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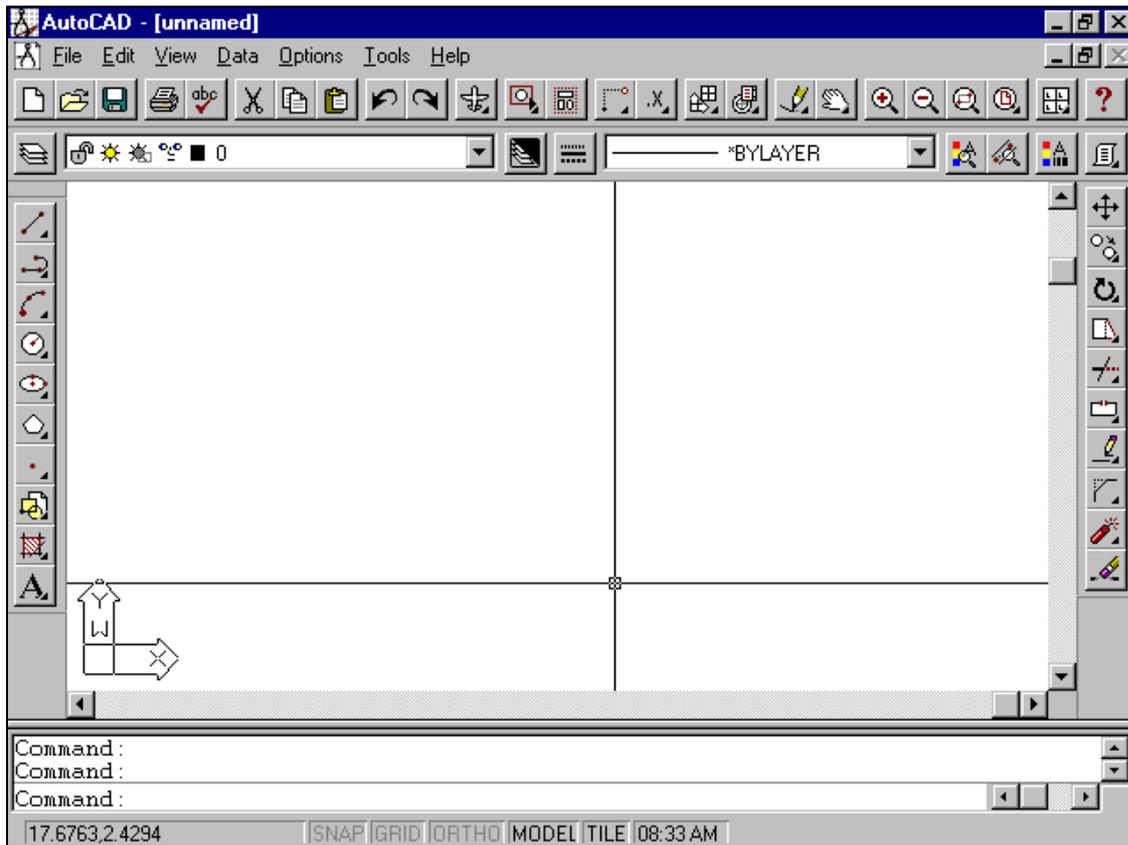
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AUTOCAD SCREEN

The autocad interface consists of several different screen elements most of which you can customize.



The drawing area:

The main blank area in the middle of the screen is where you draw. You can compare this space to a piece of drafting paper except for that this space can be of any size....even the size of a factory!

To specify a point it is conventional to put the x cor-ordinate first then a comma and then the y coordinate

The ucs icon

This is the symbol with two arrows at the bottom left corner of the drawing area. This symbol is called the ucs (User Cordinate Sytem). The arrow points to the Positive x and y directions.

The crosshairs

When you move your mouse across the drawing space you note that t5here are two intersecting lines with a small box at their intersection. The small box is calleed the pickbox as it khelps in

picking objects. The lines are called crosshairs as they help you in judging the location of the mouse pointer on the screen in relation to other objects.

The menus and toolbars

At the top of your screen is the title bar and directly beneath the title bar is the menu bar. The menus and toolbars together allow you to give autocad commands to draw, edit, get information, and so on. You can customize the menus and toolbars to suit your needs so that your screen may appear somewhat differently. There are many more toolbars that you may display when you need them. Some of these toolbars are dimensioning, solids, render etc

The command line

At the bottom of the screen you see a separate window showing approximately three lines of text (it can be changed to show as many lines as you like)

Notice the word **command**. This is the command line and it's where all commands can be executed when typed in here.

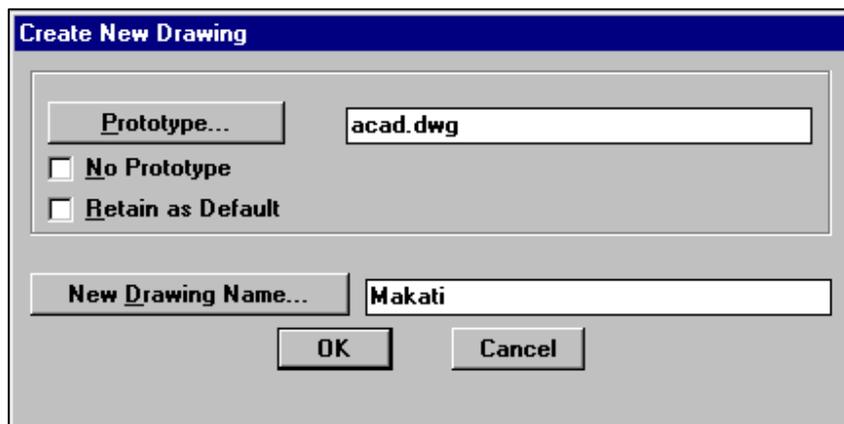
The status bar.

At the very bottom of the screen is the status bar. At the left are the **x,y** coordinates. As you move your mouse around they change accordingly.

Opening autocad and starting a drawing.

You start autocad by clicking the start button then programs then autocad r13.

Once you do this autocad starts and prompts you with the following screen:



Type in the new drawing name you are creating in the text box and click ok.

Autocad creates a new drawing with this name you provided.

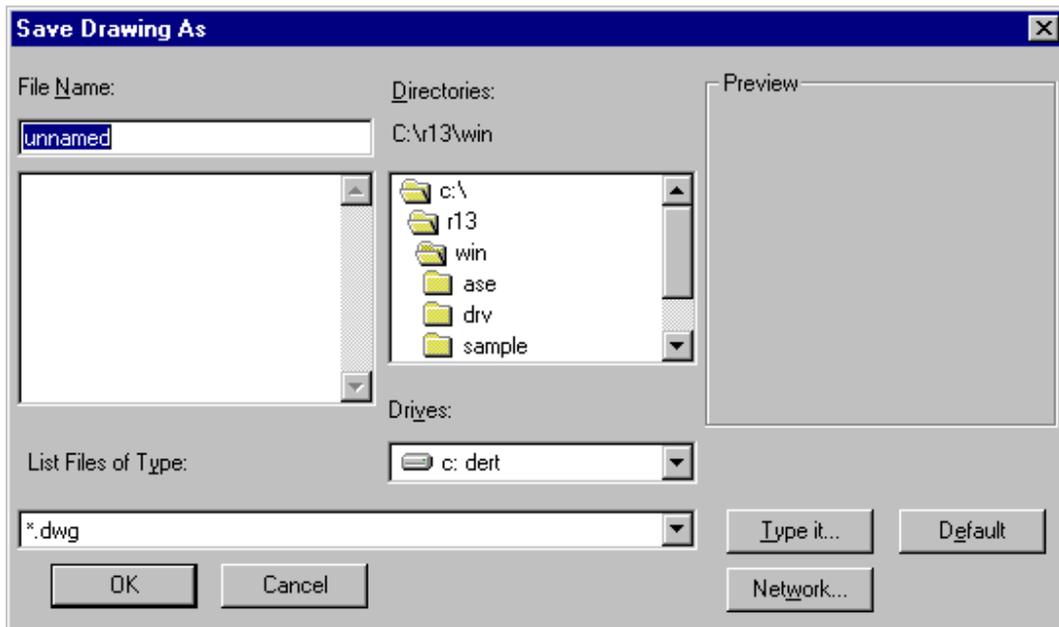
Now use the **line** command to draw lines anyhowly on the screen.

Once you finish it is time to save your drawing.

Saving your drawing

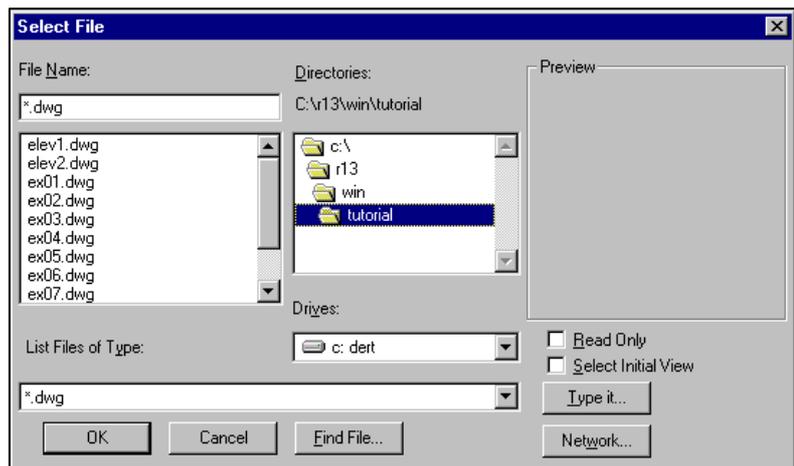
Saving a drawing in autocad is similar to saving a file like in any other application. You should get into the habit of saving your work in every 10 to 15 minutes interval to avoid losing your drawings.

To save your work go to **File Save** and the file will be updated if you had given it a name initially. If you had not a dialog box will appear in which you may type the name you want to give your file .



Opening a drawing

AutoCad offers a number of options for opening a drawing. When you are creating a new drawing you may choose to use a template. A template is a drawing with some ready made settings. These settings may include the size of the drawing (limits), the unit type ,tittle blocks,standardized texts etc.



Exiting a drawing

You can open only one drawing at a time,thus you can never exit a drawing while still remaining in AutoCad. Instead you can either exit AutoCad entirely or open a new or existing drawing without exiting AutoCad.

BASIC COMMANDS.

In Autocad you can issue commands by :-

- Choosing a tool from a toolbar.
- Choosing a command from the menu.
- Typing a command at the command line.

