



# **AutoCAD Training Manual**

## **2D Level 1**



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AutoCAD 2D Level I

## AutoCAD Level 1 2D

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## Lesson 1

### Basic Information

AutoCAD, created by Autodesk, is the first most widely used technical drawing program anywhere. AutoCAD according to Autodesk stands for Computer – aided design, but can also stand for Computer – aided drafting or drawing.

The first version of AutoCAD came out in 1982, running under DOS. So it became the first to run on Desktop computer.


AutoCAD is the most flexible drafting program available, applicable to all fields. AutoCAD has unequalled features and capabilities, including 3D surface and modeling and visualization, access to external databases, intelligent dimensioning, importing and exporting of other file formats, internet support, and much more.

These are major disciplines that use AutoCAD:

- Architectural (also called AEC for architectural, engineering, and construction).
- Mechanical
- GIS (Geographic Information System)
- Facilities management
- Electrical/ Electronic
- Multimedia

There are many other lesser-known uses, such as pattern making in garment industry, sign making etc.

### Starting AutoCAD

Launching AutoCAD is very simple. Just double-click the  shortcut or start the application by clicking,

Start  Program  Autodesk  AutoCAD  AutoCAD.

### Using the wizard

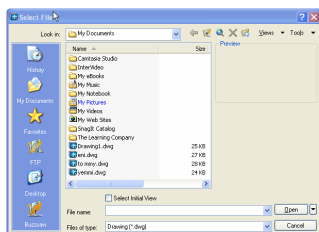
- Start from scratch enables you to choose English / Metric and create a drawing with just the minimum default settings.
- Wizards enable you to choose either Quick or advanced Setup to guide you through the process of setting up a drawing before you start.

You can also create a new drawing from within AutoCAD. Choose **File** **→** **New** or click the New button on the standard toolbar.

**new**

**File**  **New...**

When you invoke this command AutoCAD displays the Select Template dialog box.

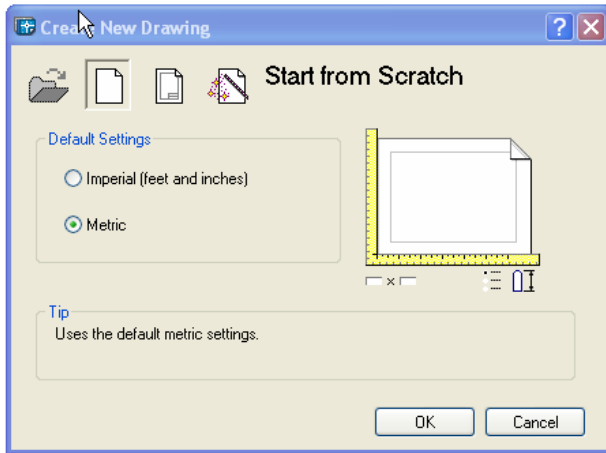



A drawing template file contains standard settings. You can select one of the template files supplied, or create your own template files. Drawing template files have a **.dwt** file extension. By default, drawing template files are stored in the template folder.

You can create a new drawing with Metric or Imperial settings by clicking the Open button in the right corner of the Select Template dialog box.

The behavior of the NEW command is determined by the Startup setting on the System tab of the Options dialog box.

By default, **Do not show a startup dialog option** is selected in the system tab of the options dialog box. When **Show Startup dialog box** option is selected, AutoCAD displays the Start up dialog box as shown below.



The first icon  in this dialog box is disabled, only the other three icons are used to create a new drawing.



### Start From Scratch

Starts an empty drawing using default imperial or metric settings. AutoCAD stores this setting in the MEASUREINIT system variable. You can change the measurement system for a given drawing by using the MEASUREMENT system variable.

#### Imperial

Starts a new drawing based on the Imperial measurement system. The default drawing boundary (the drawing limits) is 12 × 9 inches.

#### Metric

Starts a new drawing based on the Metric measurement system. The default drawing boundary (the drawing limits) is 429 × 297 millimeters.



Use a Template

Starts a drawing based on a drawing template file.



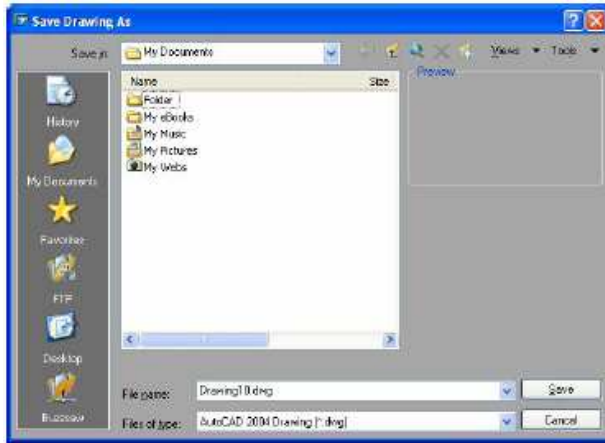
Use a Wizard

Sets up a drawing using a step-by-step guide. You can choose from two wizards: Quick Setup and Advanced Setup.

### File Management Commands



**File**  **Save/Save As...** If the drawing is named, AutoCAD displays the Save Drawing As standard file selection dialog box. If you enter a different file name, AutoCAD saves a copy of the drawing under that name also.



If the drawing is unnamed, AutoCAD displays the Save Drawing As standard file selection dialog box. Enter a file name to both name and save the drawing.

**Note:**

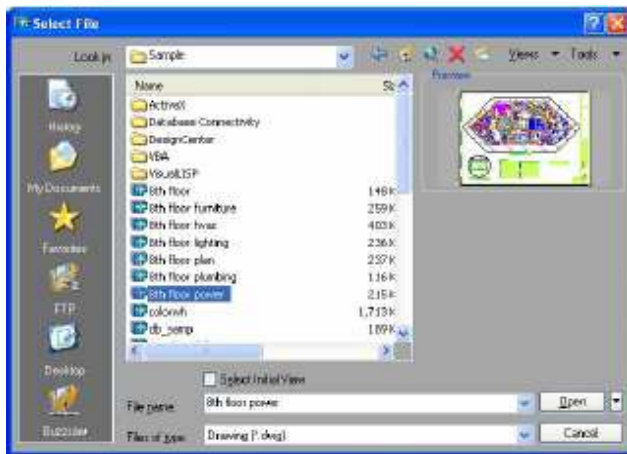
*SAVE is available only from the command line. The Save option on the File menu or on the Standard Toolbar is QSAVE. If the drawing is named, QSAVE saves the drawing without displaying the Save Drawing As dialog box. If the drawing is unnamed, AutoCAD displays the Save Drawing As dialog box. Enter a filename to both name and save the drawing.*

*If the drawing is read-only, use the SAVEAS command to save the changed file under a different name.*

*If you want to change an entry in an input box, you can double click to highlight the whole entry, and then replace the entry by simply typing in a new one. If you just want to change a part of the entry, click on the input box and then use the cursor keys to move the vertical bar cursor to the exact character you want to change. You can use the Backspace key to delete characters.*

**File Selections and Navigations**

Several AutoCAD commands display standard file selection dialog boxes. In this dialog box you can navigate through local and network drives and through FTP sites and Web folders to select files. While each dialog box may vary slightly, following is a list of the possible options.



