ctb or stb files to C:\Program Files\GStarCAD2009 Professional\PrintStyles.

You can open the Plot Style Table Editor by click  icon, The Plot Style Table Editor displays the plot styles contained in the specified plot style table.



In this dialog box you can set the related parameters about plot style, such as color, linewidth.

### 1. Set Color, Screening, Grayscale, and Dither in Plot Style Tables

You can use a plot style to assign color, screening, grayscale, and dither properties.

- **Assign Plot Style Colors**

The default setting for plot style color is Use Object Color. With this setting, the object retains its layer or individually set color. If you assign a plot style color, the color overrides the object's color at plot time. You can specify one of 255 ACI colors, a true color, or a color book color. The plotter must be configured for True Color if you want to plot True Color plot styles.

- **Use Screening**

You can select a color intensity setting that determines the amount of ink placed on the paper while plotting. The valid range is 0 through 100. Selecting 0 reduces the color to white. Selecting 100 displays the color at its full intensity. Screening is effective only if your plotter is configured to plot colors or grayscale. Also, dithering must be enabled.

- **Use Dithering**

A plotter uses dithering to approximate colors with dot patterns, giving the impression of plotting with more colors than the ink colors available in the plotter. If the plotter does not support dithering, the dithering setting is ignored.

- **Convert to Grayscale**

When you select Convert to Grayscale, the object's colors are converted to grayscale if the plotter supports grayscale. Light colors, such as yellow, are plotted with light gray values. Dark colors are plotted with dark gray values. If you clear Convert to Grayscale, the RGB values are used for the object's colors. Conversion to grayscale is available whether you use the object's color or assign a plot style color.

## **2. Assign Pens to Plot Styles**

The Use Assigned Pen Number setting in the Plot Style Table Editor specifies which physical pen to use for each plot style.

## **3. Control Plotted Lineweight and Linetype**

Both linewidth and linetype can be set as an object property or controlled when plotted by using a plot style. Lineweight or linetype settings in the plot style override the



object's lineweight or linetype at plot time.

- **Assign and Display Lineweights**

When you select the Lineweight field in the Plot Style Table Editor, a sample of the lineweight as well as its numeric value are displayed. The default setting for plot style lineweight is Use Object Lineweight. You can modify an existing lineweight if the one you need is not available.

- **Assign Linetypes**

When you select the Linetype field in the Plot Style Table Editor, a list with a sample and a description of each linetype are displayed. The default setting for plot style linetype is Use Object Linetype.

#### **4. Assign Plotted Line End and Join Styles**

You can set the line end and join styles for objects that have lineweight assigned, either as an object property or as a plot style override.

- **Assign Line End Style**

The default setting for Line End Style is Use Object End Style, which is rounded. Assign a line end style in a plot style to override the object's default line end style at plot time.

- **Assign Line Join Style**

The default setting for Line Join Style is Use Object Join Style, which is rounded. Assign a line join style in a plot style to override the object's default line join style at plot time.

#### **5. Assign Plotted Fill Styles**

You can assign a variety of fill style options when plotting wide polylines, donuts, objects hatched with a solid fill, and solids.

The default setting for Fill Style is Use Object Fill Style. Assign a fill style in a plot style to override the object's fill style at plot time.

You can apply a global scale factor to non-ISO linetypes and fill patterns in plot styles.

### 12.8 Preview a Plot

It is good practice to generate a preview of the plotted drawing before sending the drawing to the printer or plotter.

You can preview the drawing from the Plot dialog box. The preview shows exactly how the drawing will look when plotted, including lineweights, fill patterns, and other plot style options.

When you preview your drawing, the active toolbars and tool palettes are hidden and a temporary Preview toolbar is displayed that provides buttons to plot, pan, and zoom the drawing.

Pull-down menu: File>Plot Preview

Toolbar: 

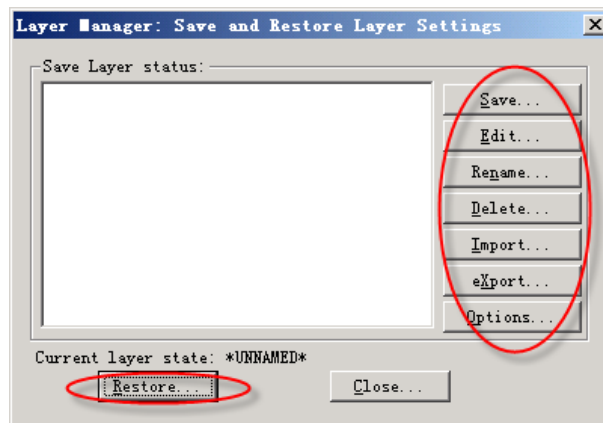
Command: Preview

## Chapter 13 Express Tools

### 13.1 Layer

#### 13.1.1 Layer Manager

Manage layer settings by allowing you to save and restore layer configurations into layer states. Layer states can be modified, restored, or renamed. Layer states are saved in the drawing, but they can also be exported to or imported from a LAY file.



**Save:** Save the current layer settings into the selected layer state. If a new name is specified, a new state is created.

**Edit:** Display the Layer Properties Manager dialog box, where changes can be made to the selected layer state. If no layer states exist, a new one is created.

**Rename:** Rename a layer state.

**Delete:** Delete a layer state and displays the following message: Are you sure you want to delete <layer state name>?

**Import:** Display the Import File Name dialog box to specify a LAY file name, drive, and directory for importing into the current drawing. A Find File button opens the Browse/Search subdialog box to locate saved .lay files.

**Export:** Display the Export File Name dialog box to specify a LAY file name, drive, and directory for exporting from the current drawing.

**Options:** Set options to use for layer status restore operations. This option allows you to specify the layer properties to be restored (on/off, freeze/thaw, color, linetype, and so

on).

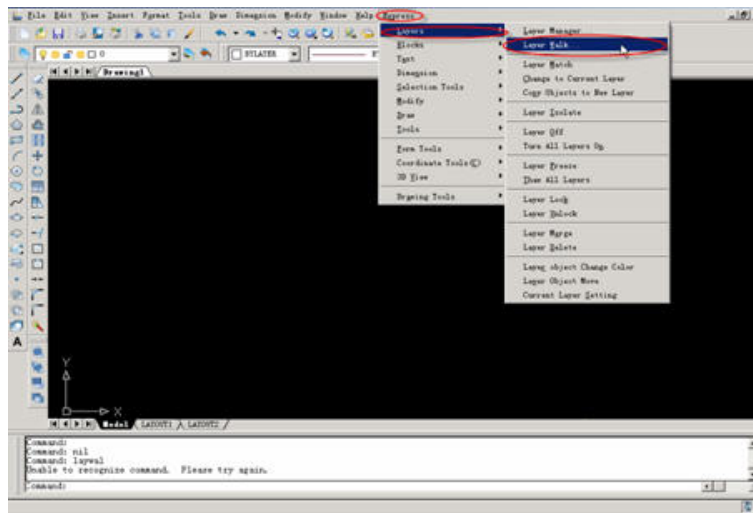
**Restore:** Restores the layer settings in the selected layer state without closing the dialog box.

Help Provides help for the Layer Manager.

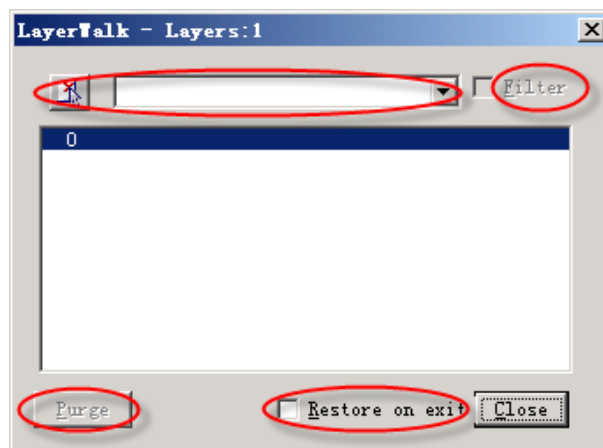
### 13.1.2. Layer Walk

Dynamically displays objects on selected layers

Pull-down menu: Express>Layer> Layer Walk



Displays objects on layers when the layer names are selected in the dialog box. The number of objects on the selected layers and the number of layers in the drawing are displayed. With this tool, you can alter the layer state when you exit, save layer states, and purge layers that are not unreferenced.



### LAYWALK Dialog Box Options

**Filter:** Turns the active filter on and off. When this checkbox is selected, the list displays the layers that match the active filter. When this checkbox is cleared, the full list of layers is displayed. (This option is not available when there is no active filter.)

**Purge:** Purges layers that are not referenced from the current selection. This button is unavailable if the current selection does not contain one or more unreferenced layers. For a list of layers that can be purged, choose Select Unreferenced from the shortcut menu.

**Restore on Exit:** If this option is selected, the layers are returned to their previous state when you exit the command.