In this course we will use all the three ways where convenient.

Responding to command options

Many commands have a number of options from which you need to choose .

To choose an option on the command line type the letter that is capitalized in the option name usually the first letter of the option (But not always) and then press ↵

To choose a default option on the command line which appears in the angled brackets simply press the ENTER .

At this point additional options may appear or you may be prompted to select a point or an object.

DRAWING A LINE

Drawing a line using absolute coordinates

- 1 From the Draw toolbar, choose the Line flyout, then .
- 2 Enter the X and Y values of the coordinate separated by a comma (no spaces) as in the following example: 1,3
- 3 Enter a different set of X and Y values, as in the following example: 2,2

AutoCAD draws a line segment between the two points.

- 3 Enter a distance, a < sign, and an angle, as in the following example: .
- 4 25<45

AutoCAD continues the line with a second segment at the angle and distance you specified.

Drawing a line using relative coordinates

- 1 From the Draw toolbar, choose the Line flyout, then .
- 2 Specify a start point (From point).
- 3 Enter the X and Y values of the coordinate separated by a comma (no spaces), and precede the X value with an at sign (@), as in the following example: @3,5

You can also draw a line by specifying a distance and an angle, as follows:

- 1 From the Draw toolbar, choose the Line flyout, then .
- 1 Specify a start point (From point).
- 2 Enter an at sign (@), a distance, a less-than sign (<), and an angle as in the following example: @2<45

Drawing a line using an object snap mode

- 1 Draw a line and a rectangle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 From the Options menu, choose Running Object Snap.
- 4 In the Running Object Snap dialog box, choose Endpoint and press enter key .
- 5 Click near the end of the line.
- 6 From the Options menu, choose Running Object Snap.
- 7 From the Running Object Snap dialog box, choose Intersection and press enter key .
- 8 Click on the nearest corner of the rectangle.

Drawing a line using Direct Distance Entry

- 1 From the Draw toolbar, choose the Line flyout, then .
- 2 Enter 2,2.
- 3 Move your cursor any distance at a 45-degree angle.
- 4 Enter 3.

You should see a line 3-units long at a 45-degree angle, originating at 2,2.

Drawing a circle

- 1 From the Draw toolbar, choose the Circle flyout, then choose
- 2 Use your pointing device or enter X and Y values to specify a center point for the circle.
- 3 Specify a second point, enter a radius value type **D** for Diameter entry value and press enter key .

Drawing a rectangle

- 1 From the command line type **Rectang**
- 2 Use your pointing device or enter X and Y values to specify a center point for the circle.
- 3 Specify a second point and press enter key . A rectangle with the two points as the corners of the rectangle is automatically drawn .

Drawing a Polygon

- 1 From the command line type **POLYGON**
- 2 Specify the number of sides that you require
- 3 Use your pointing device or enter X and Y values to specify a center point for the polygon.
- 4 Specify whether you want your polygon inscribed or circumscribed about a circle.
- 5 Use your pointing device to specify a second point or type the radius of the circle and press enter key .

Drawing an arc

1. From the command line type **arc**
2. Use your pointing device or enter X and Y values to specify a start point for the arc
3. . Use your pointing device or enter X and Y values to specify a second point for the arc
4. Use your pointing device or enter X and Y values to specify an end point for the arc
An arc with the specified specifications is drawn.

Drawing an ellipse

1. From the command line type **ellipse**
2. Use your pointing device or enter X and Y values to specify the first axis endpoint.↵
3. Use your pointing device or enter X and Y values to specify a second point for the second axis endpoint.↵
4. Adjust your pointing device to get the ellipse you want.
5. An ellipse with the specified specifications is drawn.

Creating a point object

1. From the Draw toolbar, choose the Point flyout, then ↵
2. At the Command prompt, enter **point** then↵
3. Point: Specify a point
Points can act as nodes to which you can snap objects. The current elevation is assumed if you omit the Z coordinate.

Repeating commands

The most common way to repeat a command is to press ENTER at the Command Prompt. The last command appears again. Clicking the right Mouse Button has the same effect as pressing the ENTER key.

If you know in advance that you will be using a command several times then you may use another technique-Type **Multiple** a space then the command name. The command automatically reappears on the command line until you press escape key.

Cancelling commands

As in any windows program when you press Esc the current command is canceled. In some instances you may need to press the Esc key twice.

Undoing a command.

Most windows applications offer the **Undo** and **Redo** commands. AutoCAD is no different . AutoCAD remembers every command you enter since the drawing was opened. Thus you can undo and redo actions on your drawings.

However there are some actions that you may not be in a position to undo. Examples are Printing,saving.You cant unprint or unsave.

The U command

Each time you click undo on the standard toolbar AutoCAD undoes one command. If you undo all the commands you get the message “Everything has been undone”

The UNDO command

The **Undo** command is more complex and offers several options. It can only be typed at the command line. The several options involved are **Auto/Control/BEGIN/End/Mark/<Number>**:

Redoing a command.

The **Redo** command has the effect of redoing the last **U** or **UNDO** command. It only works on one command thus you can only use it once.

Nb. It must be used immediately after the **U** or **UNDO** command

The OOPS command

The **OOPS** command undoes the most recent command that erased an object even if you have used other commands in the meantime. This command brings back the object on the screen.

Transparent commands

Certain commands can be used within other Commands. These type of commands are called transparent Commands. Once the transparent Command is complete the original Command continues its regular operation.

THE COORDINATES SYSTEM

Specifying coordinates on the screen is one of the most fundamental tasks in autocad. Unless you know how to specify a point you can't draw anything real.

Every point on your screen is defined in terms of **X** and **Y** coordinates.

Absolute cartesian coordinates

When you type a line and enter the actual coordinates such as line from point 3,2 to 5,8 you are using absolute cartesian coordinates. Absolute coordinates are measured from 0,0.

Relative cartesian coordinates

Relative coordinates specify the X and the Y distance from the previous point. They are called relative coordinates because they have meaning relative to a point previously defined.

You tell AutoCAD that the coordinates are relative by using the @symbol

Polar Coordinates:

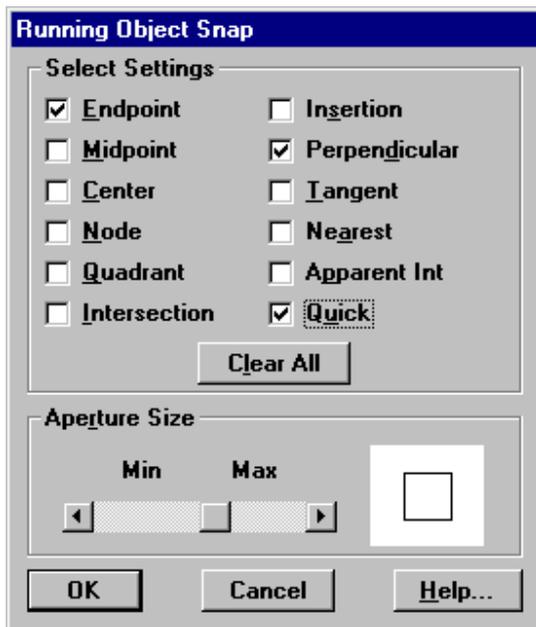
Another common situation is to know the distance and angle of a point from either 0,0 or a previous point. In this case you can use the polar coordinates, which can be either absolute or relative.

Polar Coordinates take the form of *distance<angle*.

Object Snap settings

The **SNAP** command is an alternative to tedious entry of coordinates.

Often you need to draw objects relative to the previously drawn objects. AutoCAD offers **OSNAP** (Object snap) that allows you to specify a point by snapping to a geometrically defined point on an existing object.



Snapping specific points on an object

Snapping to an endpoint

- 1 Draw a rectangle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Endpoint and press enter key .
- 6 Select the side of the rectangle closest to the cursor and press enter key .

Snapping to a midpoint

- 1 Draw a rectangle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Midpoint and press enter key .
- 6 Select the side of the rectangle closest to the cursor and press .ENTER KEY

Snapping to the center point of a circle

- 1 Draw a circle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Center and press enter key .
- 6 Select the circle and press enter key .

Snapping to a node

- 1 Draw a point using the POINT command.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Node and press enter key .
- 6 Select the point and press enter key .

Snapping to a quadrant

- 1 Draw a circle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Quadrant and press enter key .
- 6 Select the circle and press enter key .

AutoCAD snaps to the closest 0-, 90-, 180-, or 270-degree point on the circle.

Snapping to the intersection of two objects

- 1 Draw two lines that intersect.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 1 In the Running Object Snap dialog box, choose Intersection and press .ENTER KEY
- 6 Select a point near the intersection of the two lines and press .ENTER KEY

Snapping to a perpendicular point

- 1 Draw a rectangle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Perpendicular and press enter key .
- 6 Select the rectangle and press enter key .

Snapping to a tangent point

- 1 Draw a circle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Tangent and press enter key .
- 6 Select the circle and press enter key .

Snapping to the nearest point

- 1 Draw a rectangle.
- 2 From the Draw toolbar, choose the Line flyout, then .
- 3 Specify the start point (From point) of the line.
- 4 From the Options menu, choose Running Object Snap.
- 5 In the Running Object Snap dialog box, choose Nearest and press enter key .
- 6 Select the rectangle and press enter key .

EDITING DRAWINGS

Copying commands.

Four commands allow you to copy objects in very specific ways. These are mainly the **Mirror**, **Array**, **Copy** and **Offset** commands.

Mirror Command

Many drawings have symmetrical elements. Often in mechanical drawings you can create one-half or one-quarter of an object and complete it with simply by mirroring what you have drawn. To mirror select an object and then choose mirror from the modify toolbar (or type mirror at the prompt) Choose two points that will be used as mirror line.

Making a mirror image of an object

- 1 From the Draw toolbar, choose the Line flyout, then .
- 2 Draw a triangle.
- 3 From the Modify toolbar, choose the Copy flyout, then .
- 4 Select each side of the triangle and press enter key .
- 5 Specify the first point of the mirror line.
- 6 Specify the second point of the mirror line.
- 7 Press enter key to retain the original triangle.

Array command

The array command creates a rectangular or circular pattern by copying the object(s) you select as many times as you specify.

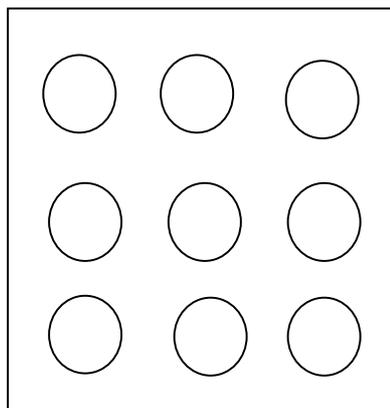
To create an array select the object or objects and choose array from the modify toolbar.