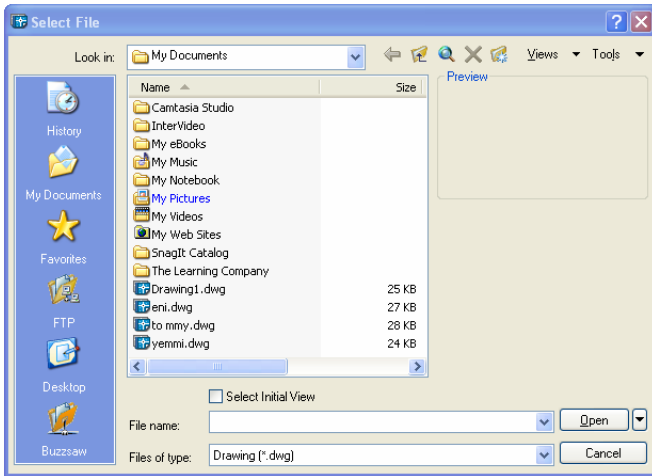
**QNEW**

QNEW starts a new drawing from the current default drawing template file and folder path specified in the Options dialog box on the Files tab. You can set the default drawing template file to any drawing template file or to None.

When a default drawing template file is set to None or is not specified, QNEW displays the Select Template File dialog box (a standard file selection dialog box).



**File**  **Open...** AutoCAD displays the Select File dialog box.



You can open and load a portion of a drawing, including geometry on a specific view or layer. In the Select File dialog box, click the arrow next to Open and choose Partial Open or Partial Open Read-Only to display the Partial Open dialog box.

When FILEDIA is set to 0 (zero), OPEN displays a prompt on the command line.

Enter name of drawing to open <D:\acad\MENUTREE.dwg>:

All options in this dialog box are based on the standard file selection dialog box discussed above.

### close

**File**  **Close**

AutoCAD closes the current drawing; if there have been no changes since the drawing was last saved. If you have modified the drawing, AutoCAD prompts you to save or discard the changes.

You can close a file that has been opened in Read-only mode only when you have not made any changes or willing to discard changes. To save changes to a read-only file, you must use the SAVEAS command.

### closeall

**Window**  **Close All**

When you invoke this command, a message box is displayed for each unsaved drawing, in which you can save any changes to the drawing before closing it.

### quit/exit

**File**  **Quit/Exit**

Quits AutoCAD if there have been no changes in all opened drawing since the drawings were last saved. If the drawing has been modified, AutoCAD displays the Drawing Modification dialog box to prompt you to save or discard the changes before quitting.

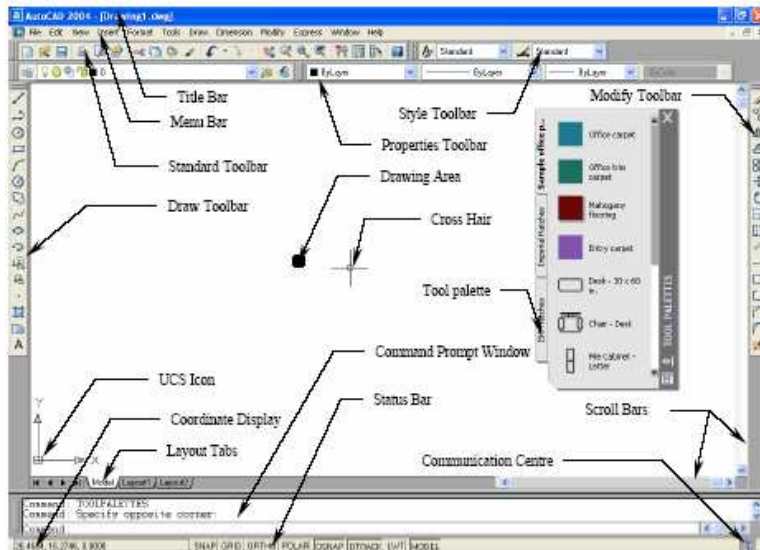
## Introduction to AutoCAD Environment

### The First Look at AutoCAD

Launching AutoCAD is very simple. Just double-click the AutoCAD shortcut or start the application by clicking, Start  Program  Autodesk  AutoCAD  AutoCAD; whichever version you are using from AutoCAD upwards. After a few moments, you could see the AutoCAD interface as shown in the following figure.

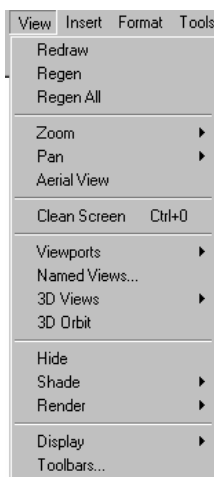
Take a look at it and familiarize with the various components of the Graphical User Interface (GUI) of AutoCAD.





### The Pull-down Menus

The Pull-Down menus available on the menu bar offer a quicker way to access the commands and functions that are the heart of AutoCAD.



The pull-down menu options perform three basic functions:

- Display additional menu choices (options with □ symbol), which are called as cascading menu.
- Display a dialog box that contains settings, you can change (options with ellipsis ... marks).
- Issue a command that requires keyboard or drawing input.

As you select commands and options, AutoCAD flashes a single line help in the status bar.

### Toolbars and Icons

Just as the pull-down menus, the commands in the toolbars do the nitty-gritty work of creating new objects and editing the existing ones. These icons are grouped by action type. For example, the Draw Toolbar contains tools needed to create new objects. The Modify Toolbar contains functions that modify existing objects in a drawing and so on.

The icon buttons in the toolbars perform three types of actions,

- Display further options known as flyouts (buttons with ↘ symbol at the bottom right corner)
- Open dialog boxes
- Issue commands that require keyboard or cursor input.

### **The drawing area**

The blank area in the middle of the screen, the graphics window, is where you draw. You can think of this area as a sheet of drafting paper, except that this piece of paper can be any size – even the size of a huge factory!

At the bottom of the drawing area is a tab labeled Model. You draw on this tab. You use the layout tab to lay your drawing for plotting.

### **The UCS Icon**

The symbol with two arrows at the bottom – left corner of the drawing area in fig. 1 is called UCS (User Coordinate System) icon. The arrows point to the directions of the X and Y axis to help you keep your bearings. You can change the look of this icon (to be discussed later).

### **The crosshairs**

Notice the two intersecting lines with a small box at their intersection. The small box is called pickbox because it helps you to pick objects. The lines are called crosshairs. They show the location of the mouse cursor in relation to other objects in your drawing. As you move your mouse around, the pickbox and the crosshair move with your mouse. At the bottom of your screen, at the left end of the status bar, (described later), you can see the X, Y coordinates (discussed later) changing as you move the mouse.