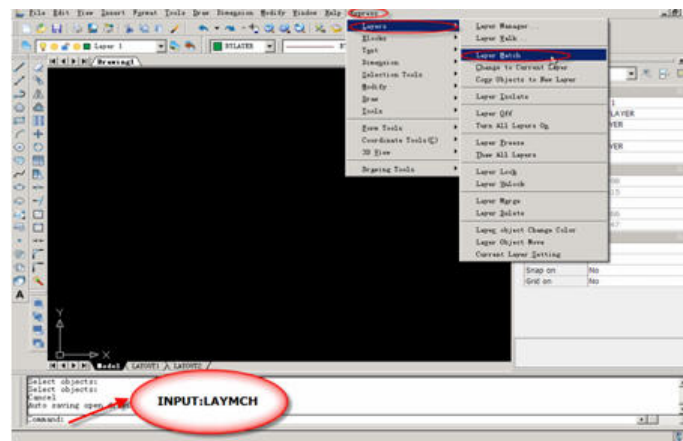
**Select Objects:** Using this button, you select objects on the screen in order to select their layers.

### 13.1.3 Layer Match

Change the layer(s) of selected object(s) to match the layer of a selected destination object.

Pull-down menu: Express>Layer>Layer Freeze

Command: LAYMCH



#### Usage Example:

Select an object, such as a door, whose layer assignment you want to change; then select an object on the layer you want to match.

#### Command: LAYMCH

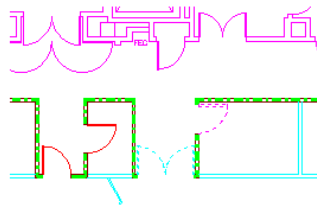
Select objects: Select the doors for the layers to be changed

Select objects: Press ENTER when you finish selecting objects

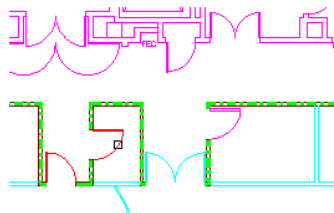
Select object on destination layer or [Type-it]: Select the door on layer N\_DOOR to match

2 objects changed to layer N-DOOR.

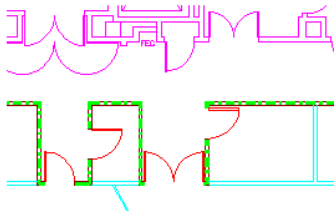
*Objects (doors are highlighted) are selected:*



*Select an object (for example, a door) on the layer to match:*



*Objects (doors) changed to match the layer of the selected object:*



### 13.1.4 Change to Current Layer

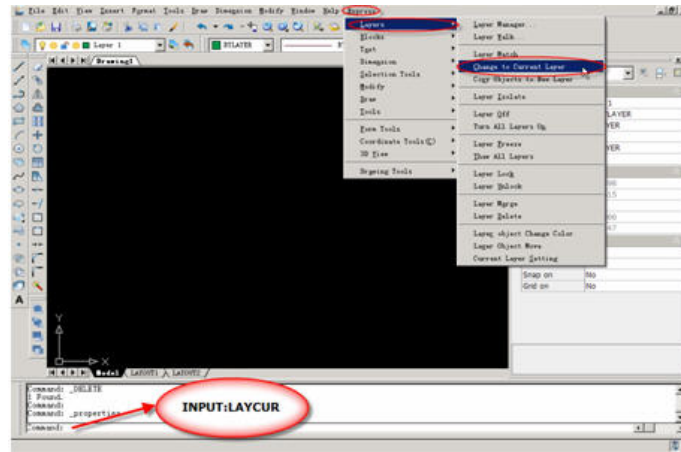
Change the layer of one or more objects to the current layer.

Pull-down menu: Express>Layer> Change to Current Layer

Command: LAYCUR

Select objects to be changed to the current layer:

Select objects: Use an object selection method



### Usage Example:

Select objects, such as the chairs on the layer CHAIR, and change them to the current layer, FURN.

**Command: LAYCUR**

Select objects to be changed to the current layer:

Select objects: Specify the object(s) you want changed to the current layer

Select objects: Press ENTER when you finish selecting objects

1 object(s) changed to layer LAYER1 (the current layer).

### 13.1.5 Copy Objects to New Layer

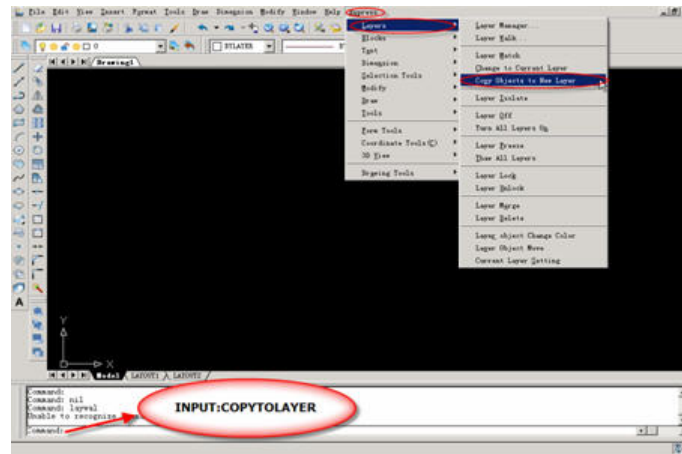
Copies selected objects to a different layer, leaving original objects intact.

You can select the target layer from a list of current layers, or you can create a new layer.

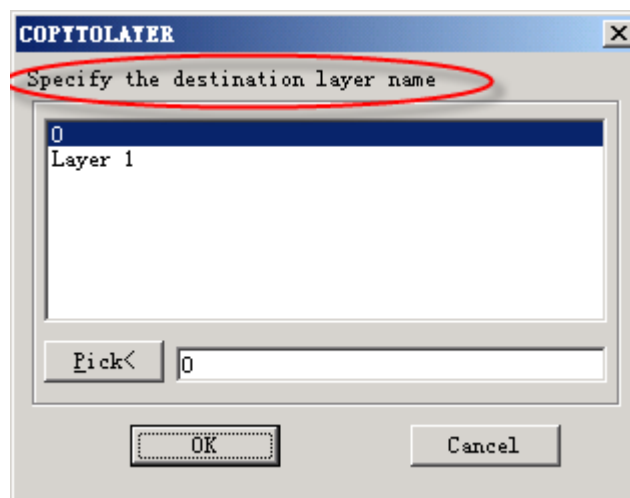
Pull-down menu: Express>Layer> Copy Objects to New Layer

Command: COPYTOLAYER

Select objects: Specify object(s) you want to copy to a different layer



### COPYTOLAYER Dialog Box Options



To specify the destination layer, you can select a current layer from the list or type a new layer name in the text box.

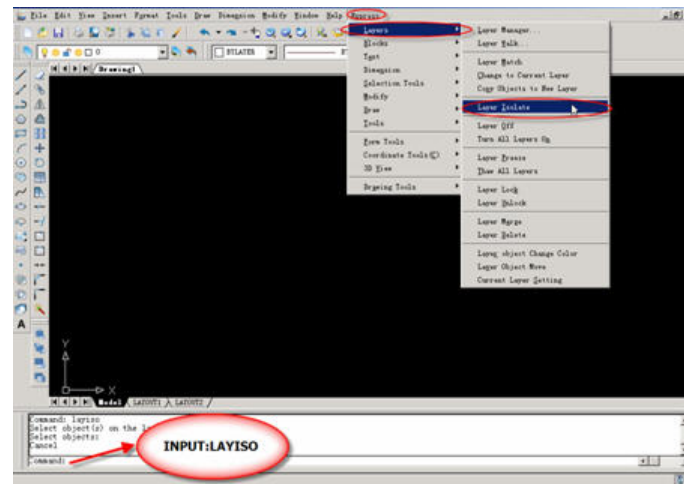
### 13.1.6. Layer Isolate

Isolate the layer(s) of one or more selected objects by turning all other layers off.

Pull-down menu: Express>Layer> Layer Isolate

Command: LAYISO





Select object(s) on the layer(s) to be isolated:

Select objects: Use an object selection method

### Usage Example:

Select an object, such as the hatch pattern, whose layers you want to isolate.

Command: LAYISO

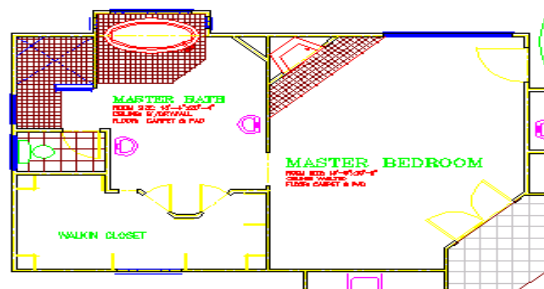
Select object(s) on the layer(s) to be isolated:

Select objects: Specify the object(s) whose layers are to be isolated

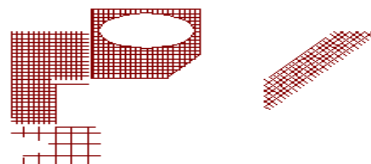
Select objects: Press ENTER when you finish selecting objects

Layer A-HATCH has been isolated.

*All layers on:*



*Hatch pattern isolated (turned on); all other layers off*



### 13.1.7 Layer off and Turn All Layers On

Layer Off: Turns off the layer of selected object(s).