Rectangular arrays

These are arrays that are produced in rows and columns. Thus in rectangular arrays you must specify the number of columns and rows that you need.

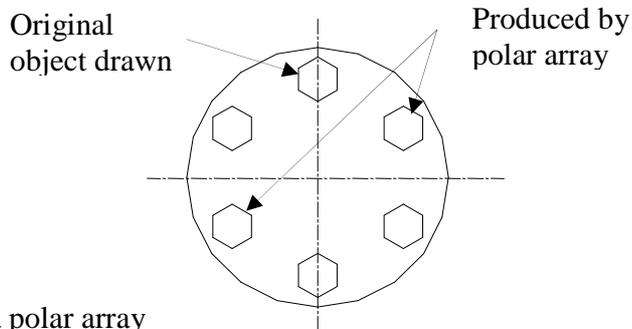
Creating a rectangular array

- 1 From the Modify toolbar, choose the Copy flyout, then .
- 2 Select an object to copy and press enter key .
- 3 Enter the number of rows.
- 4 Enter the number of columns.
- 5 Enter the distance between rows.
- 6 Enter the distance between columns.



Polar (circular) arrays

These are arrays that are produced round in a circular form. Thus in polar arrays you must specify the number of objects and the angle through which you need them spread over.



Creating a polar array

- 1 From the Modify toolbar, choose the Copy flyout, then .
- 2 Select an object to copy and press enter key .
- 3 Specify the center point of the array.
- 4 Enter the number of items.
- 5 Enter the angle that you want the array to fill.
- 6 Press enter key to rotate the objects or enter N not to rotate them.

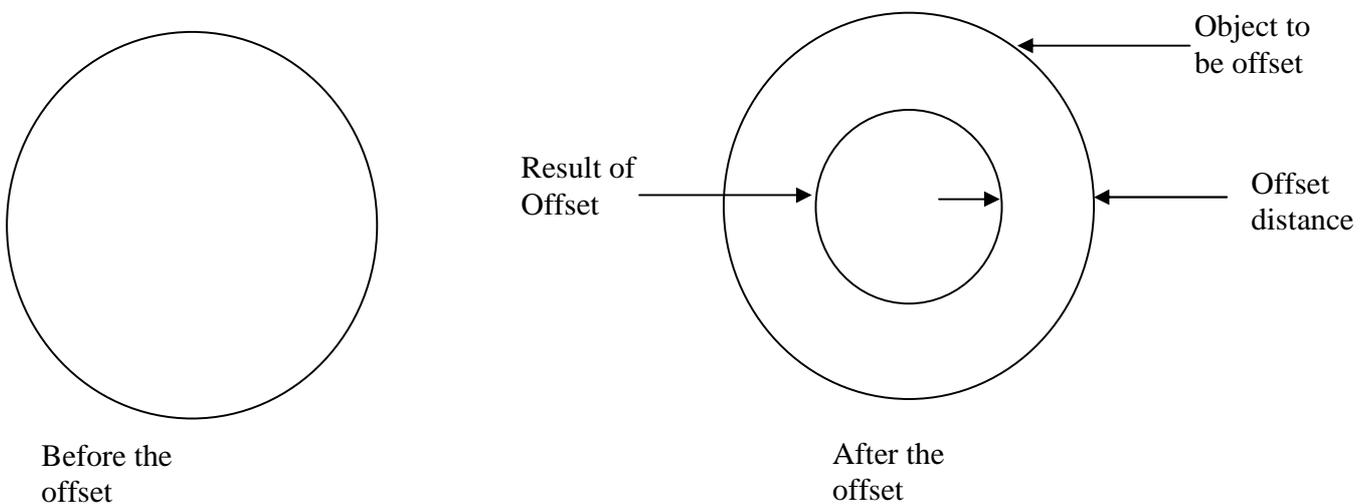
Offset command

OFFSET creates a new object at a specified distance from an existing object or through a specified point.

Type the command **offset** command

When you specify a distance, offset Creates an object at a specified distance from an existing object.

When you choose through, offset Creates an object through the point you have specified.



Copy command

The copy command duplicates a drawing.

Type the command **copy** then select an object you want to copy.

Copying an object to use again in your drawing

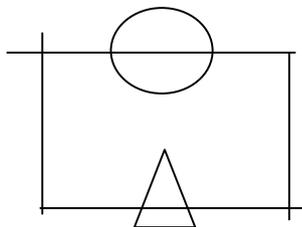
- 1 From the Modify toolbar, choose the Copy flyout, then .
- 2 Select an object and press enter key .
- 3 Specify the base point on the object.

AutoCAD copies the selected object but does not place it in your drawing until you specify a second point of displacement.

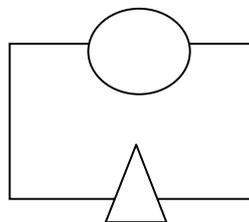
- 4 Drag your cursor across the screen.
AutoCAD copies the object and draws a rubber-band line from the specified base point to the cursor.
- 5 Drag the copied object to the Second point of displacement and click the left pointing device button.

Trimming objects

As you edit your drawing you may realise that the lines that used to meet perfectly now hang over. In this case you need to trim such objects. To trim an object you must first specify the cutting edge. When you select an object to trim you must select the object that that you want trimmed.



Objects before
the trim effect



Objects after the
trim effect

Trimming an object

- 2 From the Modify toolbar, choose Trim flyout, then .
- 3 Select an arc, circle, line, or open 2D or 3D polyline that intersects the object to trim.
- 3 Press .ENTER KEY
- 4 Select the object to trim.
- 5 Press .ENTER KEY

AutoCAD trims the second object, using the first object as the cutting edge.

Erasing an object

The Erase command Removes objects from a drawing

From the Modify menu choose Erase, while At the Command prompt, enter erase

Select objects: Use an object selection method to select the objects you want to erase.

AutoCAD removes the objects from the drawing.

Moving an object

Now and then you will be required to move objects from one position to another in your drawing. Basically the command **move** Displaces objects a specified distance in a specified direction in your drawing.

To move an object

At the Command prompt, enter **move**

Select objects: Use an object selection method to select the objects you want to move

Base point or displacement: Specify a base point.

Second point of displacement: Specify a second point or press

The two points you specify define a displacement vector that indicates how far the selected objects are to be moved and in what direction. If you press enter at the second point, the first point is interpreted as relative X,Y,Z displacement.

Rotating an object

You may require to rotate your objects to suit your needs.the command **rotate** Moves objects about a base point. To rotate an object about a point

At the Command prompt, enter rotate

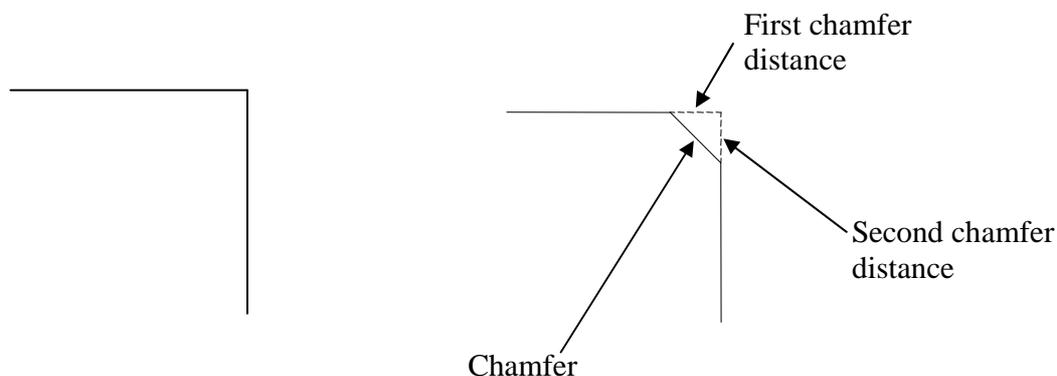
Select objects: Use an object selection method to select athe object you want rotated.

Base point: Specify a point

<Rotation angle> / Reference: Specify an angle for rotation.

Chamfer

The **chamfer** command creates corners from two nonparrallel lines. Chamfering is a two step process. First you define how you want to chamfer the corner after which the command ends. To actually chamfer you need to start the chamfer command again . AutoCAD chamfers them using the information you previously specified.



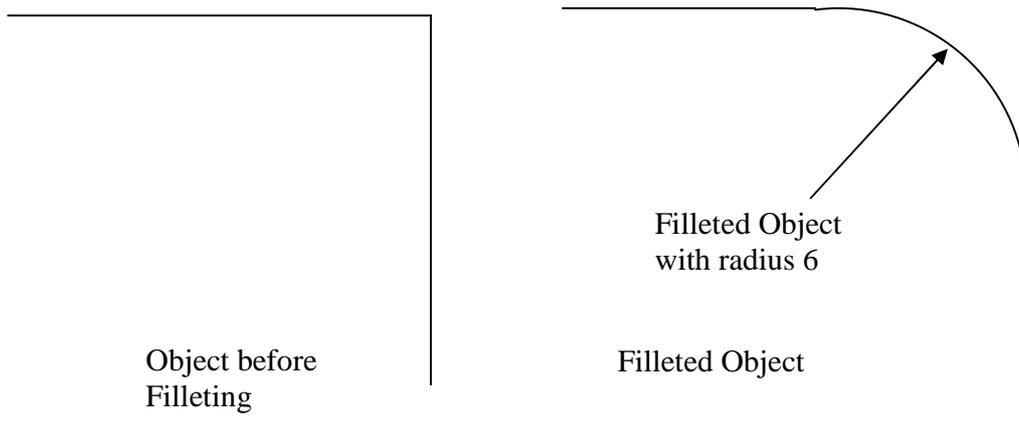
Object before
Chamfer

Object after
Chamfer

Fillet

The FILLET command rounds or fillets the edges of two arcs, circles, elliptical arcs, lines, polylines, rays, splines, or xlines with an arc of a specified radius. If the selected lines do not intersect, AutoCAD extends or trims them so that they do. FILLET also rounds or fillets the edges of solids. If both objects to be filleted are on the same layer, AutoCAD creates the fillet line on that layer. Otherwise, AutoCAD creates the fillet line on the current layer. This is also true for the fillet color and linetype.

You cannot fillet line segments that intersect outside the drawing limits when limits checking is on.