### **The Menus and Toolbars**

At the top of your screen is the titles bar, and directly beneath the titles bar is a menu bar. Below that are two rows of toolbars. In addition, your screen has two more toolbars, the draw and modifies toolbars, which are probably docked at the left side of the screen as shown in fig. 1. Use the menus and toolbars together to give AutoCAD commands to draw, edit, get information, and so on.

Because you can customize the menus and the toolbars to suit your needs, your screen may appear somewhat differently. AutoCAD provides many more toolbars that you can display when you need them.

Some examples of the toolbars are: Dimension, Solids, Render, and Web. We learn about all these later.

### **The Command Line**

At the bottom of the screen, you can see a separate window showing approximately three lines of text. (It can be changed to show as many lines as you like). All command can be executed by typing them on the command line. Even when you use menu item or toolbar button to execute a command, you may need to look at the command line to see how AutoCAD responds. Often, AutoCAD provides options that must be typed in from the keyboard. Also, anything you typed appears on the command line. E.g. when you type in coordinates specifying points, they appear on the command line. To see more of the command line, press F2 to open the AutoCAD Text Window. You can scroll back through the previous commands. Press F2 again to close the window.

### **The Status Bar**

At the very bottom of the screen is the status bar as shown in the above figure. at the left are the X, Y coordinates. As you move your mouse around, these coordinates change. (If they don't move, click them and move your mouse again). The status bar also sports several buttons that you will learn about later.

### ***Communicating with AutoCAD***

AutoCAD is a perfect servant: it does everything you tell it to and no more. You can communicate with AutoCAD using the pull-down menus, screen menus, command line and the buttons on the toolbars.

A command is a single-word instruction from the user to perform the required task. When you invoke a command, AutoCAD responds by presenting messages in the command prompt area, or by displaying a dialog box.

The messages in the command prompt area often tell you, what to do next, or they offer a lot of options.

A dialog box is like a form, you fill out on the computer screen. It lets you to adjust settings or make selections from a set of options pertaining to a command.

## Lesson 2

### Drawing Lines



Line

Draw □Line

**L**INE command allows you to create a line where the end points are specified by two-dimensional or three-dimensional coordinates.

AutoCAD prompts for points to draw continuous line segments. You may specify the points randomly with the mouse or by any of the point fixing methods, explained a little later.


Command: `_line`

Specify first point: *Use one of the point fixing methods*

Specify next point or [Undo]: *Use one of the point fixing methods*

Specify next point or [Undo]: *Use one of the point fixing methods*

Specify next point or [Close/Undo]: *Use one of the point fixing methods*

Specify next point or [Close/Undo]: *Use one of the point fixing methods or press  to terminate the command.*

### Undo

You can undo the most recently drawn line segment by entering the option U in the Specify next point: prompt.

You can undo all the line segments drawn. If everything is undone, AutoCAD flashes the message All segments already undone, and starts the LINE command afresh.

### Close

Close option joins the very last point with the very first point in a series of line segments that you had drawn using the line command, thus forming a closed shape.

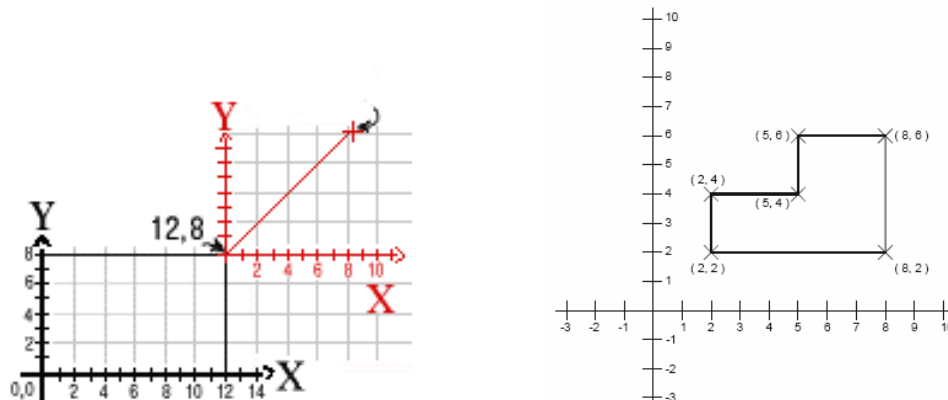
### Co-ordinate Systems in AutoCAD

All drawings are superimposed on an invisible grid, or co-ordinate system, with a horizontal X-axis and a vertical Y-axis.

You can establish *grid* and *snap* settings that match the units of the co-ordinate system or some multiple or fraction of it.

### The Co – ordinate System

Everything that you draw in AutoCAD is **exact**. It will be more accurate than you will ever need it to be. All objects drawn on the screen are placed there based on a simple X,Y co-ordinate system. In AutoCAD this is known as the World Co-ordinate System (WCS). You must understand this to know how to put things where you want them. (*3-D work has an added axis, the Z-axis, but this is not covered in this lesson.*) Below is a diagram showing you how this system works.



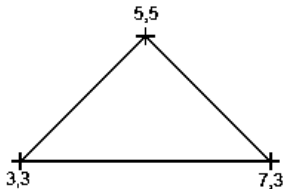
As you draw, you can enter co-ordinates to locate a point. For example, you can draw a line by starting it at the 2,2 location in the co-ordinate system and end it at the 8,2 location, as shown in the above figure. This is called the *Absolute Co-ordinate System*. The X value of a Cartesian coordinate

specifies the horizontal distance, and the Y value specifies the vertical distance. The origin point (0,0) indicates where the two axes intersect. Absolute co-ordinate values are based on the origin (0,0). Relative co-ordinate values are based on the very last point entered. They are useful for finding a series of points that are at a known distance apart. Various methods of entering 2D coordinates in AutoCAD 2004 are discussed below.

### Absolute Co-ordinate System (x, y)

To enter an absolute coordinate, specify a point by entering its X and Y values in the format **X, Y**. Use absolute coordinates when you know the precise X and Y values of the point from the origin. The following sequence of coordinates draw a triangle, as shown below.