Turn All Layers On: Turns on all layers

### 13.1.8 Layer Freeze and Thaw all Layers

Layer Freeze: Freezes the layer of the selected object(s).

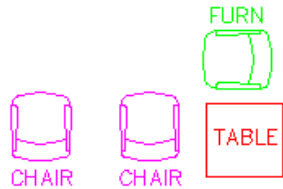
Thaw all Layers: Thaws all layers

### 13.1.9 Layer Lock and Layer Unlock

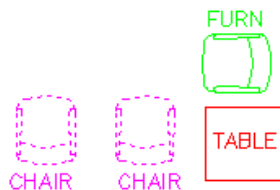
Layer Lock: Locks layer of the selected object.

Layer Unlock: Unlocks layer of the selected object.

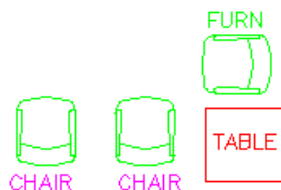
*Chairs are on the layer Chair:*



*Select two chairs to change to the current layer, FURN :*



*Two chairs changed to the current layer, FURN :*



### 13.1.10 Layer Merge

Moves all objects on the first layer selected onto the second layer selected. The first layer will be purged from the drawing.

Pull-down menu: Express>Layer>Layer Merge

Command: LAYMRG

Usage Example:

Select an object whose layer you want to merge into another layer.

**Command: LAYMRG**

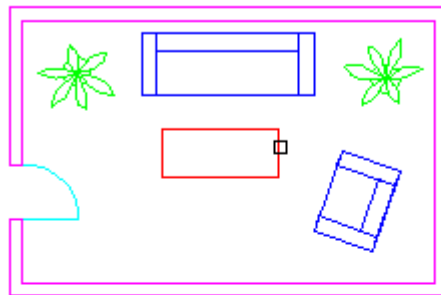
Select object on layer to merge or [Type-it/Undo]: Select sofa on layer SOFA

Selected layers: SOFA

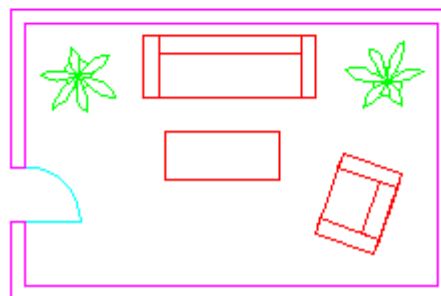
Select object on layer to merge or [Type-it/Undo] <done>: Press ENTER

Select object on target layer or [Type-it]: Select a table on layer FURN

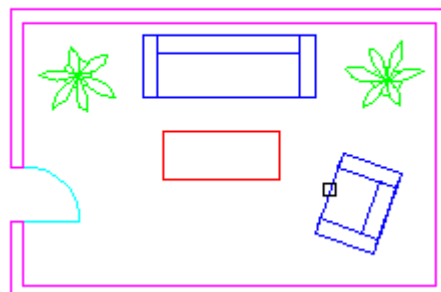
*Select table on layer FURN:*



*Layer SOFA is merged into layer FURN:*



*Select sofa on layer SOFA:*



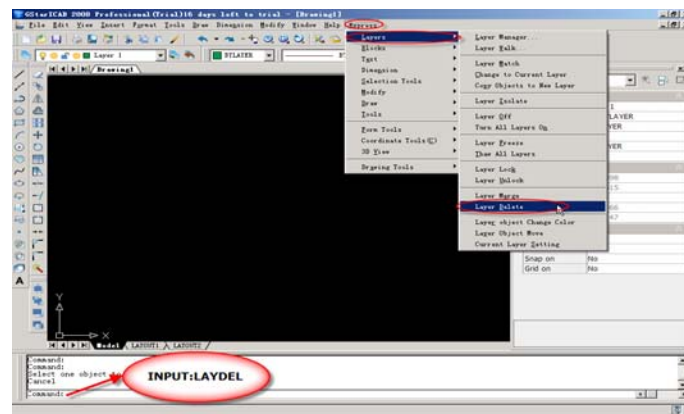
### 13.1.11 Layer Delete

Deletes all objects from the specified layer and purges the layer from the drawing.

**Pull-down menu: Express>Layer>Layer Delete**

**Command: LAYDEL**

Select object on layer to delete or [Type-it/Undo]: Use an object selection method for the layer to delete, or enter T to type the layer name



**Usage Example:**

Select an object to delete a layer.

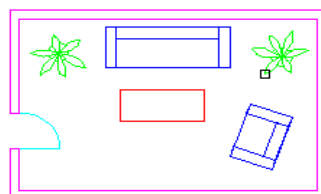
**Command: LAYDEL**

Select object on layer to delete or [Type-it/Undo]: Select a plant on the PLANTS layer

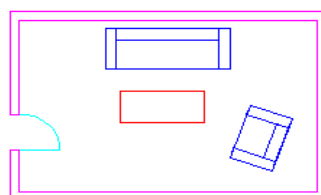
Selected layers: PLANTS

Select object on layer to delete or [Type-it/Undo] <done>: Press ENTER

*Select object on layer PLANTS:*



*Layer PLANTS deleted:*



### 13.1.12 Layer Object Change Color and Layer Object Move

**Layer Object Change Color:** After picking objects, you can change the layer color through inputting 1, you can change objects color through inputting 0

**Layer Object Move:** You can obtain layer information after picking objects, then you can choose and move the other objects which are in the same layer with the picked objects.

### 13.1.13 Current Layer Setting

Set the layer which contains the picked objects current layer.

## 13.2 Blocks

### 13.2.1. List Xref/Block Properties

List the type, block name, layer name, color and linetype of a nested object in a block or an xref.

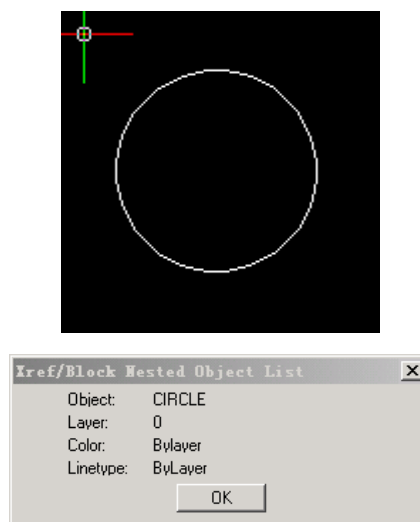
Command: XLIST

#### Usage Example:

Select an object as a nested block or xref file to display the Object and Block Name, Layer, Color, and Linetype properties.

Command: XLIST

Select nested xref or block object to list:



### 13.2.2. Copy Nested Objects

Copy objects nested in an xref or a block.

Command: NCOPY

### **13.2.3. Trim to Nested Objects**

Uses a block (or entities within a block reference) as a boundary to trim objects in a drawing

Command: BTRIM

### **13.2.4. Extend to Nested Objects**

Use a block (or entities within a block reference) as a boundary to extend objects in a drawing.

Command: BTRIM

### **13.2.5. Explode Attribute to Text**

Explode blocks, converting attribute values to text objects.

Command: BURST

### **13.2.6. Export Attribute Information**

Export block attribute values to an external file in tab-delimited ASCII format.

Command: ATTOUT

### **13.2.7. Import Attribute Information**

Import block attribute values from an external, tab-delimited ASCII file.

Command: ATTIN

### **13.2.8. Convert BLOCK to XREF**

Replace all instances of a standard block with an xref. Unbinds xrefs that are bound.

Command: BLOCKTOXREF

### **13.2.9. Replace Block with Another Block**

In the current drawing, you can use one block to replace another block

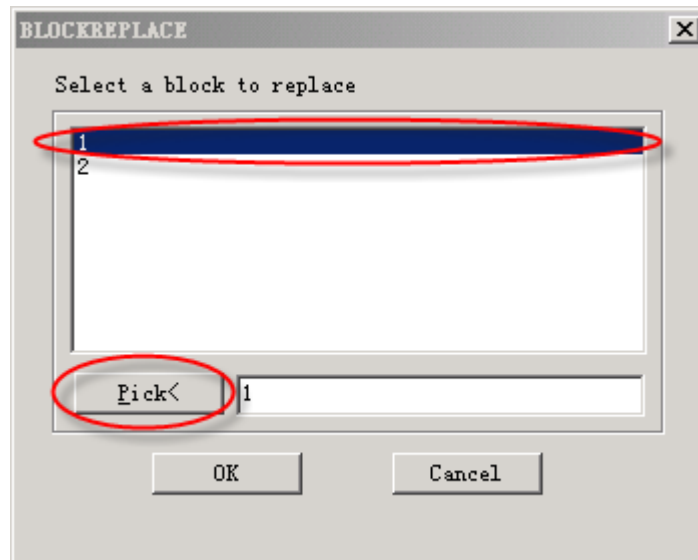
Command : BLOCKREPLACE

### **13.2.10. BLOCKREPLACE**

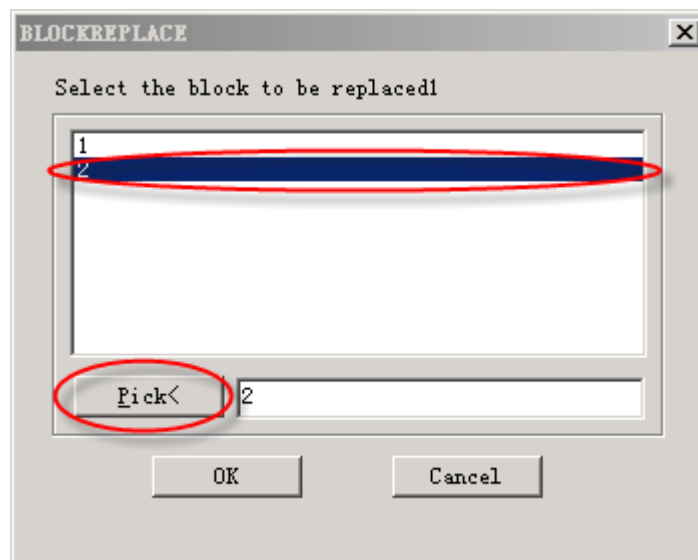
Allow you to globally replace all inserts of one block with another block.