To fillet:

At the Command prompt, enter fillet

Polyline / Radius / Trim / <Select first object>: Use an object selection method or enter an option

Select first object:

Select second object: Use an object selection method to select the second object.

If you select lines or arcs, AutoCAD extends these lines or arcs until they intersect, or trims them at the intersection, keeping the selected segments if they cross. You can fillet two lines with different extrusion directions only if the Z values of the endpoints of both lines are equal in the current UCS.

Dividing objects

The **Divide** command Places evenly spaced point objects or blocks along the length or perimeter of an object

To divide an object:

At the Command prompt, enter divide

Select object to divide: Use an object selection method

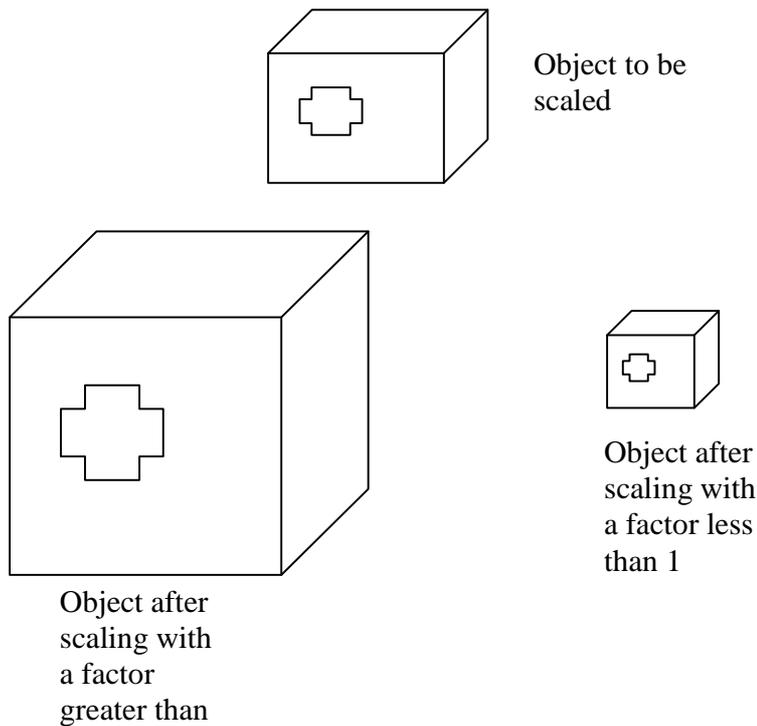
<Number of segments> / Block: Enter a value (between 2 and 32767).

For a closed polyline, the first dividing point is at its initial vertex and there are the same number of segments and dividing points. For an open polyline, the first dividing point is after the first

segment along the polyline and there is one more segment than there are dividing points. For a circle, the first dividing point is normally just right of its center, on its circumference. The point objects or blocks are placed in the Previous selection set. To select them all, enter p at the next Select objects prompt.

Scaling objects

The **Scale** command Enlarges or reduces selected objects equally in the X, Y, and Z directions



To scale an object:

At the Command prompt, enter scale

Select objects: Use an object selection method to select an object to scale

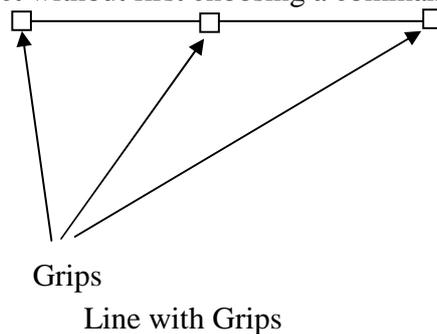
Base point: Specify a point

<Scale factor> Multiplies the dimensions of the selected objects by the specified scale. A scale factor greater than 1 enlarges the objects. A scale factor between 0 and 1 shrinks the objects.

Using Grips:

Grips offer a whole new way to edit objects without choosing commands. By using grips you can quickly stretch, move, rotate, scale and mirror objects.

When you select an object without first choosing a command the object appears highlighted with grips.



Viewing drawings

Regen

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

At the Command prompt, enter regen AutoCAD regenerates the current view and it also removes blip marks

Redraw

This command Refreshes the display of the current viewport removes marker blips and display artifacts (stray pixels) left by editing commands.

To remove marker blips:

At the Command prompt, enter redraw

AutoCAD redraws the current viewport drawing.

Pan

This command Moves the drawing display in the current viewport

PAN works in two ways. You can specify a single point, indicating the relative displacement of the drawing with respect to the screen, or (more commonly) you can specify two points, in which case AutoCAD computes the displacement from the first point to the second point. You can use PAN transparently but you cant use it transparently while in paper space

To pan:

At the Command prompt, enter pan

Displacement: Specify a point .

The point you specify indicates the amount to move the drawing or the location of the drawing to be moved.

Second point: Press Enter or specify a second point. If you press enter, AutoCAD moves the drawing by the amount you specified in the Displacement prompt. If you specify a second point, AutoCAD moves the location of the drawing to that point.

Zoom

The **Zoom** command Increases or decreases the apparent size of objects in the current viewport You can use ZOOM transparently. You can't use ZOOM transparently while in paper space or while another ZOOM, PAN, or VIEW command is in progress.

To zoom:

At the Command prompt, enter zoom

All / Center / Dynamic / Extents / Left / Previous / Vmax / Window / <Scale(X/XP)>: Enter an option or value, specify a point, or press enter

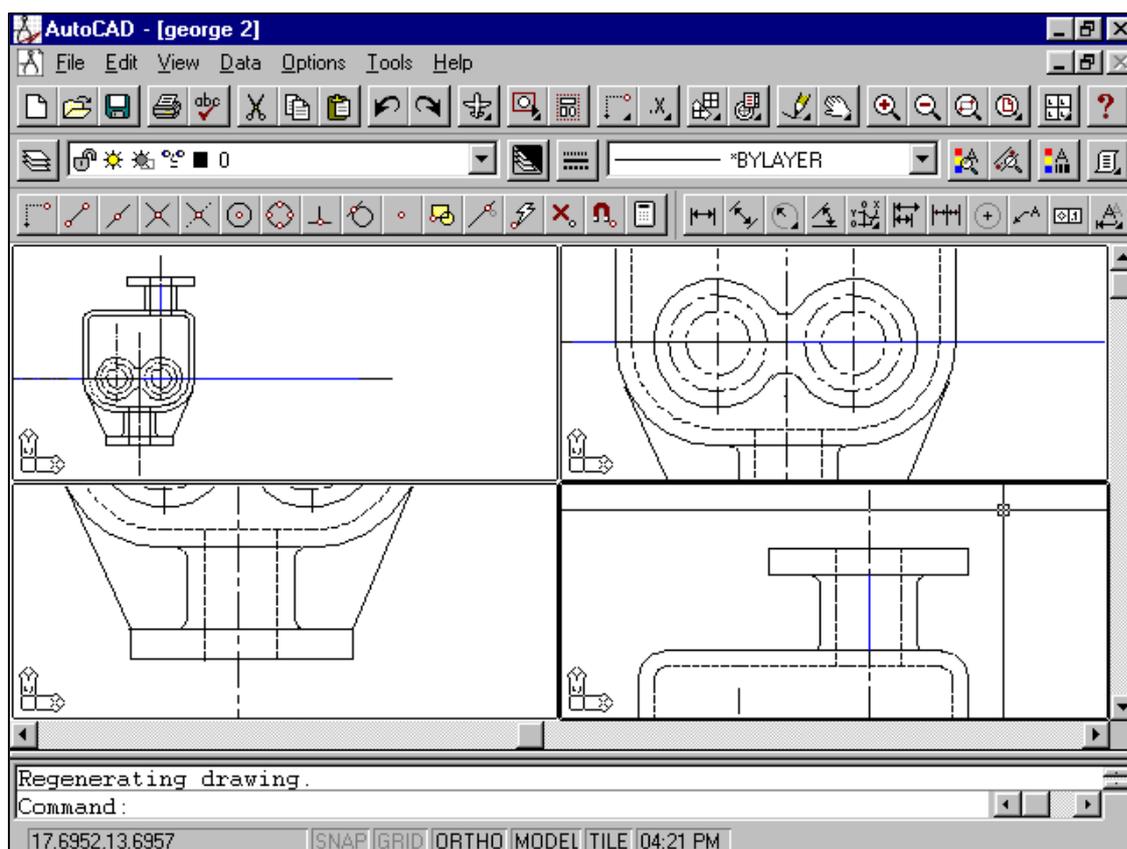
Named views:

After you have done a lot of panning and zooming in a drawing you may find that you return to the same part of your drawing again and again. In a large drawing it may take quite some time to display the part of the drawing that you want.

A view is simply a display of a drawing on your screen. It can show any part of your drawing at any magnification. Once you have the display you give the view a name and save it.

Tiled ViewPorts

Tiled viewports lets you divide up the screen into rectangular bounding boxes. You can then show a different view of your drawing in each viewport. The purpose of the viewports is to make it easier to draw.



Four tiled ViewPorts representing a single object.

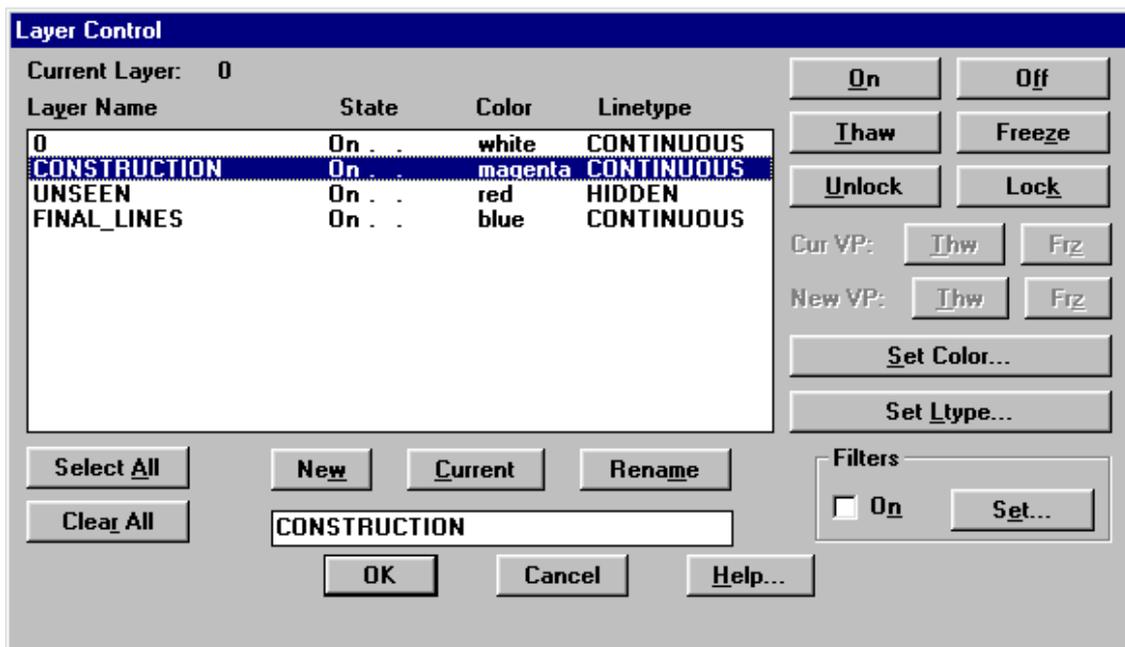
ORGANIZING DRAWINGS

Working with Layers

The best way to organize your drawing into colors and linetypes is to use layers. Layers offer powerful features that enable you to distinguish all the various elements of your drawing. Creating layers is an important part in setting up a drawing. Layers can be created and saved in your templates (Prototype drawings). Layers give you many ways to organize your drawing.