Command: `_line` Specify first point: `3,3`  
 Specify next point or [Undo]: `7,3`  
 Specify next point or [Undo]: `5,5`  
 Specify next point or [Close/Undo]: `c`

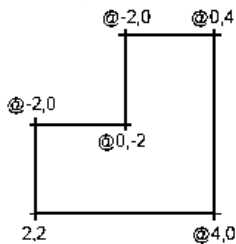


### Relative Rectangular Co-ordinate System (@ x distance, y distance)

Use relative coordinates when you know the position of a point with respect to the previous point. The relative rectangular coordinate is represented in the following format:

#### @ x-displacement, y-displacement.

Command: `_line` Specify first point: `2,2`  
 Specify next point or [Undo]: `@4,0`  
 Specify next point or [Undo]: `@0,4`  
 Specify next point or [Close/Undo]: `@-2,0`  
 Specify next point or [Close/Undo]: `@0,-2`  
 Specify next point or [Close/Undo]: `@-2,0`  
 Specify next point or [Close/Undo]: `c`



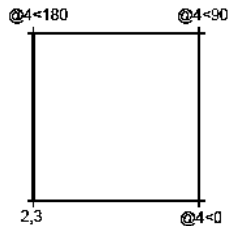
Mostly, an absolute co-ordinate is given at the Specify first point:. Distance is measured +ve towards right and up and it is measured –ve towards left and down.

### Relative Polar Co-ordinate System (@ distance < angle)

Polar co-ordinate system uses a distance and an angle with reference to the previous point to locate a point. Angles are measured in anti-clock direction, taking 0° towards right.

The relative polar coordinate is represented in the following format:

**@ distance < angle.** Command: `_line` Specify first point: `2,3`  
 Specify next point or [Undo]: `@4<0`  
 Specify next point or [Undo]: `@4<90`  
 Specify next point or [Close/Undo]: `@4<180`  
 Specify next point or [Close/Undo]: `c`



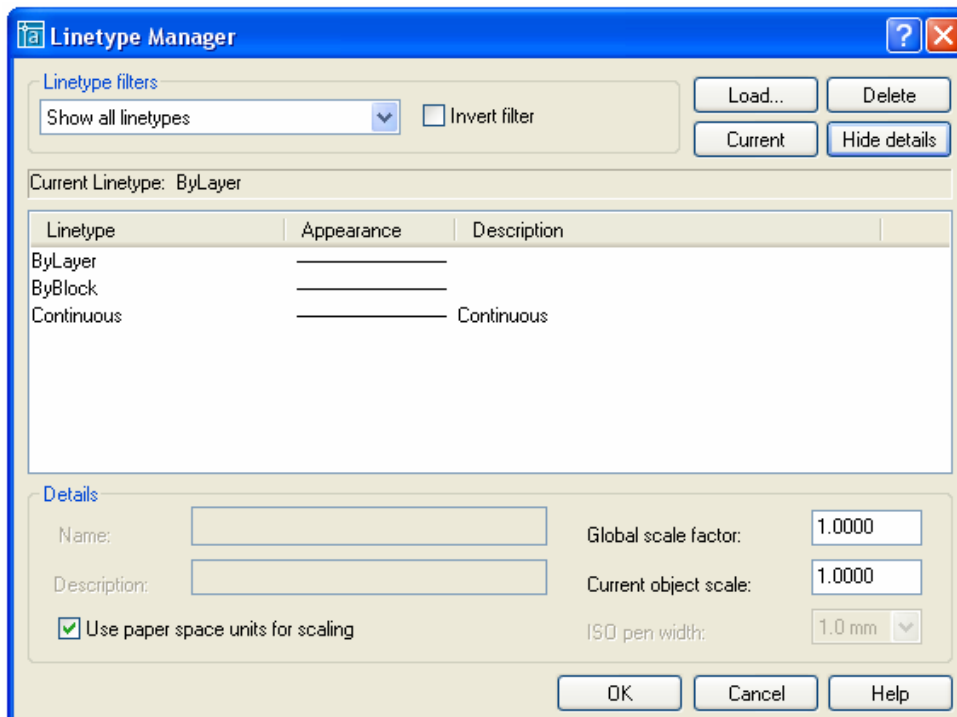
### Changing an object's linetype

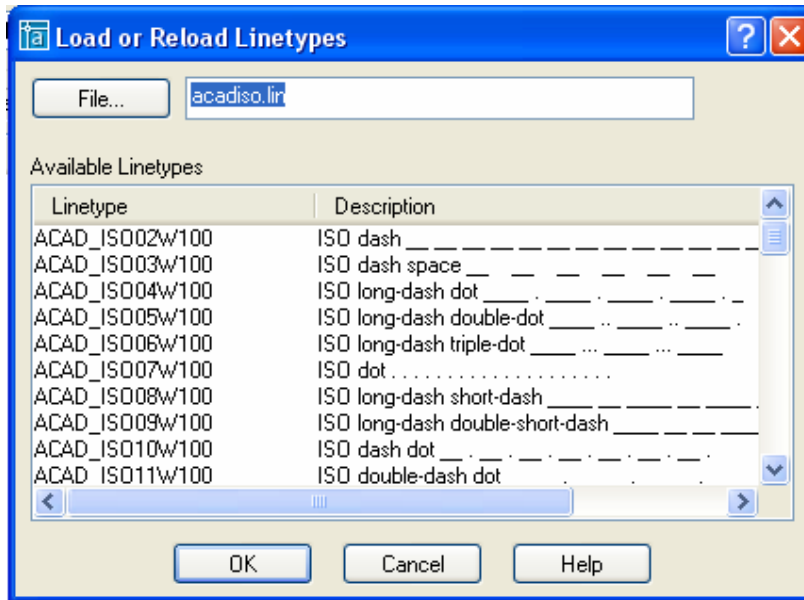
You can change the linetype of an existing object or objects. You can control an object's linetype using the Linetype Control drop-down list on the Object Properties toolbar. The Linetype Control drop-down list shows the linetype of any selected object. When you create a layer, assign it a linetype, and draw with that layer, AutoCAD displays the linetype as ByLayer. The ByLayer linetype simply means that the linetype of the object is taken from the linetype of the object's layer. The best way to organize a drawing is to assign linetypes by layer. It is common practice throughout the AutoCAD world to organize linetypes by layer.

If you need to change the linetype of an object, here are two ways to do it:

To change an object's linetype, select the object. Click the Linetype Control drop-down list and choose the linetype you want.

You can also load the linetype by clicking Format – linetype... to bring out the linetype manager shown below. Click Load to load the linetypes.





### Changing object ltypescale

To change the ltypescale, choose Format - Linetype to open the Linetype Manager, shown above. Click Show Details if the lower portion of the dialog box is not displayed.

A noncontinuous linetype is a repeating pattern of dots, dashes, and spaces. Linetypes can also include repeating text or shapes. You may find that the linetype patterns in your drawing are too long or short for clarity. The ltypescale may even be so big or so small that the line looks continuous.

How often the pattern is repeated is affected by three factors:

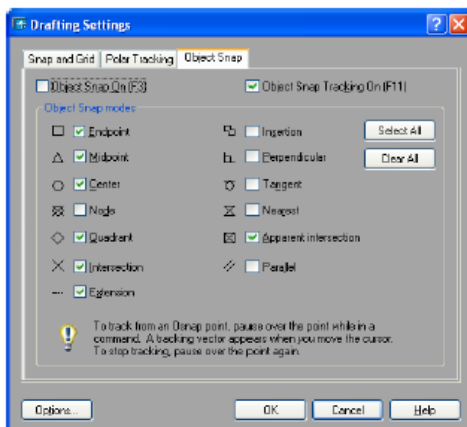
- The linetype definition
- The global ltypescale
- The individual object's ltypescale



### dsettings




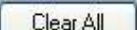
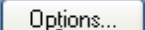
Tools - Drafting Settings...