BLOCKREPLACE searches the entire drawing for a specified block and replaces it with another block that you specify. You can have the unreferenced block purged when done.

Specify the block name to search for by selecting it in the list or by clicking the Pick button and selecting a block in the drawing:



Specify the replacement block:



### 13.2.11 Statistics Blocks Number

You can statistics the number of various blocks through this function.

### 13.3 Text

#### 13.3.1. Text Fit

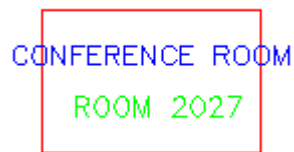
Stretch or shrink text object(s) by selecting new start and/or end points.

Command: TEXTFIT

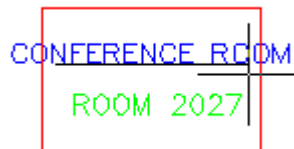
##### Usage Example:

Fit labels within enclosed borders by setting a new text width.

Select text



Select a new endpoint



Text size shrinks



#### 13.3.2. Text Mask

Place a mask object behind selected text or mtext. The rectangular mask is offset from the text by a specified value.

Command: TEXTMASK

#### 13.3.3 Text Unmask

Remove mask from text that has been masked with the TEXTMASK command.

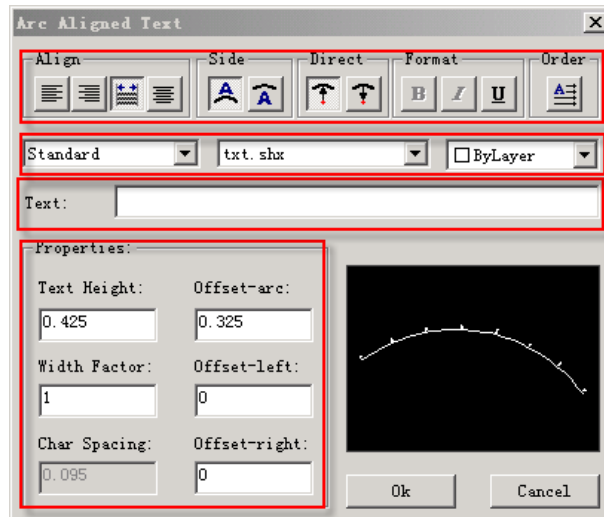
Command: TEXTUNMASK



### 13.3.4. Arc text

Place text along an arc.

Dialog Box Options



**Pick boxes:** Set controls for text location relative to the arc and its length. The Drag Wizard controls the behavior of the text when the arc is moved.

**Color:** Specify the text color.

**Text:** Specifies the text.

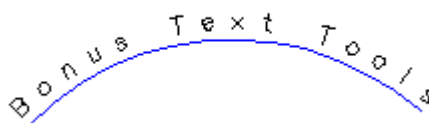
**Style Name:** Specify the style name for a selected font. Also specifies bold, italic, and underline settings.

**Font:** List the TrueType fonts and compiled shape (SHX) fonts

**Properties Section:** Specify the text height, width factor, character spacing, offset from arc, offset from left, and offset from right.

#### Usage Example

Draw an arc, and then use ARCTEXT to place text along the arc's curve.



### 13.3.5. Rtext

Creates reactive text

Reactive text (RText) objects are displayed the same way normal Text or MText objects are displayed, but the source for the text is either an external text file or the value of a DIESEL expression. You can edit an RText object with the RTEDIT command.

Command: RTEXT

Enter an option [Style/Height/Rotation/File/Diesel] <Diesel>: Specify an option

Options

**Style:** Select a text style.

**Height:** Specify a text height.

**Rotation:** Specify a rotation value.

**File:** Use an external text file.

**Diesel:** Use DIESEL code.

### 13.3.6. Redit

Allows you to edit existing remote text (Rtext) objects

### 13.3.7. Justify Text

Change a text object's justification without changing its position. Works with text, mtext, and attribute definition objects.

Command: TJUST

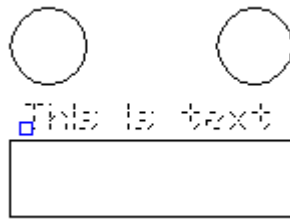
Select objects: Use an object selection method

[Start/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Start>:Choose the new justification

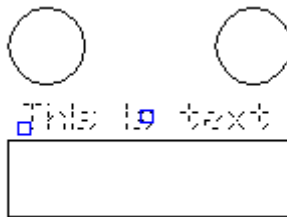
**Usage Example:**

Covert text to middle justified

Text left justified:



Text after justification change:



### 13.3.8. Rotate Text

Aligns text, mtext and block attribute objects to new orientation

Rotates text, mtext, and attribute definition objects so that they end up being as close as possible to horizontal or right-read (similar to the behavior of dimension text). Objects are rotated about their middle point in 180-degree increments. For example, if the text is upside-down, it becomes right-side up after a TORIENT operation. Similarly, left-read text becomes right-read text. The overall object location is not changed. As an option, you can specify a new absolute orientation angle for all selected text objects.

Command: TORIENT

Select objects: Use an object selection method

New absolute rotation <Most readable>: Enter an absolute angle or press ENTER

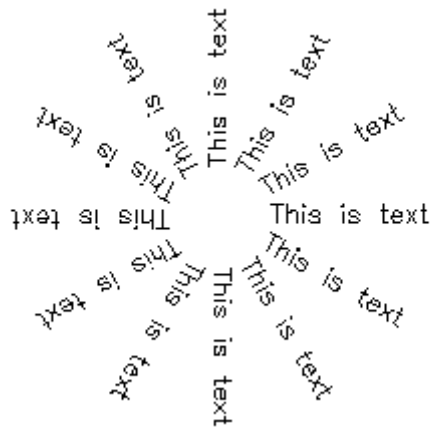
Rotation Options

New absolute rotation: Enter a new absolute orientation angle for all selected text

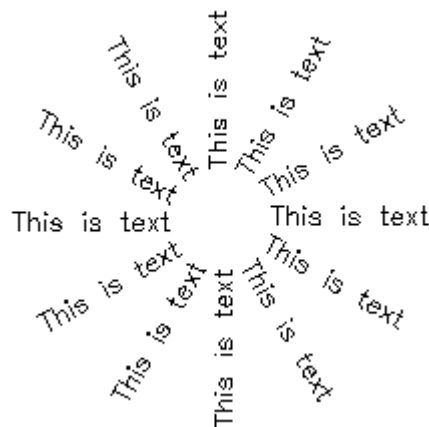
Most Readable: Orients selected text objects to the most readable right-side-up, right-read orientation

**Usage Example:**

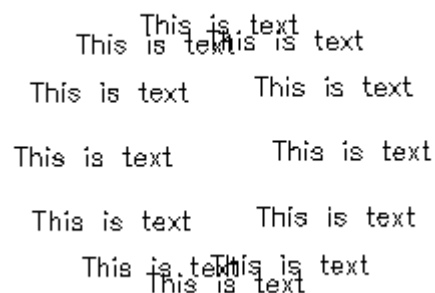
Text before rotation:



Text after rotation - most readable option:



Text after rotation - specified angle of 0 degrees:



### 13.3.9. Enclose Text with Object

Place a circle, a slot, or a rectangle around each selected text or mtext object.

The diameters of the circles can be constant or fitted to each object. The width and height of the slots and rectangles can be constant or variable.

Command: TCIRCLE

### Usage Example:

Circles: Constant radius:



Circles: Variable radius:



### 13.3.10. Tcount

Add sequential numbering to text objects. The numbering can appear as a prefix, suffix or replacement text.

Command: TCOUNT

Select objects: Use an object selection method

Sort selected objects by [X/Y/Select-order] <Select-order>:

Sorting Options

X: Places numbers by increasing the x-coordinate value of the selected text.

Y: Places numbers by decreasing the y-coordinate value of the selected text.

Select-order: Places numbers according to the order in which the text was selected.