- You can assign different colors and linetypes to layers.
- You can assign the various colors to different pens in a plotter, resulting in a drawing with varying colors or line widths.
- You can control which objects are plotted.
- You can lock a layer so that objects in that layer can not be edited.

The command LAYER creates new layers, selects the current layer, sets the color and linetype for designated layers, turns layers on and off, locks or unlocks layers, freezes or thaws layers, and lists defined layers.



To use layers: At the Command prompt, enter layer

The following prompts appear:

? / Make / Set / New / On / OFF / Color / Ltype / Freeze / Thaw / LOCK / Unlock:

Enter an option you require.

The above options have the following meanings.

? -- List Layers

Displays a list of the currently defined layers, showing their names, states, color numbers, linetypes, and any externally dependent layers.

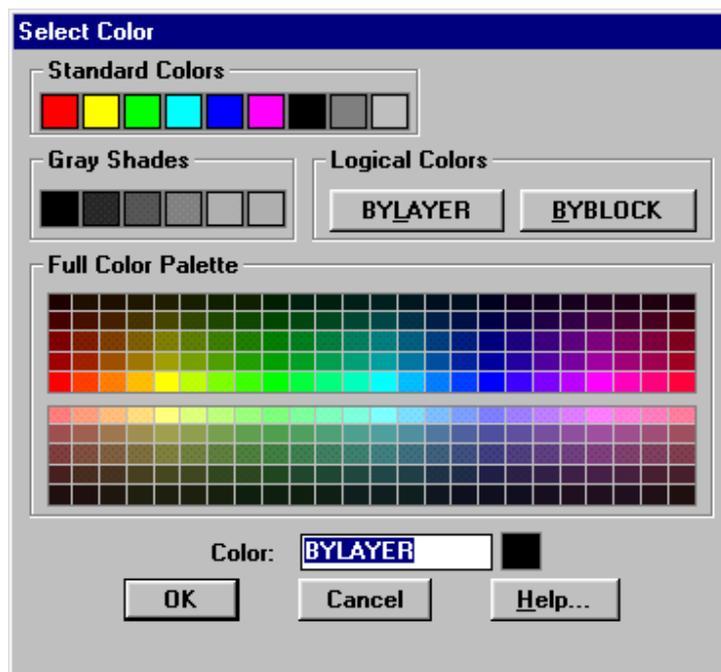
	Layer name(s) to list <*>: Enter a name list or press enter
Make	<p>Creates a layer and makes it current. New objects are placed on the current layer as they are drawn.</p> <p>New current layer <current>: Enter a name or press enter</p> <p>If no layer exists for the name you enter, AutoCAD creates a new layer by that name. That layer is turned on and is assigned color number 7 and the CONTINUOUS linetype. If the layer exists but is turned off, AutoCAD turns it on.</p>
Set	<p>Specifies a new current layer but does not create the layer if it doesn't already exist. If the layer exists but it is turned off, the Set option turns it on and makes it current.</p> <p>New current layer <current>: Enter a name</p>
New	<p>Creates one or more layers. You can create two or more layers by entering two or more names, separated by commas.</p> <p>New layer name(s): Enter a name list</p>
On/off	Turns on/off layers.
Color	<p>Changes the color associated with a layer.</p> <p>Color: Enter a color name or color number 1-255</p> <p>AutoCAD prompts for a list of layer names to which the color should be applied.</p> <p>The layers are turned on. To assign a color but turn off the layer, precede the color with a minus sign (-).</p>
Ltype	<p>Changes the linetype associated with a layer.</p> <p>Linetype (or ?) <CONTINUOUS>: Enter a currently loaded linetype name, enter ?, or press enter</p> <p>If you enter a linetype name or press , AutoCAD prompts for a list of layer names to which the linetype should be applied.</p> <p>Layer name(s) for linetype xxx <current>: Enter a name list or press enter</p> <p>If you enter ? at the Linetype (or ?) prompt, AutoCAD prompts for the linytypes to list.</p> <p>Linetype(s) to list <*>: Enter a name list or press</p>
Freeze	Freezes layers, making them invisible and excluding them from regeneration and plotting.

	Layer name(s) to Freeze: Enter a name list
Thaw	Thaws frozen layers. Layer name(s) to Thaw: Enter a name list
Lock	Locks layers, preventing editing on those layers. Layer name(s) to Lock: Enter a name list
Unlock	Unlocks selected locked layers, permitting editing on those layers. Layer name(s) to Unlock: Enter a name list

Working with color

CHANGING COLOR

You can change the color of an existing object(s). You can also change the current color. When you change the current color all future objects you draw have that color and are not drawn according to their layers assigned color.



The command **color** Sets the color for new objects.
To set the color:

At the Command prompt, enter color

New object color <current>: Enter a value (1-255), color name, byblock, or bylayer

Enter the color number in the range from 1 to 255 or enter a standard color name. The standard color names are red, yellow, green, cyan, blue, magenta, black, and white. If you enter byblock, AutoCAD draws new objects in the default color (white or black, depending on your

configuration) until they are grouped into a block. When the block is inserted in the drawing, the objects in the block inherit the current setting of the COLOR command.

Warning: If you used a mixture of color methods to draw the objects that make up a block, inserting that block or changing its color produces unpredictable results. If you enter bylayer, new objects assume the color of the layer upon which they are drawn. See the LAYER command for information on assigning a color to a layer.

Changing Linetypes

You can change the linetype of an existing object(s). You can also change the current linetype. When you change the current linetype all future objects you draw have that linetype and are not drawn according to their layers assigned linetype.

You can control an objects linetype using the linetype control drop down list on the object properties toolbar. The BYLAYER linetype simply means that the linetype of the object is taken from the linetype of the objects layer.

To change linetypes use the command **Linetype** which creates, loads, and sets linetypes The LINETYPE command defines line characteristics consisting of combinations of dashes, dots, and spaces. LINETYPE can either load linetype definitions from a library file or add new definitions to a library file.

At the Command prompt, enter linetype

? / Create / Load / Set: Enter an option.

Available Options

? Lists Linetypes .

The ? option Lists the linetypes available in a file.

File to list <current>: Enter a linetype file name or press enter.

Create

This option Creates a new linetype and stores it in a library (.lin) file.

Name of linetype to create: Enter a name

AutoCAD displays the Create or Append Linetype File dialog box. Specify the file to which you want the linetype added.

Descriptive text: Enter optional descriptive text

Enter a linetype description up to 47 characters long. The description can be a comment or a series of underscores, dots, dashes, and spaces to show a simple representation of the linetype pattern.

Enter pattern (on next line)

A, Enter pattern definition

Enter a series of numbers separated by commas. Enter positive values to specify lengths of dashes and negative values to specify lengths of spaces. Use a zero to represent a dot. The "A" in the pattern definition prompt specifies the pattern alignment used at the ends of individual lines, circles, and arcs. AutoCAD supports A-type alignment only. With A-type alignment, AutoCAD guarantees that lines and arcs start and end with a dash. The A is automatically included in the definition. If you use a text editor to create a linetype, you must enter a at the beginning of the definition. After creating a linetype, you must load it to make it accessible. You cannot create complex linetypes with the linetype command.

Load

This option Loads a linetype whose definition exists in a file. The acad.lin file contains the standard linetypes.

Linetype(s) to load: Enter a name list

AutoCAD displays the Select Linetype File dialog box. Enter or select the file name in which the linetype is stored.

Set

Sets the current linetype for subsequently drawn objects. You can control the linetype of objects individually or by layer.

New object linetype (or ?) <current>: Enter a linetype name, ?, bylayer, or byblock, or press
The linetype you enter becomes the current linetype. AutoCAD draws all new objects with this linetype, regardless of the current layer. If the linetype you request isn't loaded, AutoCAD searches for its definition in acad.lin. If the linetype is neither loaded nor in acad.lin, AutoCAD reports this deficiency and returns to the main Command prompt. Enter ? to list all loaded linetype names. If you enter bylayer, new objects inherit the linetype associated with the layer on which the object is drawn. If you enter byblock, new objects are drawn using the CONTINUOUS linetype until they are grouped into a block. Whenever you insert that block, the objects inherit the linetype of the block.

GETTING INFORMATION FROM YOUR DRAWING CALCULATIONS

Autocads standard calculator CAL

The command **Cal** Evaluates mathematical and geometric expressions.

To perform mathematical calculations:

At the Command prompt, enter cal

CAL is an on-line geometry calculator that evaluates point (vector), real, or integer expressions.

The expressions can access existing geometry using the object snap functions such as CEN, END, and INS. You can insert AutoLISP variables into the arithmetic expression, and assign the value of the expression back to an AutoLISP variable. You can use these arithmetic and vector expressions in any AutoCAD command that expects points, vectors, or numbers.