This command provides some of the drawing aids to create the drawing at a faster pace. When you invoke this command AutoCAD displays the Drafting settings dialog box.



This dialog box contains three tabs. In this lesson, the Object Snap tab is dealt in detail. Object snaps constrain point specification to exact locations, such as a midpoint or an intersection, on existing objects. Using object snaps, you can locate an exact position on an object without knowing the coordinate or by drawing construction lines. For example, you can use an object snap to draw a line to the center of a circle or to the midpoint of a polyline segment. You can specify an object snap when AutoCAD prompts for a point.

Icon/Button	Description
<input checked="" type="checkbox"/> Object Snap On (F3)	Turns running object snaps on and off. When this is checked on, the object snaps selected under Object Snap modes are active.
<input checked="" type="checkbox"/> Object Snap Tracking On (F11)	Turns object snap tracking on and off.
<input type="checkbox"/> <input checked="" type="checkbox"/> Endpoint	Snaps to the closest endpoint of an arc, elliptical arc, line, multiline, polyline segment, or ray or to the closest corner of a trace, solid, or 3D face.
<input type="checkbox"/> <input checked="" type="checkbox"/> Midpoint	Snaps to the midpoint of an arc, ellipse, elliptical arc, line, multiline, polyline segment, solid, spline, or xline.
<input type="checkbox"/> <input checked="" type="checkbox"/> Center	Snaps to the center of an arc, circle, ellipse, or elliptical arc.
<input checked="" type="checkbox"/> <input type="checkbox"/> Node	Snaps to a point object.
<input type="checkbox"/> <input checked="" type="checkbox"/> Quadrant	Snaps to a quadrant point of an arc, circle, ellipse, or elliptical arc.
<input checked="" type="checkbox"/> <input type="checkbox"/> Intersection	Snaps to the intersection of an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline. Intersection snaps to the edges of regions and curves, but does not snap to the edges or corners of 3D solids. Extended Intersection snaps to the imaginary intersection of two objects that would intersect if the objects were extended along their natural paths. AutoCAD automatically turns on Extended Intersection when you select the Intersection object snap mode.
<input type="checkbox"/> <input checked="" type="checkbox"/> Extension	Causes a temporary extension line to display when you pass the cursor over objects, so you can draw objects to and from points on the extension line.
<input type="checkbox"/> <input type="checkbox"/> Insertion	Snaps to the insertion point of an attribute, a block, a shape, or text.
<input type="checkbox"/> <input type="checkbox"/> Perpendicular	Snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, solid, spline, or xline.
<input type="checkbox"/> <input type="checkbox"/> Tangent	Snaps to the tangent of an arc, circle, ellipse, or elliptical arc.
<input type="checkbox"/> <input type="checkbox"/> Nearest	Snaps to the nearest point on an arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, spline, or xline.
	Apparent Intersection includes two separate snap modes: Apparent Intersection and Extended Apparent Intersection. You can also locate

	<p>Intersection and Extended Intersection snap points when Apparent Intersection object snap is on.</p> <p>Apparent Intersection snaps to the intersection of two objects (arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, spline, or xline) that do not intersect in 3D space but might appear to intersect in the drawing display. Extended Apparent Intersection snaps to the imaginary intersection of two objects that would appear to intersect if the objects were extended along their natural paths.</p>
	<p>Draws a vector parallel to another object whenever AutoCAD prompts for the second point of a vector. After specifying the first point of a vector, if you move the cursor over a straight-line segment of another object, AutoCAD acquires the point.</p>
	<p>Turns on all object snap modes.</p>
	<p>Turns off all object snap modes.</p>
	<p>Displays the Drafting tab in the Options dialog box. You cannot access the Options dialog box from the Drafting Settings dialog box if you are running DSETTINGS transparently.</p>

### Selecting Objects in AutoCAD

By now you have probably seen a rectangle appear on your screen when you left-click and the move the crosshairs around. You'll learn all about these in this lesson. AutoCAD uses what's called a **selection set** to allow you to group objects together and then modify them. For example, if you want to erase several lines, you could press **E** <ENTER>, pick on the line, press <ENTER> again and repeat until you're done. Another way would be to press **E** <ENTER>, and then pick the lines one by one until they are all selected, and press <ENTER>. By selecting a group of objects, you have created a selection set. Whenever you want to modify an object, and are asked by AutoCAD to "**select objects**" you can create a selection set and then apply the command.

REMEMBER TO PRESS **ENTER** WHEN YOU ARE FINISHED SELECTING THE OBJECTS YOU WANT.

There are also a few other ways to create a selection set. The most common way is to use a window.

### There are two very different types of windows you can use.

One is a '*crossing window*' and the other is a *box*. If you create the window from **right to left**, you make a crossing window. This means that any object that crosses the border of the window is added to the selection set. This is shown as a dotted line on the screen. If you create the window from **left to right**, you create a box. Using this method you'll add only the items that are completely within the box. This is shown as a solid line on the screen.

The difference is very important. 



To summarize:

DIRECTION	CALLED	DISPLAY	EFFECT
LEFT TO RIGHT	WINDOW SELECTION	SOLID	SELECTS ON OBJECTS THAT ARE COMPLETELY WITHIN THE BOX
RIGHT TO LEFT	CROSSING SELECTION	DOTTED	SECLECTS ANY OBJECT THAT EITHER CROSSES THE BOUNDARY OR IS INSIDE IT

There are other ways to select objects and here a few of the more common ways. These can also be used in combination.

**REMOVE** - by typing **R** when asked to select objects, AutoCAD will select change to allow you to Remove objects from the selection set. You can also remove individual objects by pressing the **Shift Key** when selecting (known as a 'shift-select')

**LAST** - by typing **L** when asked to select objects, AutoCAD will select the last object that you created. This is handy if you create something and want to move or modify it right away.

**PREVIOUS** - by typing **P** when asked to select objects, AutoCAD will select the previous selection set. This is used if you select a few objects, modify them and then want to modify them again.

**FENCE** - by typing **F** when asked to select objects, AutoCAD allows you to draw a series of lines (called a fence) to select objects. This is convenient if you want to select a group of lines to be trimmed to a single cutting edge.

**CROSSING POLYGON** - typing **CP** when you are selecting objects give the ability to create a crossing polygon for object selection. This is similar to a crossing box, but you can pick points on the screen to create a polygon. By default, this is a crossing polygon, therefore any objects that cross the polygon will be added to the selection set.