### Usage Example:

Text item one  
Text item two  
Text item three  
Text item four  
Text item five

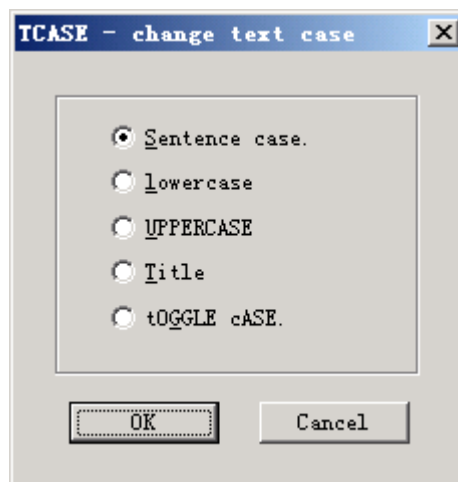
1 Text item one  
2 Text item two  
3 Text item three  
5 Text item four  
4 Text item five

### 13.3.11. Change Text Case

Change the case of selected text, mtext, attributes and dimension text.

Command: TCASE

Select objects:



### 13.3.12. Convert Text to Mtext

Convert one or more lines of text to mtext.

Command: TXT2MTXT

Select text objects or press Enter to set Options:

Select objects: Use an object selection method to select text

### Usage Example:

Select three text objects and convert them to mtext objects.

Select text items:

Normal text to  
be converted  
to Mtext

Text converted to mtext:

Normal text to  
be converted  
to Mtext

### 13.4 Dimension

#### 13.4.1. Attach Leader to Annotation

Attach leader line to mtext, tolerance, or block reference object.

Command: QLATTACH

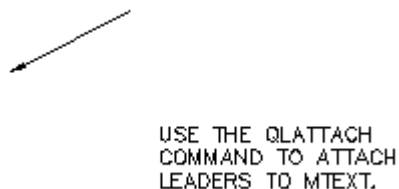
Select Leader: Select a leader

Select Annotation: Select the mtext, tolerance, or block reference object for the annotation

#### Usage Example:

Select a leader line, and then specify the mtext object to attach.

Separate leader line and mtext objects:



Leader line with mtext attached as one object:



### 13.4.2. Detach Leaders from Annotation

Detach leader line from mtext, tolerance, or block reference object.

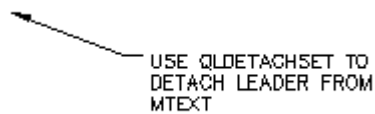
Command: QLDETACHSET

Select objects: Select leader line

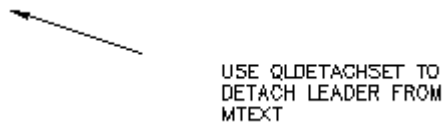
#### Usage Example:

Select a leader line to detach from an mtext object.

Leader with mtext object attached:



Leader detached from the mtext object, which can then be moved



### 13.4.3. Global Attach Leader to Annotation

Globally attaches leaders line to mtext, tolerance, or block reference objects

Command: QLATTACHSET

Select objects: Select one or multiple leader lines and annotation objects

#### Usage Example:

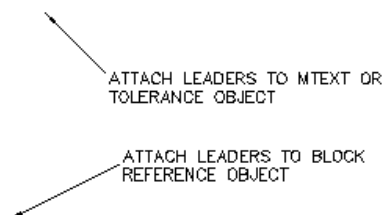
Select leader line(s), and mtext objects to attach.

Select objects: Select one or multiple leader lines and annotation objects

Number of Leaders = 2

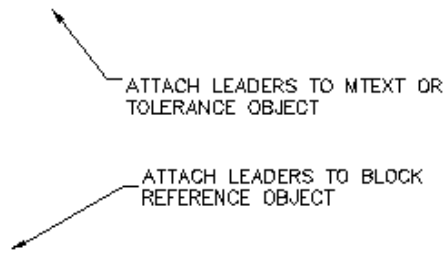
Number with annotation detached = 2.

Leader with detached mtext object:





Leader attached to mtext object:



### 13.4.4. Dimstyle Export

Exports named dimension styles and their settings to an external file.



DIMEX Dialog Box Options

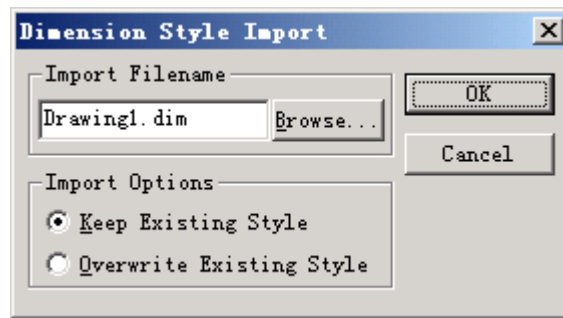
**Export Filename:** Creates or opens a DIM file. Enter the DIM file name or select Browse to locate the file. If the file does not exist, it is created. If the file does exist, a prompt requests permission to overwrite. The new file is an ASCII file.

**Available Dimension Styles:** Selects the dimension styles to write to the ASCII file. The current dimension styles are displayed. Each selected style is written to the file indicated in the Export Filename group.

**Text Style Options** Saves either the dimension's Full Text Style Information or the Text Style Name Only to the ASCII file.

### 13.4.5. Dimstyle Import

Imports named dimension styles from a DIM file into the current drawing.



### DIMIM Dialog Box Options

**Import Filename:** Names the file you want to import. Enter an existing file name or select Browse to specify a DIM file to import. The file must be of the form generated by the corresponding DIMEX command. All dimension styles in the selected file are imported into the current drawing. Text styles are also imported.

**Import Options:** Includes Keep the Existing Style, to keep the dimension style in the current drawing and Overwrite Existing Style, to overwrite styles in the current drawing. The Keep the Existing Style option ignores the similarly named style being imported; Overwrite Existing Style uses the similarly named style being imported.

### 13.4.6. Reset Dim Text Value

Restore a measurement value to overridden or modified dimension text.

Command: DIMREASSOC

### 13.4.7. Update Dimension

You should select a base

You should select a standard dimension at first; then update other dimension to the standard dimension with the same settings

Command: Dimupdate

### 13.4.8. Change Dimension Text

Command: Chgdimtxt

Change the dimension content into the specified value

### 13.4.9. Change Dimstyle

Command: Dimstyle

Apply the current dimension style to the dimensions in the drawing

### 13.5 Selection Tools

#### 13.5.1 Get Selection Set

Create a temporary selection set of objects.

Command: GETSEL

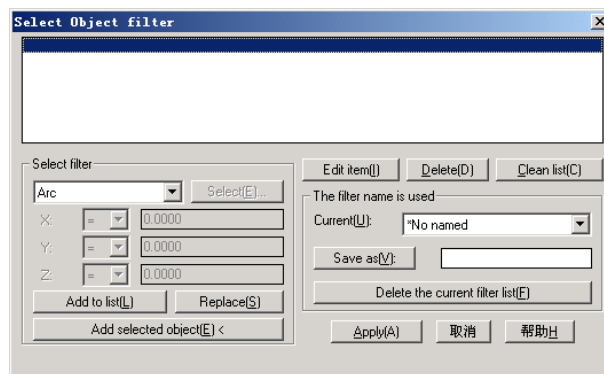
Select an object on the Source layer <\*>: Use a selection set method or press ENTER for all layers

Select an object of the Type you want <\*>: Use a selection set method or press ENTER for all object types

#### 13.5.2 Filtrate Selection

Select, edit, and name filters for object selection.

Command: Filter



Options:

##### 1. X, Y, Z Parameters

Define additional filter parameters depending on the object.

In the filter parameters, you can use relative operators such as < (less than) or > (greater than).

##### 2. Select

Display a dialog box listing all items of the specified type in the drawing. Select the items to filter.

##### 3. Add to List

Add the current Select Filter property to the filter list. Filter properties that you add to

the unnamed filter remain available during the current work session unless you manually delete them.

### **4. Replace**

Replace the filter property selected in the filter property list with the one displayed in Select Filter.

### **5. Add Selected Object**

Add one selected object in the drawing to the filter list.

### **6. Edit Item**

Move the selected filter property into the Select Filter area for editing. To edit a filter property, select it and choose Edit Item. Edit the filter property and choose Substitute. The edited filter replaces the selected filter property.

### **7. Delete**

Delete a selected filter property from the current filter.

### **8. Clear List**

Delete all the listed properties from the current filter.

### **9. The filter name is used**

Displays, saves, and deletes filters.

### **10. Current**

Display saved filters. Select a filter list to make it current. The named filter and its list of properties are loaded from the default file, filter.nfl.

### **11. Save As**

Save a filter and its list of properties. The filter is saved in the filter.nfl file. Names can contain up to 18 characters.

### **12. Delete Current Filter List**

Delete a filter and all its properties from the default filter file.

### **13. Apply**

Exit the dialog box and displays the Select Objects prompt, where you create a selection set. The current filter is used on the objects you select.

## **13.6. Modify**

### **13.6.1. Move/Copy/Rotate**

Move, copy, rotate, and scales object(s) with a single command.

Command: MOCORO

Select objects: Use an object selection method

Base point: Specify a base point

[Move/Copy/Rotate/Scale/Base/Undo]<eXit>: Specify an option

Options

**Move:** Move object(s).

**Copy:** Make multiple copies of object(s).

**Rotate:** Rotate object(s).

**Scale:** Dynamically scale object(s).

**Base:** Specifies a base point for object(s) to be moved, copied, scaled or rotated

### **13.6.2 Multiple Copy**

Copy multiple objects with Repeat, Array, Divide and Measure options.