Standard Numeric Functions

CAL supports these standard numeric functions:

Function	Description
sin(angle)	Sine of the angle
cos(angle)	Cosine of the angle
tang(angle)	Tangent of the angle
asin(real)	Arcsine of the number. The number must be between -1 and 1
acos(real)	Arccosine of the number. The number must be between -1 and 1
atan(real)	Arctangent of the number
ln(real)	Natural log of the number
log(real)	Base-10 log of the number
exp(real)	Natural exponent of the number
exp10(real)	Base-10 exponent of the number
sqr(real)	Square of a number
sqrt(real)	Square root of a number. The number must be non-negative
abs(real)	Absolute value of the number
round(real)	Number rounded to the nearest integer
trunc(real)	Integer portion of the number
r2d(angle)	Angles in radians converted to degrees. For example, r2d(pi) converts the constant pi to 180 degrees
d2r(angle)	Angles in degrees converted to radians. For example, d2r(180) converts 180 degrees to radians and returns the value of the constant pi
pi	The constant pi

Calculating distances(Cal)

Autocad makes it easy to calculate the distances between two points. The command

`dist(p1,p2)` Determines the distance between two points p1 and p2. This is the same as the vector expression `abs(p1 - p2)`.

`dpl(p,p1,p2)` Determines the shortest distance between point p and the line passing through points p1 and p2.

`dpp(p,p1,p2,p3)` Determines the distance from a point p to a plane defined by three points (p1,p2,p3).

The following example returns half the distance between the centers of two selected objects:

```
dist(cen,cen)/2
```

The following example finds the distance between the point (3,2,4) and a plane you define by selecting 3 endpoints:

```
dpp([3,2,4],end, end, end)
```

Obtaining a Radius (CAL)

The `rad` function determines the radius of a selected object.

`rad` Determines the radius of a selected object. The object can be a circle, an arc, or a 2D polyline arc segment.

The following example uses `rad` with the `CIRCLE` command. The radius of the new circle is two-thirds of the radius of the selected polyline arc segment:

```
Command: circle 3P / 2P / TTR / <Center point>: cen
```

```
of Select the circle
```

```
Diameter / <Radius>: 'cal
```

```
>> Expression: 2/3*rad
```

```
>> Select circle, arc or polyline segment for RAD function: Select the circle
```

'DIST.

The command **Dist** Measures the distance and angle between two points

At the Command prompt, enter `dist`

First point: Specify a point

Second point: Specify a second point

Distance = calculated distance

Angle in XY plane = angle Angle from XY plane = angle

Delta X = change in X, Delta Y = change in Y, Delta Z = change in Z

AutoCAD reports the true 3D distance between points. The angle in the XY plane is relative to the current X axis. The angle from the XY plane is relative to the current XY plane. **DIST** assumes the current elevation value for the first or second point if the Z coordinate is omitted. The distance is displayed using the current units format.

AREA

This Command **Area** Calculates the area and perimeter of objects or of defined areas

At the Command prompt, enter `area`

<First point> / Object / Add / Subtract: Specify a point or enter an option

Using text

Single line text

This Command **Text** Creates a single line of text

AutoCAD can create text with a variety of character patterns, or fonts. These fonts can be stretched, compressed, obliqued, mirrored, or aligned in a vertical column by applying a style to the font. Text can be rotated, justified, and made any size.

To create Single line text

At the Command prompt, enter text

Justify / Style / <Start point>: Specify a point or enter an option

If TEXT was the last command entered, pressing **Enter** at the Justify/Style/<Start point> prompt skips the prompts for height and rotation angle and immediately displays the Text prompt. The text is placed directly beneath the previous line of text.

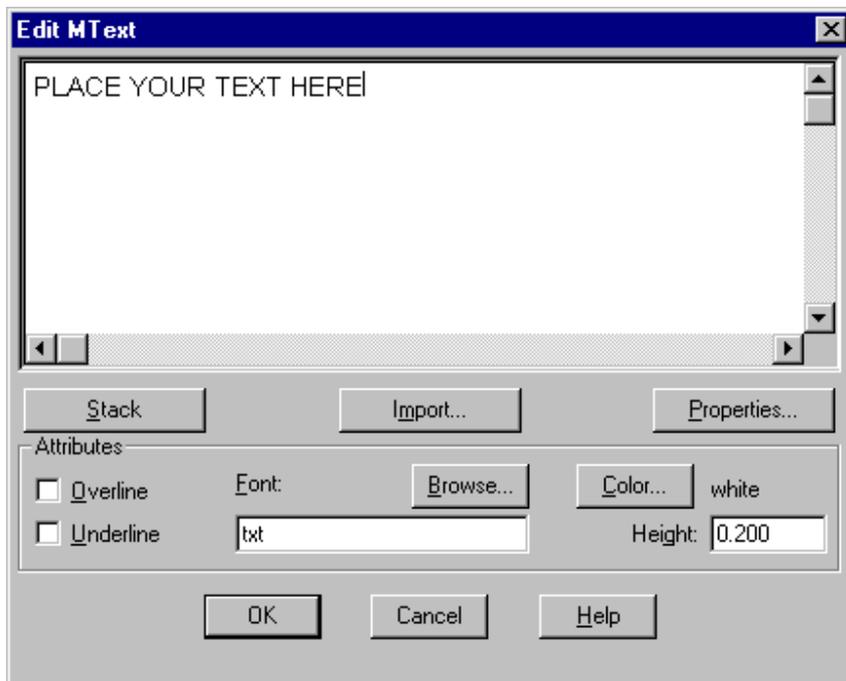
The prompts that appear are the same as those for the DTEXT command, except the TEXT command displays the Text prompt only once. For a description of each option. When you use the Insertion Object Snap mode, AutoCAD snaps to the point you specified at the Start point prompt.

Multiple text

It is also possible to create multiple text in the form of paragraphs. This Command

MTEXT Creates paragraph text

MTEXT creates paragraphs that fit within a nonprinting text boundary. You define the text boundary, which specifies the width and alignment of the paragraph. Each Mtext object is a single object, regardless of the number of lines it contains. The text boundary, although not plotted or printed, remains part of the object's framework.



To create multiple text

At the Command prompt, enter mtext

Attach / Rotation / Style / Height / Direction / <Insertion point>: Specify a point or enter an option

If you enter -mtext at the Command prompt, MTEXT displays prompts on the command line.

Spell checking

'SPELL -- Check Spelling

After you have worked you may need to check for spelling errors in your texts accompanying your drawings. The SPELL command corrects the spelling of text objects created with the TEXT, DTEXT and MTEXT commands.

The Check Spelling dialog box appears only if AutoCAD finds a dubious word in the specified text.

To Checks spelling in a drawing

At the Command prompt, enter spell

Select objects: Use an object selection method

DIMENSIONING

Dimensioning are important part of most autocad drawings. Dimensions indicate the measurements of the models you have created and are used in the manufacturing process. Dimensioning is usually done once you have completed your drawing. Dimensions are blocks and they are also associative i.e there is a connection between the object and the dimensions so that should the objects size change the dimensions also change automatically.

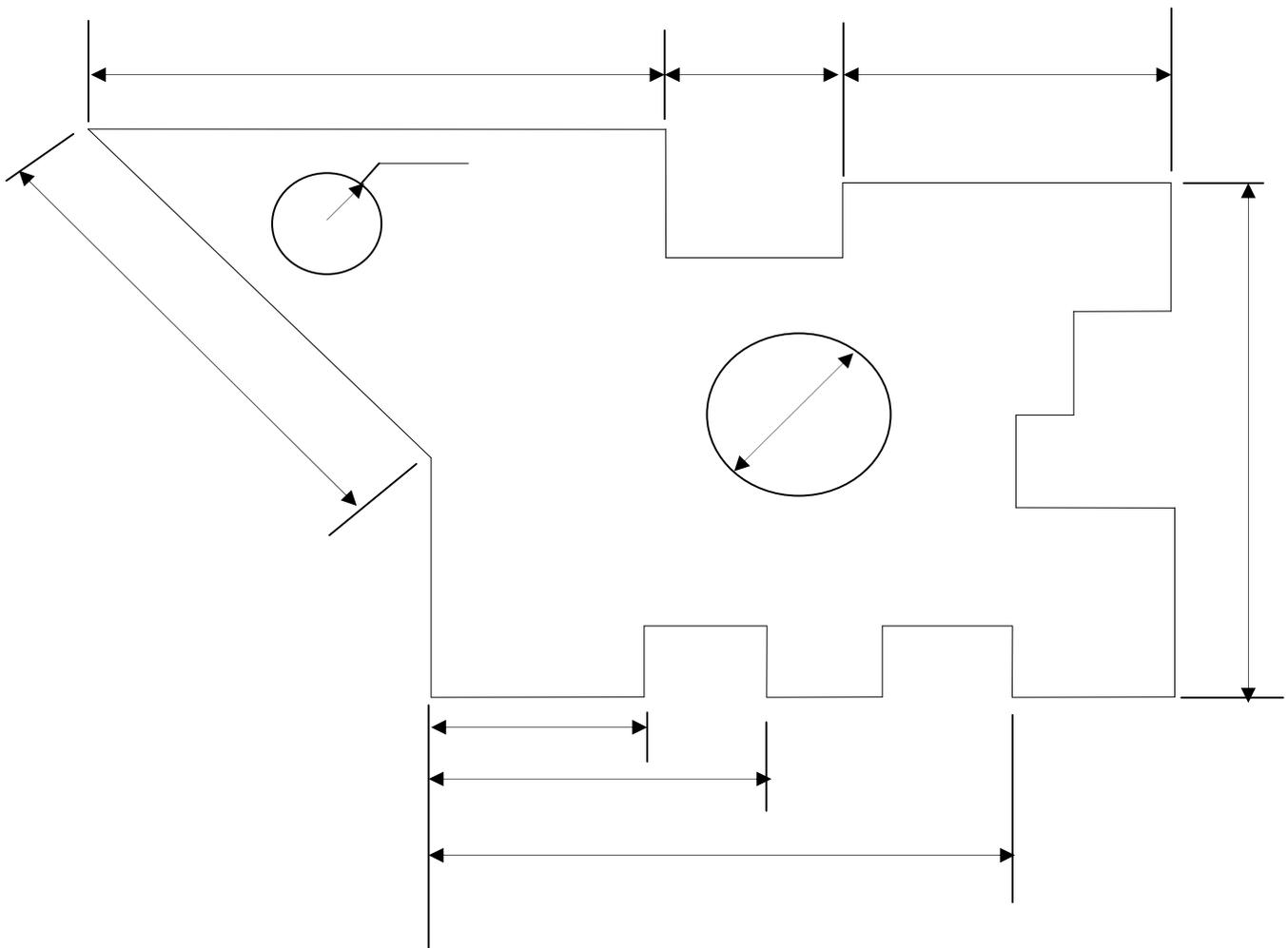
The parts of a dimension line are:

- ◆ Extension lines
- ◆ Dimension text
- ◆ Dimension line
- ◆ Arrow heads
- ◆ Definition point

Drawing Linear Dimensions

Linear dimensions are used for dimensioning lines, straight segments of apolyline or astraight segment in a block. You can also draw a linear dimmension for arcs and circles.