**CROSSING WINDOW** - typing **CW** when you are selecting objects give the ability to create a crossing window for object selection. This is similar to a crossing box, but you can pick points on the screen to create a polygon. This is a crossing window, therefore any objects that are completely within the polygon will be added to the selection set.

**CYCLING** - When are ready to select objects, hold down the Ctrl key on your keyboard and then pick 2 points of rectangle around the area you want to select. This will highlight a few objects. Keep clicking on the screen until the one object you want to modify is highlighted. Then press <enter>. This is a good approach if you have many objects in a small area.

To unselect everything you have selected, you just press the **escape** key on your keyboard.

For practice, open one of your previous drawings and try selecting them using all of the methods described above. You need to quickly select objects in AutoCAD to be a good operator.

#### **Use Shift to add to selection**

“Use Shift to add to selection” is the second option in the Options dialog box, Selection tab. By default, this option is not checked. In AutoCAD, you often select more than one object at a time for editing. Therefore, AutoCAD simply lets you select object after object—when you select a second object, the first object stays selected—so that you can easily select any number of objects you want.

**TIP:** Remember, if you select a point by mistake while using the LINE command; type U to undo the error. You can make polar tracking snap to the angles you use most often, rather than at evenly incremented angles. In the Drafting Standards dialog box, on the Polar Tracking tag, select the new button and specify an angle you want to be able to snap to. Repeat this procedure to specify other

angles as required. Select the Polar Tracking On check box. Choose OK to exit the dialog box, and then draw objects using polar tracking and direct distance entry.

## Editing Commands



### Erase objects

#### Modify Erase

So far, we have seen two *drawing* tools LINE and CIRCLE. Now, let us see the first *editing* tool ERASE. This allows you to remove the *selected objects* from the AutoCAD drawing.

Command: `_erase` Select objects: *Pick an object to delete*

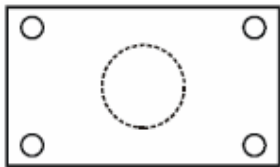
Select Objects: *Pick an object or press `Esc` to terminate the command.*

Various Object selection methods available to select the objects quickly are discussed later in this volume.

### oops

When OOPS is entered at the Command: prompt, AutoCAD restores the objects erased by the last ERASE command.

The following illustration narrates the above situation.



**Object Selected**



**After Erase**



**After Oops**

You can also use OOPS after BLOCK or WBLOCK because these commands can erase the selected objects after creating a block. You cannot use OOPS to restore objects on a layer that has been removed with the PURGE command.

## Offsetting objects

The OFFSET command creates lines or curves parallel to one existing object. The beauty of this command is apparent when you start to create complex objects, such as polylines. You may remember that polygons and rectangles are polylines, meaning that they are treated as one object.

Using OFFSET, you can create concentric polygons, for example, in one step. The figure below shows two concentric polygons. The outside polygon was created with the POLYGON command, and the inside polygon was created using OFFSET.



To offset an object, choose Offset from the Modify toolbar. You cannot select objects before choosing the command.

AutoCAD responds with the Specify offset distance or [Through] `<1.0000>`: prompt. AutoCAD offers two slightly different ways to specify the offset:

If you type an offset distance, AutoCAD responds with the Select object to offset or `<exit>`: prompt. You can select one object. Then AutoCAD displays the Specify point on side to offset: prompt. Pick a point to indicate on which side of the object AutoCAD should create the offset copy.

AutoCAD creates the offset and continues to show the Select object to offset or `<exit>`: prompt so you can offset other objects using the same offset distance. Press Enter to exit the command.


If you want to indicate a through point, which is a point that the offset passes through, such as an object snap on another object, type **t**  or right-click and choose Through from the shortcut menu. Then AutoCAD displays the Select object to offset or <exit>: prompt. Pick one object. At the Specify through point: prompt, pick a point through which you want the offset to go. AutoCAD creates the offset.

### Copying and Moving Objects

Three commands enable you to copy objects in very specific ways. MIRROR creates a mirror image. ARRAY creates a rectangular or circular pattern of an object. OFFSET creates parallel objects. Although these commands make copies of objects, they produce a result that would be difficult or impossible to produce simply by using the COPY command. The ALIGN command moves objects by aligning them with other objects in the drawing.

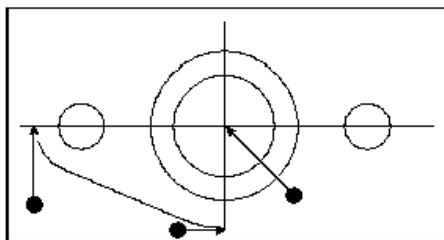
### Using the MIRROR command

Many drawings have symmetrical elements. Often, especially in mechanical drawing, you can create one-half or one-quarter of a model and complete it simply by mirroring what you have drawn.

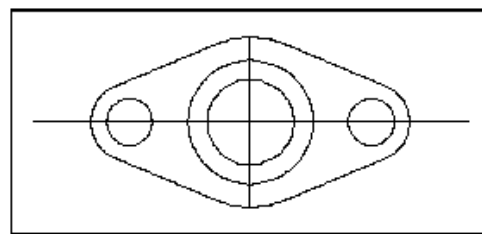
 To mirror, select an object or objects and then choose Mirror from the Modify toolbar. Alternatively, choose Mirror from the Modify toolbar and then select an object or objects. AutoCAD prompts for the first and second points of the mirror line. This is an imaginary line across which AutoCAD creates the mirrored object. The length of the line is irrelevant—only its start point and direction are important.

Most mirror lines are orthogonal. Therefore, once you specify the first mirror point, turn on ORTHO and move the mouse in the direction of the second point. You can then quickly pick the second point. Polar tracking can also easily guide you to specify an orthogonal mirror line.

AutoCAD then asks if you want to delete the source objects. The source objects refer to the objects you have selected to mirror. If you want to keep them, type **n**  or press Enter. You would keep the source objects when you are building a symmetrical model and want the mirror image to be added to the original object(s). Type **y**  when you want to edit an object—change its orientation—so that only the mirror image is retained in the drawing.



A partially completed mounting plate




The completed mounting plate

AutoCAD completes the mounting plate.

### Moving objects

Use the MOVE command to move objects in your drawing. Moving an object is more complex than erasing one because you need to tell AutoCAD the distance and direction you want the object to move.

 To move an object, select it and choose Move on the Modify toolbar. Alternatively, choose Move and then select the object. When you choose the MOVE command and once you have selected an object, AutoCAD responds with the following prompt:

Specify base point or displacement:

You now have two ways of telling AutoCAD how to move the object or objects:

**Displacement method:** At the Specify base point or displacement: prompt, state the entire displacement as an X,Y coordinate such as 2,3 or a polar coordinate such as 2<60. Because the word displacement already implies the relative distance from the object, you do not use @. (AutoCAD actually uses 0,0 as the relative point.) AutoCAD responds with the Specify second point of displacement or <use first point as displacement>: prompt. Because you have already given AutoCAD all the information it needs, press Enter. AutoCAD uses the first point you indicated as the displacement (the default) and moves the object.