Command: COPYM

Select objects: Select the objects to be copied

Base point: Specify a base point

Second point or [Repeat (last)/Divide/Measure/Array (dynamic)/Undo] <exit>:

COPYM Options

**Repeat:** Repeat the last copy using the same offset distance.

**Divide:** Linear divide option. Prompt for an additional point and the number of copies to make. Makes evenly spaced copies of the selected objects between the base point and the second selected point.

**Measure:** Linear measure option. Prompt for an additional point and the distance between copies. Makes evenly spaced copies at the specified distance between the base point and the second selected point.

**Array:** Display the Array Options prompt

### 13.6.3 Extended Offset

This enhanced version of the OFFSET command offers several advantages over the standard command, including layer control, undo, and a multiple option.

Command: EXOFFSET

Specify offset distance or [Through] <T>: Specify the offset distance or through point

Select object(s) to offset or [Options/Undo]: Select an object to offset, change options or undo

Specify an option to set [Distance/Layer/Gaptype]: Specify a option

EXOFFSET Options

**Distance:** Specify the offset distance or through point

**Layer:** Specify the layer for new objects. Current Layer or Source Object Layer

**GapType:** Set or changes the offset mode as controlled by the OFFSETGAPTYPE variable. The options are: Normal, Fillet or Chamfer.

### 13.6.4 Multiple Pedit

This command is the same as the standard command nearly.

Command: MPEDIT

### 13.6.5. Cookie Cutter Trim

Trim one side of a line which is intersecting a selected lines

Command: EXTRIM

### 13.6.6 Full-Page Proof Cut Drawing

This function is added according to user's requirement .You can select a part of a drawing as full-page proof then copy it to a specified location.

Command: ICAD\_DYJT

### 13.7. Draw

#### 13.7.1 Breakline Symbol

Creates a polyline and inserts the breakline symbol.

Command: BREAKLINE

Options are provided to control the relative size and appearance of the breakline symbol and an extension of the polyline beyond the selected start and end points.

##### Options

**Block:** Determine the block to be used as the breakline symbol.

**Size:** Determine the size of the breakline symbol.

**Extension:** Determine the length of the line extension beyond selected points.

### 13.8. Tools

#### 13.8.1 Xdata Attachment

Attaches extended object data (xdata) to a selected object.

Command: XDATA

Select object: Use an object selection method

Application name: Specify application name

[3Real/DIR/DISP/DIST/Hand/Int/LAyer/LOnG/Pos/Real/SCale/STr/eXit]<eXit>: Enter an option

##### Options

**Application name:** An ASCII string up to 255 bytes long (group code 1000).

**Layer:** A layer name (group code 1003).

**Hand:** An object handle (group code 1005).

**3Real:** 3 real numbers (group code 1010).

**Pos:** A 3D World space position (group code 1011).

**Disp:** A 3D World space displacement (group code 1012).

**Dir:** A 3D World space direction (group code 1013).

**Real:** A real number (group code 1040).

**Dist:** A distance (group code 1041).

**Scale:** A scale factor (group code 1042).

**Int:** A 16-bit integer (group code 1070).

**Long:** A 32-bit signed long integer (group code 1071).

### 13.8.2. List Entity Xdata

Lists the xdata associated with an object.

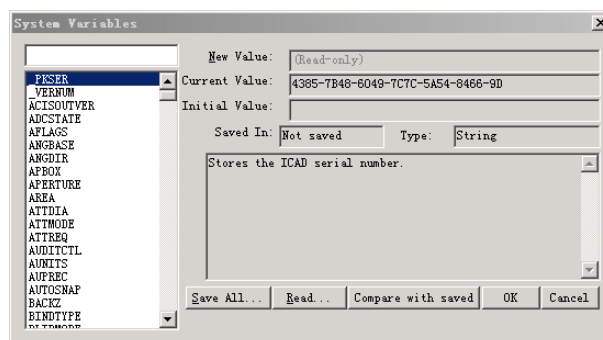
Command: XDLIST

Select object: Use an object selection method

Application name <\*>: Specify an application name or press ENTER for all applications

### 13.8.3. System Variable Editor

Allows you to view, edit and save system variable settings.



#### SYSVDLG Dialog Box Options

**List Box:** Display a list of system variables. The edit field above the list box allows for filtering the list box display based on user input. For example, entering DIM\* in the edit field filters the list to display only system variables that start with DIM. If nothing is entered in the edit field, the list displays all system variables. If more than one system variable is highlighted, all edit and display options are unavailable.

**New Value:** Display the current value of the system variable highlighted in the list box and allows you to edit that value. No changes are committed until the dialog is closed with the OK button. If the selected variable is a read-only type, this edit field will be unavailable.



**Current Value:** Display the value of the system variable highlighted in the list box as of when SYSVDLG was started.

**Initial Value:** Display the initial value of the system variable highlighted in the list box. This is the value the system variable has in a new drawing.

**Saved In:** Display where the value of the highlighted system variable is saved.

**Type:** Display the type of the highlighted system variable; for example, integer, real, string, and so on.

**Save All:** Save the current system variable settings to an SVF or SCR file. SVF files can be read into SYSVDLG and SCR files are AutoCAD scripts that can be run from the AutoCAD command line. The button will display Save All, Save Selected or Save Filtered depending on the condition of the system variable list. If only one system variable is selected and the list is not filtered, all the system variables will saved. If more than one system variable is selected, only the highlighted items are saved. If the list has been filtered, as with a wild-card, the list is saved as displayed.

**Read:** Restore saved settings from a SVF file.

No changes are saved to the drawing until you choose OK. All pending changes can be discarded with the Cancel button. SYSVDLG will validate new system variable values as they are changed.

### 13.8.4 Make Linetype

Creates a linetype based on selected objects.

Command: MKLTYPE

## 13.9. Form Tools

You can draw a form through dragging, then you can edit the form such as uniting form, comparting form, adding bias, adding one row, adding one column and etc, you can also fill in the form and adjust the text location.

### **13.10. Coordinate Tools**

#### **13.10.1. Coordinate Point**

Dimension the point coordinate on the drawing

Command: DIMCOORD

#### **13.10.2. Change Z Coordinate**

You can choose several objects then change the Z coordinates at one time.

Command: CHANGEZ

### **13.11. 3D View**

You can adjust viewport direction through this command.

### **13.12. Drawing Tools**

#### **13.12.1. Drawing Compare**

Drawing compare is an exclusive function in GStarICAD which is added according to user's requirement, Other CAD software has no the feature function.

The Drawing Compare feature enhances collaboration by using color-coded displays to show items on a drawing that have been changed, added, or deleted by other members of the design team. Changes to items such as styles, fire ratings, or other non-graphical properties are also tracked. The Drawing Compare features make communication across the design team clearer than ever.

Some engineers often meet such problems: sometimes they need to judge what amendments had been made between two drawings which were drawn at different time. The usual solution is to mark the amended contents with text. This way is difficult to drawer and reader.

The function makes users convenient to locate the changed parts. The program will analyze the difference between the two drawings automatically, as long as you select the drawings that you want to compare. The difference will be marked with different colors.

Users also can set the marking color by clicking “option” button.