The command **Dimlinear** Creates linear dimensions



At the Command prompt, enter `dimlinear`

First extension line origin or RETURN to select: Specify a point or press `<space>` for automatic extension lines

Point Specification

If you specify a point for the first extension line, AutoCAD prompts for the second extension line origin.

Second extension line origin: Specify a second point

If you press `<space>` to select an object, AutoCAD automatically determines the origin points of the first and second extension lines.

For polylines and other explodable objects, only the individual line and arc segments are dimensioned. You can't select objects in a nonuniformly scaled block reference. If you select a line or an arc, its endpoints are used as the origins of the extension lines.

Drawing aligned Dimensions

When you want to dimension a linear object that is not orthogonal use an aligned dimensioning which is always parallel to the object and measures the actual length of an object as opposed to the vertical and horizontal distances that you dimension with linear dimensioning.

The command **DIMALIGNED** Creates an aligned linear dimension.

To create aligned dimensioning:

At the Command prompt, enter `dimaligned`

First extension line origin or RETURN to select: Specify a point for manual extension lines, or press `<space>` for automatic extension lines

If you select a circle, the endpoints of its diameter are used as the origin points. When the point used to select the circle is close to the north or south quadrant point, AutoCAD draws a horizontal dimension. When the point used to select the circle is close to the east or west quadrant point, AutoCAD draws a vertical dimension.

Dimension line location (Text / Angle / Horizontal / Vertical / Rotated): Specify a point (3) or enter an option

If you specify a point, AutoCAD uses the point to locate the dimension line and determine the direction to draw the extension lines. After you specify the location, AutoCAD draws the dimension and DIMLINEAR ends.

Drawing Baseline and aContinued Dimensions

Often you want to create a whole series of attached and connected dimensions

Baseline dimensions are a series of lines that start from the same point. The first extension line is the same for all the dimensions. The second dimension includes the first and an additional distance. Continued dimensions are series of dimensions that are all attached. The first extension line of the second dimensions is the second extension line of the first dimensions.

To draw a baseline dimensions or Continued dimensions, first create one linear angular, or ordinate dimensions in the regular way.

This Command **baseline** Continues a linear, angular, or ordinate dimension from the baseline of the previous or selected dimension

DIMBASELINE draws a series of related dimensions measured from the same baseline. AutoCAD uses a baseline increment value to offset each new dimension line and to avoid overlaying the previous dimension line. Baseline dimensioning is also known as parallel dimensioning.

To use baseline dimensioning

At the Command prompt, enter **dimbaseline**

Second extension line origin or RETURN to select: Specify a point, or press enter to select a base dimension

The prompts that follow depend on the type of dimension that was last created in the current session of AutoCAD: no prior dimension, linear, ordinate, or angular.

The command **DIMCONTINUE** Continues a linear, angular, or ordinate dimension from the second extension line of the previous or a selected dimension

DIMCONTINUE draws a series of related dimensions, such as several shorter dimensions, that add up to the total measurement. Continued dimensioning is also known as chain dimensioning. Note: When you create linear continued dimensions, the first extension line is suppressed and the placement of text and arrowheads might include a leader line. From the Dimensioning toolbar, choose

To use continued dimensioning.

At the Command prompt, enter **dimcontinue**

The prompts that follow depend on the type of dimension that was last created in the current session of AutoCAD: no prior dimension, linear, ordinate, or angular. If the previous dimension was a linear, angular, or ordinate dimension, AutoCAD uses the origin of that dimension's second extension line for the origin of the next dimension's first extension line.

DIMDIAMETER

This Command **Dimdiameter** Creates diameter dimensions for circles and arcs

DIMDIAMETER draws different types of diameter dimensions depending on the size of the circle or arc, the options in the DDIM Format dialog box, and the position of the cursor.

To dimension the diameter of the circle:

At the Command prompt, enter **dimdiameter**

Select arc or circle: Select an arc or a circle

The position of the cursor determines the location of the dimension line. As you move the cursor, the dimension moves around or inside the circle or arc. For horizontal dimension text, if the angle of the diameter line is greater than 15 degrees from horizontal and is outside the circle or arc, AutoCAD draws a hook line, also called a landing or dogleg, one arrowhead long, next to the dimension text, as shown in the first two illustrations.

Different types of diameter dimensions

AutoCAD measures the diameter and displays the text with an \varnothing symbol in front of it. The next prompt is Dimension line location (Text / Angle): Specify a point or enter an option. If you specify a point, AutoCAD uses the point to locate the dimension line. After you specify the location, DIMDIAMETER ends.

Center Marks and Center Lines

You control center marks or lines with the options in the DDIM Geometry dialog box. AutoCAD doesn't draw a center mark or line when a dimension line is drawn inside the arc or circle. The value of center marks and lines is specified under Center in the DDIM Geometry dialog box. AutoCAD stores the value in the DIMCEN dimensioning system variable.

DIMCENTER

This command Creates the center mark or the center lines of circles and arcs

At the Command prompt, enter dimcenter

Select arc or circle: Use an object selection method

You can choose between center marks and center lines and specify their size in the Geometry dialog box of the DDIM command. To do this at the Command prompt, use the DIMCEN dimensioning system variable.

DIMEDIT

Edits dimensions

DIMEDIT affects dimension text and extension lines. The Home, New and Rotate options affect dimension text. The Oblique option affects extension lines. DIMEDIT can operate on one or more dimension objects at a time.

At the Command prompt, enter dimedit

Dimension Edit (Home / New / Rotate / Oblique) <Home>: Enter an option or press enter

LEADER

Leader are lines pointing to objects. At the end of the leader you place any text that you want. Use leaders to label objects or provide explanatory text.

A leader line is an object that can be composed of an arrowhead attached to splines or straight line segments. In some cases, a short horizontal line, called a hook line, dogleg, or landing, connects text and feature control frames to the leader line. You can use various dimensioning system variables to format the leader line, such as placing the text above the hook line. .

LEADER creates complex leader lines. DIMDIAMETER and DIMRADIUS create simple automatic leader lines for circles and arcs. The command **Leader** Creates a line that connects annotation to a feature

To use leaders :

At the Command prompt, enter leader

From point: Specify a point or use an Object Snap mode to attach the leader line to an object

To point: Specify a point

AutoCAD draws a leader line segment to the point specified and continues to prompt for other points and options.

COMPLEX OBJECTS:

Autocad offers a number of complex objects that help you create accurate professional drawings. Polylines are single objects that can combine line segments and arcs. Splines are mathematically controlled curves based on points you specify. Hatches create a solid or patterned fill. Multilines are sets of parallel lines. Sketching is a way of creating free hand drawings.

Polylines.

. Polylines are single objects that can combine line segments and arcs they are unique in that they can allow you to make thick lines and arcs.

PLINE

This command Creates two-dimensional polylines

At the Command prompt, enter pline

From point: Specify a point.

Current line-width is <current>

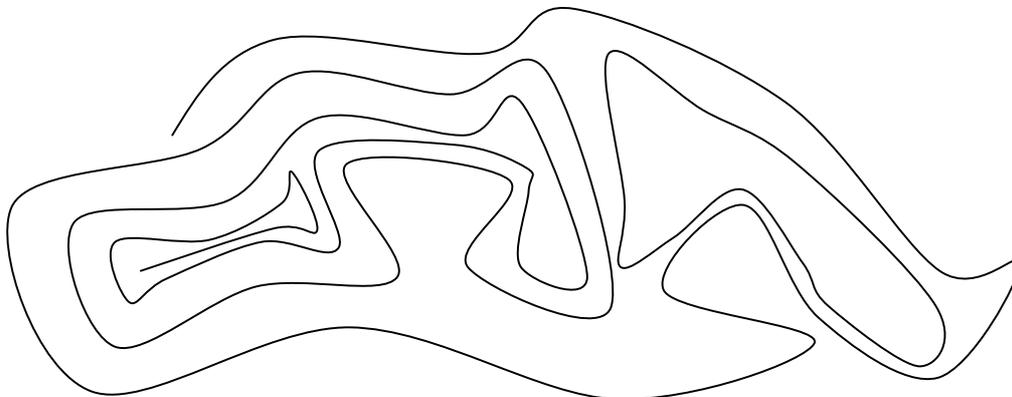
Arc / Close / Halfwidth / Length / Undo / Width / <Endpoint of line>: Specify a second point or enter an option

Splines

SPLINE

This command Creates a quadratic or cubic spline (NURBS) curve

SPLINE fits a smooth curve to a sequence of points within a specified tolerance.



To use spline:

At the Command prompt, enter spline

Object / <Enter first point>: Specify a point or enter o

Enter First Point

Creates a spline using points you specify.

Enter point: Specify a point

Enter points until you have defined the spline curve. After you have entered two points, AutoCAD displays the following prompt:

Close / Fit Tolerance / <Enter point>: Specify a point, enter an option, or press enter

Hatching an object.

Hatch patterns are blocks and are associative. Autocad stores hatch pattern definitions in the AutoCad.pat file

HATCH

HATCH fills the specified hatch boundary with a nonassociative hatch. A nonassociative hatch is not updated when its boundaries are modified. A hatch boundary consists of an object or objects that completely enclose an area. If the boundary is made up of multiple objects, their endpoints must coincide for the hatch to be created properly. You can also define a polyline hatch boundary with the direct hatch option. Unless otherwise specified, HATCH combines the lines that make up the hatch into a block.

At the Command prompt, enter hatch
Pattern (? or name / U,style) <current>: Enter a predefined pattern name, enter u
, enter ?, or press

SKETCH

This command lets you Create a series of freehand line segments and drawings
Drawing with the SKETCH command controls a screen-based pen with a pointing device.
SKETCH is useful for entering map outlines, signatures, or other freehand drawings. Sketched lines are not added to the drawing until they are recorded. The standard button menu is disabled while SKETCH is in progress.



At the Command prompt, enter sketch
Record increment <current>: Specify a distance or press
The record increment value defines the length of the line segments. The pointer must be moved a distance greater than the increment value to generate a line.
AutoCAD captures sketching as a series of independent lines. Setting the SKPOLY system variable to a nonzero value produces a polyline for each contiguous sequence of sketched lines rather than multiple line objects.

Sketch. Pen eXit Quit Record Erase Connect . (period) Enter an option, or press a pointer button.