**Base point/second point method:** At the Specify base point or displacement: prompt, pick a base point. This can be anywhere in your drawing. At the Specify second point of displacement or <use first point as displacement>: prompt, specify the distance and angle of movement either by picking a second point on the screen or by typing a relative coordinate, using @.

The displacement method requires less input and is simpler when you know the exact displacement so that you can type it in. The only disadvantage is that once you type in the displacement, AutoCAD sometimes displays a confusing drag line and copy of your object or objects. Ignore this display, press Enter, and your object or objects move as you specified.

The base point/second point method works best when you want to move an object relative to another object on the screen.

#### **TIP:**

*When you move an object, choose an object snap on the object or a nearby related object as the base point for exact results.*

You can use polar snap to move an object. Turn on polar snap and, if necessary, use the Drafting Settings dialog box to set the increment. You can set the polar distance; if the polar distance is 0, AutoCAD uses the Snap X spacing value. Use polar snap with the base point/second point method. At the first prompt, pick a point such as an object snap on the object. At the second prompt, drag the object in the desired direction. Use the tooltip to guide you, and then click.

You can use object drag-and-drop to move objects if you are not too particular about where they end up. Here's how it works:

1. Pick an object.
2. Continue to pick as many objects as you want to move. They all show grips.
3. Pick any of the objects, but not on a grip. Keep the mouse button pressed until the cursor changes to an arrow with a small rectangle.
4. Drag the object(s) to any other location in your drawing.

For a little more control over the placement of your objects, use the Windows clipboard. Pick the object(s) you want to move. Right-click and choose Cut from the shortcut menu. Right-click a second time and choose Paste. AutoCAD asks you for an insertion point, which you can pick with an object snap, or by typing coordinates. However, you can't control the base point (AutoCAD uses 0,0) as you can with the MOVE command.

#### **Resizing commands**

There are four additional commands that resize objects. The TRIM and EXTEND commands bring the endpoint of an object to another object. LENGTHEN lets you lengthen or shorten a line, polyline, arc, or elliptical arc. STRETCH is used to stretch (larger or shorter) a group of objects, letting you change their direction at the same time.

#### **Breaking objects**

It is often much easier to draw a long line and then break it into two or more shorter lines. A common use for BREAK is to break a wall at a door or a window in an architectural floor plan. You specify two points on the object, and AutoCAD erases whatever is between those two points. Typically, you use object snaps to specify the points. Sometimes, you can use TRIM to break an object, but if there is no convenient cutting edge, you may find BREAK more efficient.

You can break lines, polylines, splines, xlines, rays, circles, arcs, and ellipses.

To break a line, choose Break from the Modify toolbar. You cannot select the object first and then the BREAK command. AutoCAD responds with the Select object: prompt. (Notice that you can only select one object to break.) At this prompt, you have two choices:

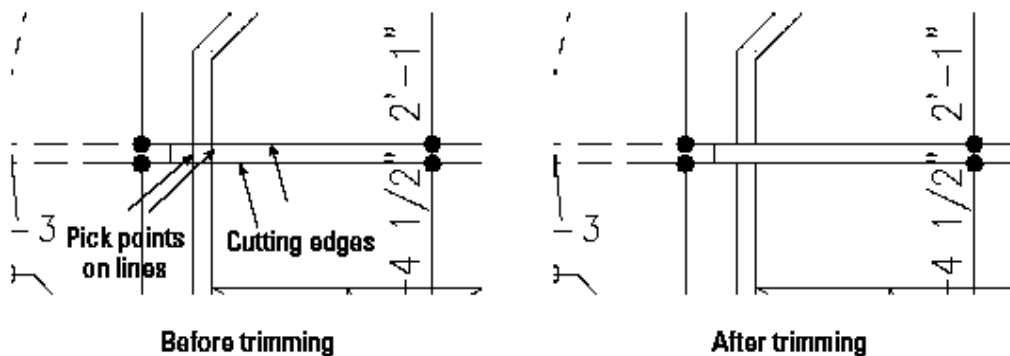
- Select the object at one of the break points you want to create. AutoCAD then responds with the Specify second break point or [First point]: prompt. Because you have already specified the first point, you can now specify the second point. AutoCAD breaks the object between the two points.
- Select the object using any method of object selection. AutoCAD then responds with the Specify second break point or [First point]: prompt. Right-click and choose First point. At the Specify first break point: prompt, pick the first break point. At the Specify second break point: prompt, pick the second break point. AutoCAD breaks the object between the two points.

*Note: Sometimes you may want to break an object into two pieces at a point, without erasing any part of the object. AutoCAD 2002 has a new button on the Modify toolbar to help you easily break an object at a point. After selecting the object, pick where you want to break the object at the Specify second break point or [First point]: prompt. The two new objects look the same as before on the screen until you select one of the objects. To do this, AutoCAD uses @, which always signifies the last point entered, to specify the second break point.*

You can use BREAK to shorten an object. Pick one point on the object where you want the new endpoint to be. Pick the other point past its current endpoint. AutoCAD cuts off the object at the point you picked on the object.

### Trimming objects

As you edit a drawing, you may find that lines or arcs that once perfectly met other objects now hang over. To trim an object, you must first specify the cutting edge, which defines the point at which AutoCAD cuts the object you want to trim. You define the cutting edge by selecting an object. You can select several cutting edges and several objects to trim at one time, as shown in the figure below. When you select an object to trim, you must pick the object on the side that you want trimmed (not on the side that you want to remain). A common use for the TRIM command is to create intersections of walls in architectural floor plans.



*Generally, you pick the objects to be trimmed individually. You cannot use Windows to select them. However, you can use the Fence object-selection method. AutoCAD trims the side of the object that the fence line crosses. You can also press Enter at the Select cutting edges: prompt. Then, when you select the objects you want to trim, AutoCAD automatically uses the cutting edge nearest your pick point. You can use this feature to trim to multiple cutting edges without having to specify each cutting edge.*

The object you want to trim does not have to actually intersect the cutting edge. AutoCAD can trim an object to a cutting edge that would intersect the object if extended. This is called *trimming to an implied intersection*, shown in below.