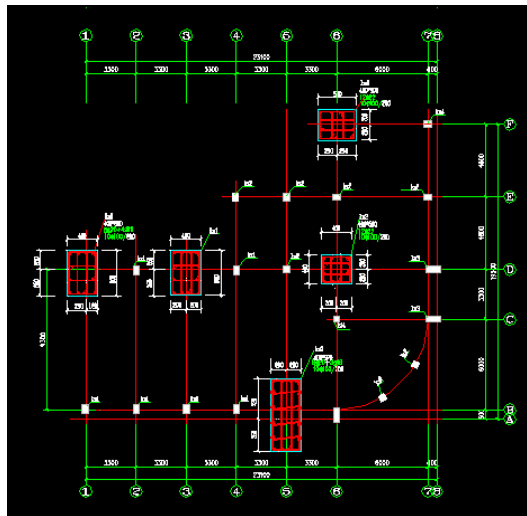
### (1). Execute

Pull-down Menu: Express>Drawing Tools>Drawing compare

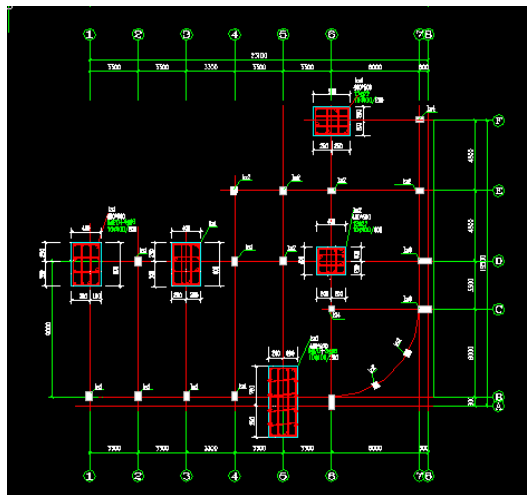
Command: CMP

### (2) The usage of the function

1) Open the original drawing and the changed drawing

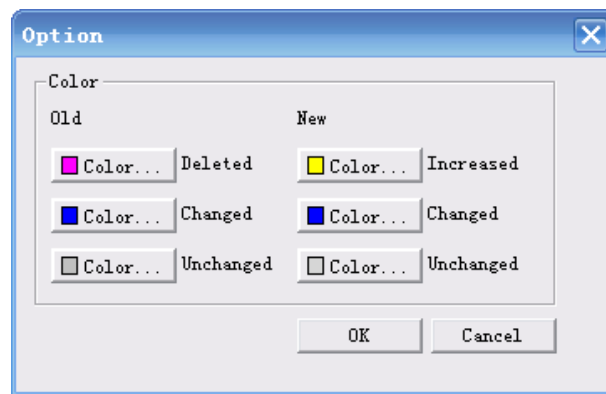


The original drawing

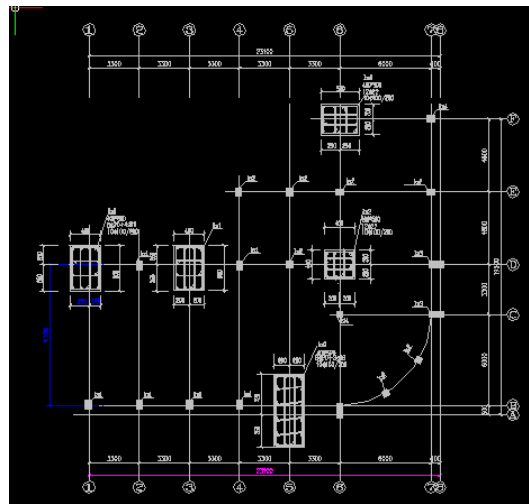


The changed drawing

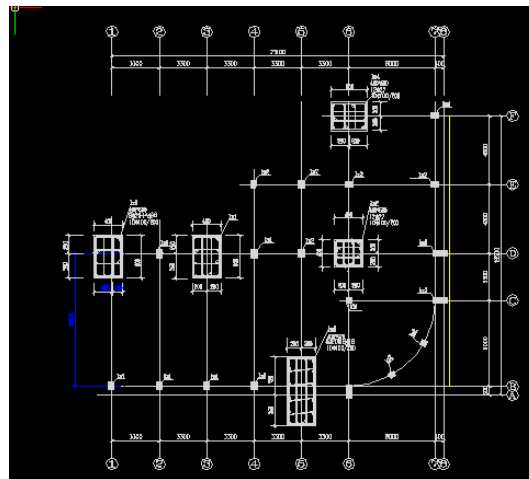
2) Set the colors in the option dialog



3) The below are the drawings which were compared



The old drawing



The new drawing

From the last two drawings you can see the amendments which have been marked with different colors. Drawing compare is developed according to users' requirement, currently AutoCAD has no similar function. This function is very helpful to coordinate

design between different fields. It saves much time from amending contents to reading drawings. It is very popular with engineers.

### 13.2.2. Drawing Lock

Designers often let others see own drawings, but don't want others to modify the drawings at the same time. "drawing lock" function can make the drawing as a whole, if others want to modify the drawing he only can edit the whole drawing, and can not edit the specific entity, it is a good way to prevent others from modifying own drawing.

Command: lockup

Command prompt: Select objects

At this time as long as you select all the entities in the current drawing

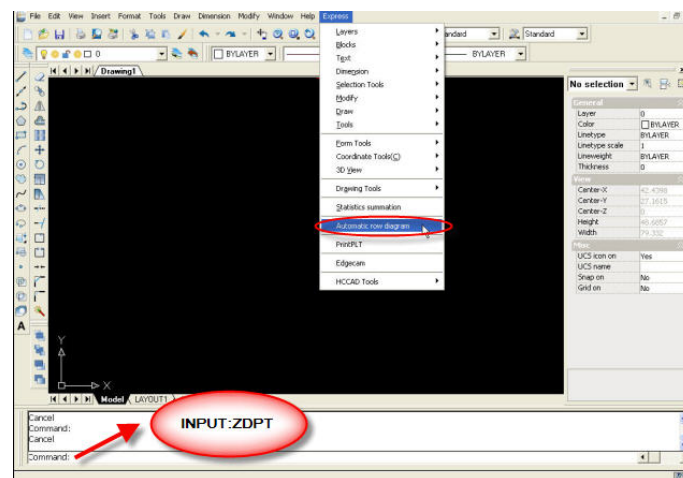
Note: Please make a copy for yourself before you use this function, because the drawing which is used for "drawing lock" is for others to see.

### 13.13. Arrange Drawings Automatically.

Many drawings can be arranged automatically through software calculation. It not only can enhance print efficiency but also can save paper.

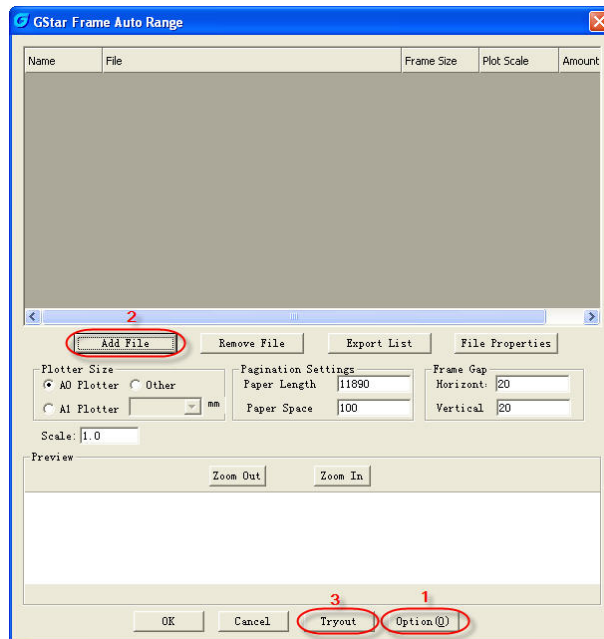
Command: ZDPT

Menu: Express>Automatic row diagram



Operation Steps:

- (1) If you use this function at first time, you should do some settings at “Option” item
- (2) Select the arranged file from “add file” item
- (3) After selecting the arranged file you can “tryout” these files, and preview the arranged effect.

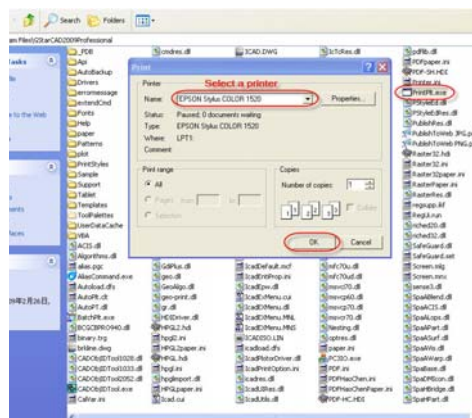


### 13.14. PLT Output Tool

Users can send PLT files to printer to print.

Operation Steps:

- 1) Right-click GStarCAD icon>Properties>Find Target
- 2) Select PrintPlt.exe in the dialog box



- 3) Double-click the PrintPlt.exe file, then the “Print” dialog box will be displayed,

in the dialog box you can select printer or do some settings.

- 4) Click OK button, the “Open” dialog box is displayed, you can select the printed file.

### 13.15. HCCAD Tools

HaoChen Tools integrated many convenient drawing tools, user can use GStarCAD platform software to finish many professional design such as pipeline design.

#### 1) Drawing Scale

Command: SASCL

Set drawing scale before starting to draw, software will adjust the scale automatically when you use other tools such as modifying lineweight.

#### 2) Control Parameters——Cross to break width

Command: CBKWID

Set the width of fast tripping cross, about “the fast tripping cross” we will introduce it later

#### 3) Rotate Cursor

Command: RTCUR

Set the cursor to the angle that you want. You can choose 0 degree or 45 degree which have been set; you also can choose “Alongside Line” to select a line, the cursor angle will become corresponding with the line; If you select “Real Time Rotate” and it seems that you enter into a little play, you can use AWSDF four key to control the rotate of the cursor till you press Enter to finish it

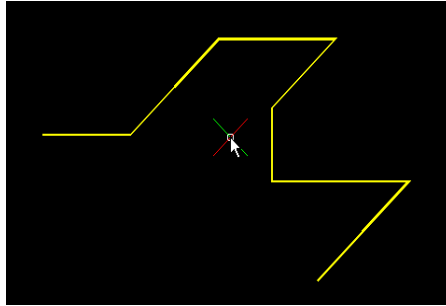
#### 4) Line Such as Chord

Command: SALPL

It is very convenient to use “Line such as chord” function to draw isometric drawing. Software will prompt you to input lineweight value, you can set the value according your need, then start to draw your entity, the graphic way is the same with drawing common multiline, software supplies the usually used angle 0 degree and 45 degree,

you can set the angle value by yourself, it will feel the drawing process is fast and easy.