BLOCKS:

A common drawing task is placing the same group of objects several times in a drawing. An architect needs to place windows and doors many times in the layout plan of a house. An electrical engineer places electrical symbols in a drawing over and over. A mechanical model may include many nuts ,bolts ,surface finish symbols etc. blocks are group of objects that you create ,save and name so that you can insert them in your drawing whenever you need them. A block is one object regardless of the number of individual objects used to create it.once an object is made a block its grouped automatically. Thus it can be ungrouped by exploding it.

BLOCK

This command Creates a block definition from a set of objects

At the Command prompt, enter block

Block name (or ?): Enter a name or ?

Setting up your page for plotting in autoCAD 2000

AutoCAD 2000 allows you to set up many layouts for plotting the same drawing. This is more convenient for different drawing requirements.

To create a layout, in the insert menu, choose layout

Layout-

- New layout
- Layout from template
- Layout wizard

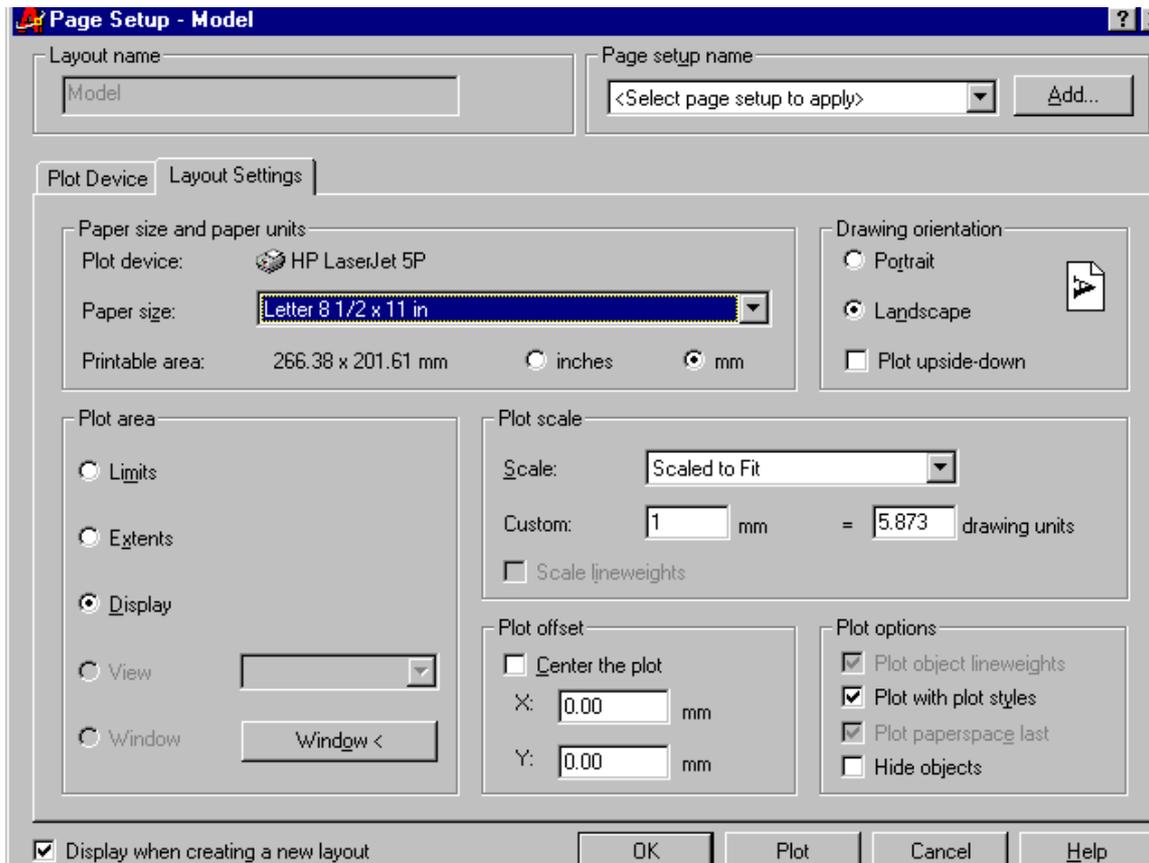
Chose new layout to create a new layout.

Chose from template to use an already designed layout from AutoCAD.

A layout wizard guides you through the processes of creating a page layout.

After creating a layout, configure its setup. In the file menu chose page setup or right click the layout bar and choose page setup.

The following dialog box comes up.



Paper size-choose the paper size you want.

Plot scale-This shows the ratio of the drawing to the real object on the ground.

Plot offset-Determines the beginning of the drawing from the paper margin.

Plot area- chose layout, extends or display.

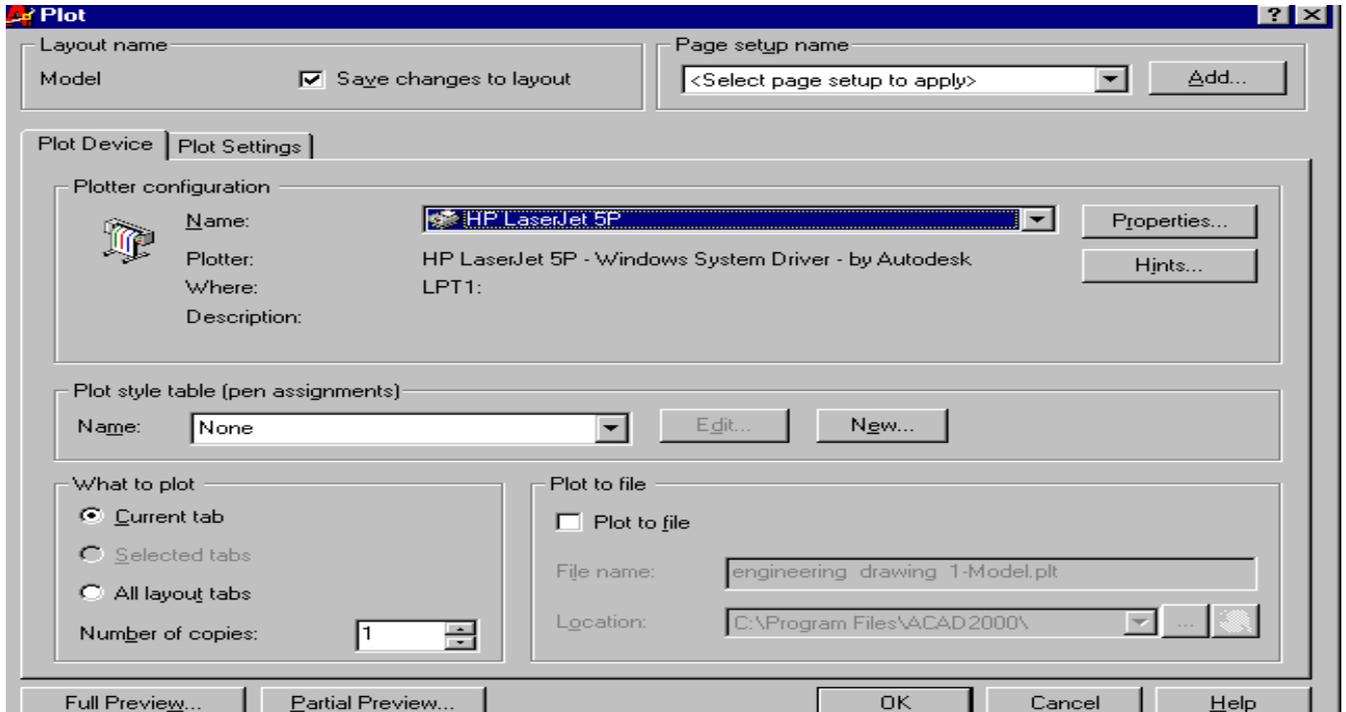
You are now ready to plot.

Plotting with autoCAD2000

1. Before plotting you need to install a plotter on your computer.
 - In the file menu choose plotter manager.
 - Follow the plotter manager wizard to install a plotter on your computer.

2. Use plot style manager to assign different styles to your drawing. Several plot styles have been designed. You can change the settings.
 - In file menu choose plot style manager.
 - Different plot styles come up.
 - Double click a style to view the pattern layout.
 - After making necessary corrections you are ready to plot.

3. In the file menu, choose plot.
 - A plot dialog box appears, allowing you to change the plot configuration.



- In the plot style table, under plot device menu, choose the plot style you configured above. Choose edit to make any changes to the style.
- Plot settings allow you change settings of your paper, scale, plot area (layout or chose your own extends).

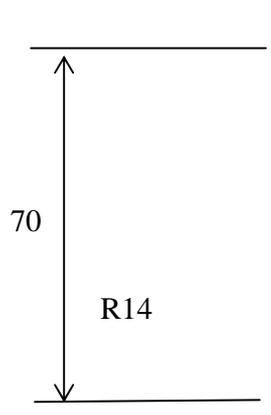
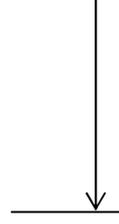
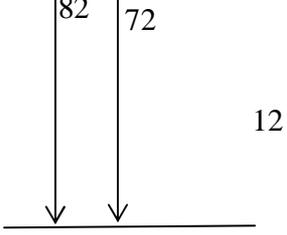
EXERCISES:

Drawing 1

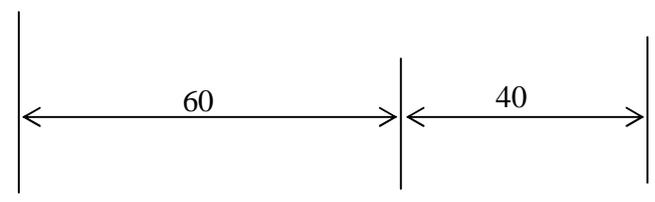
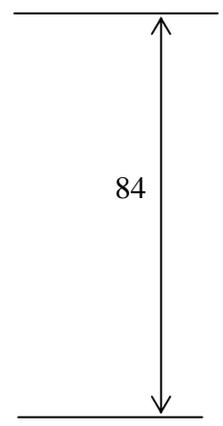
Drawing 2

Drawing 3

Drawing 4



5



4 Holes ϕ 12

All un-dimensioned radii 5
All dimensions in mm

Drawing 5

Bracket Cast Iron

Drawing 6

Drawing 7

Drawing 8

Drawing 11

Drawing 12

Drawing 13

Drawing 14

Drawing 15

Drawing 16

Drawing 17

Drawing 18

Drawing 20

Drawing 19

Drawing 21

Drawing 22

Personal Notes:

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