The sample drawing is below:



### 5) Extension

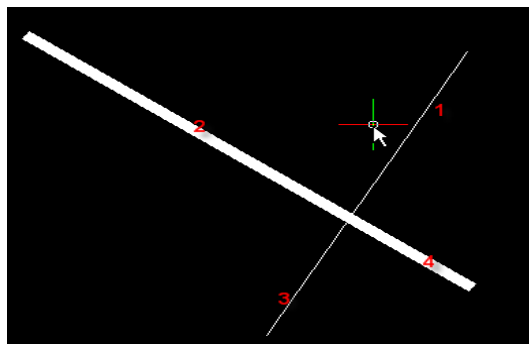
Command: ETT

The extension location can be displayed in real-time until to the final location

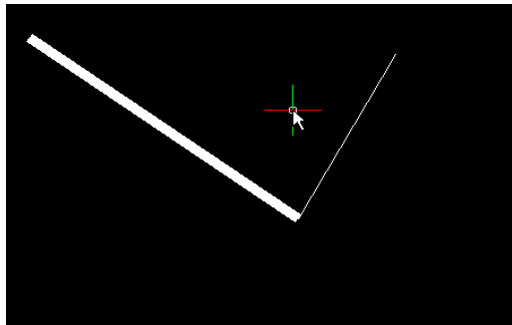
### 6) Normal Connect

Command: GXFILT

This function is similar with 0 distance chamfer. If there are two intersectant line or pline, the selected part will be retained and other part will be cut



Select 1 and 2 to use “normal connect”, and so 3 and 4 will be cut, the effect is below



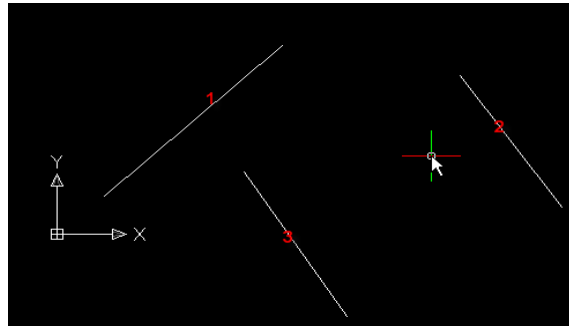
### 7) Flex connect

Command: GXFSS

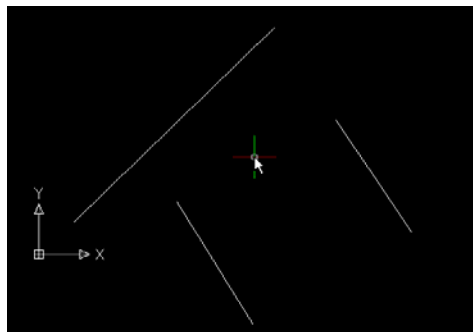


As long as two lines or pline are not parallel, they can realize “flex connect”, this function will make selected line extend to the location where it intersect with the target line or where it intersect with the extension line of the target line. The sample picture is below:

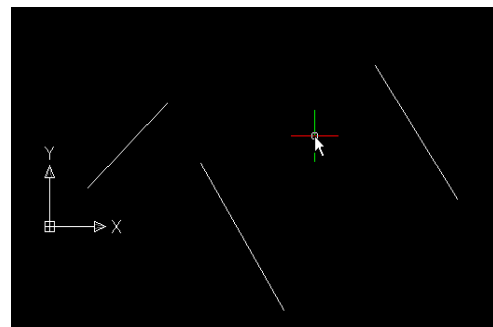
Now use 1 with 2 and 1 with 3 to use this function separately



The situation before “Flex connect”



1 with 2 to use the function

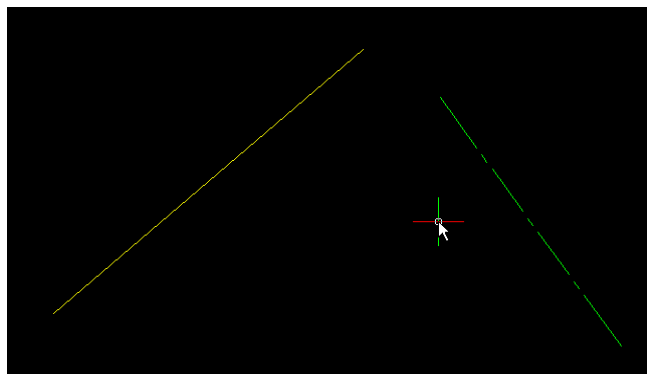


1 with 3 to use the function

### 8) Super Fillet

Command: SFILLET

Why we can this function “Super fillet”? because this function not only can finish the usual fillet operation but also can unify the two lines’ information such as layer、color、 line type and etc, the unified standard is according to the firstly selected line.



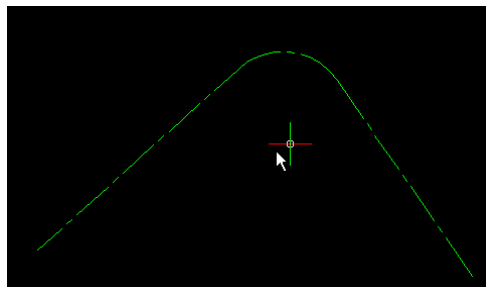
In this drawing, these two lines have different color and line type, now we use super fillet function to them.

Please enter the Fillet radius<0>:

The software will prompt you after you input the radius

Please select the first line segment:

The selected first line will determine the layer、color and line type of filleted two lines, here we select the green line as the first line, then select the yellow line to do super fillet, the result is below:



### 9) Modify linewidth

Command: LCW

Change the linewidth of specified pline, if you select a line and software will translate this line into pline and set linewidth automatically for it. It is very convenient to edit linewidth.

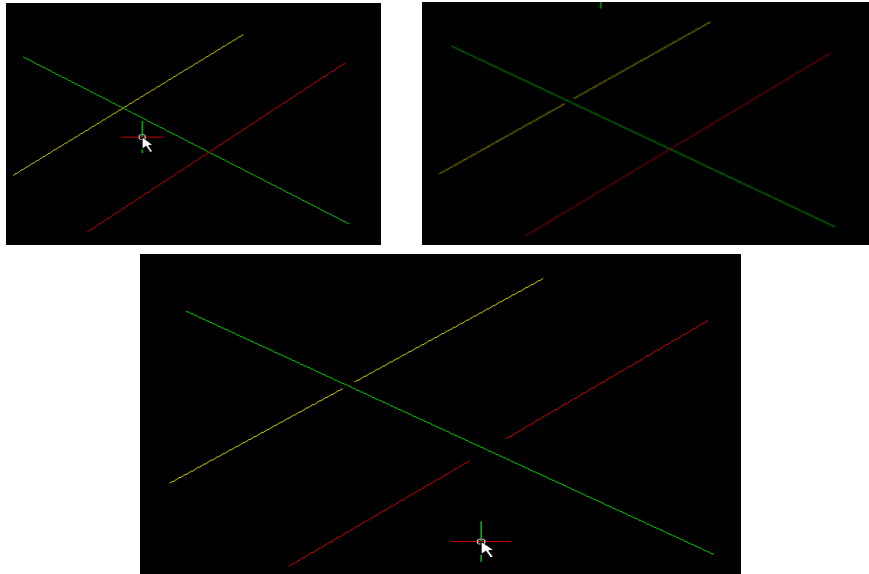
### 10) Cross to break soon

Command: CBK

We often need to break many lines when drawing, but it is very inconvenient to do this by hand. This aim can be easily finished by this function supplied by GstarCAD

Software will prompt you to select a line which is to break other lines namely the cover line segment then you can select the breaking lines.

Use the green line as the cover line to break the red line and yellow line. The effect is below:



### 11) Modify Text

Command: EDTXT

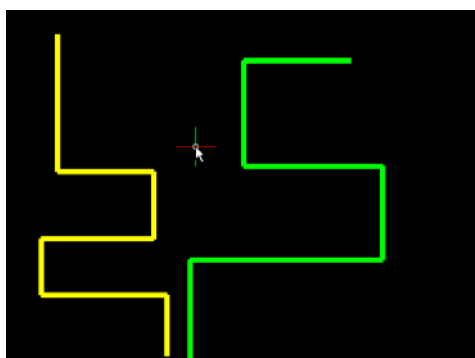
Supply some functions to single text such as: height、text width factor、angle、obliquing angle

### 12) Super Axonometric

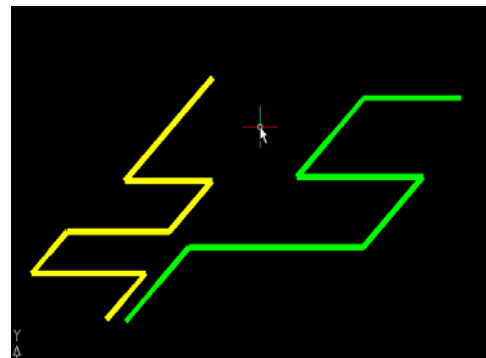
Command: ZC

It is a very helpful function, you can translate plan drawing into isometric drawing easily with it.

The operation way is very easy, you should select the translated drawing at first, then set the axonometric angle, at last press Enter to finish your operation.



Before translation



After translation