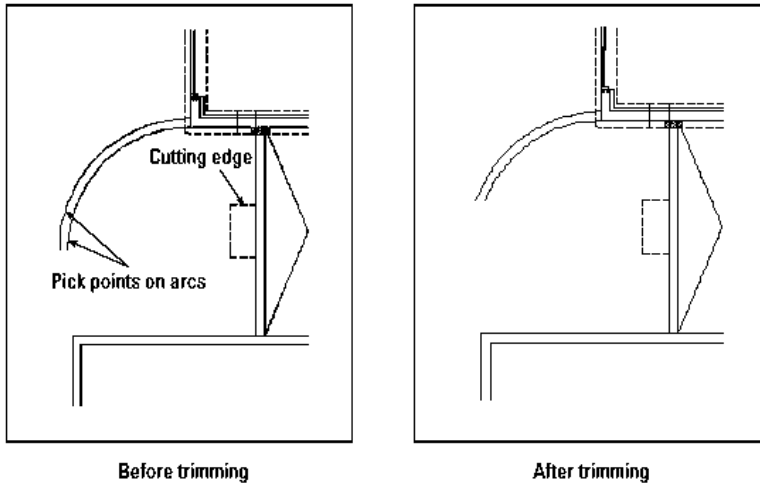
You can trim arcs, circles, elliptical arcs, lines, polylines, rays, and splines. You can use polylines, arcs, circles, ellipses, lines, rays, regions, splines, text, or xlines as cutting edges. An object can be used as both a cutting edge and an object to be trimmed in the same trimming process.



To trim an object, choose Trim from the Modify toolbar. You cannot select objects before starting the TRIM command. AutoCAD displays the Current settings: Projection=UCS, Edge=None Select cutting edges: prompt.

AutoCAD lets you know the values of the two system variables that affect trimming.

The Projection setting is used only for 3D models. The Edge setting is used for implied intersections. When Edge is set to Extend, AutoCAD trims to the implied intersection of the cutting edge and the object to be trimmed. At this prompt, pick the object(s) you want to use as a cutting edge. Press Enter to end object selection.



Trimming two arcs to an implied intersection

*If the object you want to use for the cutting edge is already selected before you start the TRIM command, AutoCAD deselects it. At the Select cutting edges: prompt, you can type **p** to reselect that object.*

You can trim to an actual or an *implied* intersection (an intersection that would exist if objects were extended):

If you want to trim to an actual intersection, at the Select object to trim or shift-select to extend or [Project/Edge/Undo]: prompt, select objects to trim. Be sure to pick each object between the cutting edge and the end you want to trim off. Press Enter to end object selection. AutoCAD trims the object(s).

If you want to trim to an implied intersection, at the Select object to trim or shift-select to extend or [Project/Edge/Undo]: prompt, type **e** ∞. AutoCAD responds with the Enter an implied edge extension

mode [Extend/No extend] <No extend>: prompt. Type **e**. Then select the objects you want to trim at the Select object to trim or shift select to extend or [Project/Edge/Undo]: prompt. Be sure to pick each object at or near the end you want to trim. Press Enter to end object selection. AutoCAD trims the object(s).

Use the Undo option if the results of the trim are not what you want. You can then continue to select objects to trim.

### Lengthening objects

The LENGTHEN command both lengthens and shortens. It works on open objects such as lines, arcs, and polylines and also increases or decreases the included angle of arcs. (You can both change the length of an arc and change its included angle using LENGTHEN.) AutoCAD offers several ways of defining the new length or included angle. Use LENGTHEN when you want to lengthen or shorten an object, but there is no available intersecting edge or boundary to use with TRIM or EXTEND.

In the LENGTHEN command, the length of an arc is measured along its circumference. Don't confuse this with the Length of chord option of the ARC command, which refers to the length of a line stretched from one endpoint of the arc to the other endpoint.



To lengthen (or shorten) an object, choose Lengthen from the Modify toolbar. You cannot select objects before the command with LENGTHEN. AutoCAD responds with the Select an object or [Delta/Percent/Total/Dynamic]: prompt. Choose one of the following options:

**Select object:** This is the default. However, its purpose is to display the current measurements of the object. This can help you to decide how to define the final length or angle of the object. The current length is displayed at the command line, and the previous prompt is repeated.

**Delta:** Right-click and choose Delta. Delta means the change, or difference, between the current and new length or included angle. AutoCAD responds with the Enter delta length or [Angle] <0.0000>: prompt. If you want to change an included angle, right-click and choose Angle. Then type the change in the included angle. Otherwise, simply type in the change in the length of the object. A positive number increases the length or included angle. A negative number decreases the length or included angle.

**Percent:** Right-click and choose Percent. At the Enter percentage length <100.0000>: prompt, type in what percent of the original object you want the final object to be. Amounts over 100 lengthen the object. Amounts under 100 shorten the object. You cannot change an included angle using this option.

**Total:** Right-click and choose Total. At the Specify total length or [Angle] <1.0000>: prompt, you can either choose the Angle suboption, as described for the Delta option, or use the default total length option. Either way, you enter the total angle or length you want.

**Dynamic:** Right-click and choose Dynamic. This option lets you drag the endpoint of the object closest to where you picked it. You can use an object snap to specify the new endpoint.

Once you have used an option to specify the length you want, AutoCAD responds with the Select an object to change or [Undo]: prompt. Here you select the object you want to change. Be sure to pick the endpoint of the object for which you want to make the change. AutoCAD then continues the same prompt so that you can pick other objects using the same length specifications. Choose Undo to undo the last change. Press Enter to end the command.

### Stretching objects

The STRETCH command is generally used to stretch groups of objects. It can be used to enlarge a room in a floor plan, for example. You can also shrink objects. You can change not only the length of the objects but the angle as well. You use a crossing window to choose the objects to be stretched. All objects that cross the boundaries

of the crossing window are stretched. All objects that lie entirely within the crossing window are merely moved. Successful stretching involves precise placement of the crossing window. Figure below shows the process of stretching a garage. Note that the walls that cross the boundaries of the crossing window are stretched.

However, the dormer, which is entirely within the crossing window, is just moved. This maintains the integrity of the model.

You cannot stretch circles, text, or blocks. You can stretch arcs, although the results may not be what you expect. The real power of the STRETCH command is in stretching a number of objects at once. However, you can also stretch one line. The results are similar to using the CHANGE command to change the endpoint of a line.



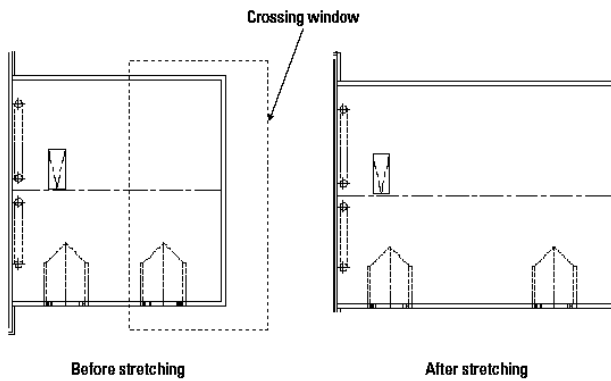
To stretch objects, choose Stretch from the Modify toolbar. AutoCAD responds with the Select objects to stretch by crossing-window or crossing-polygon... instruction and then the Select objects: prompt. Create the crossing window and select the objects you want to stretch. (You can also use a crossing polygon.) After completing the crossing window, check to see which objects are highlighted. This helps you avoid unwanted results. You can use the object selection Remove option (type **r - enter** at the command prompt) to remove objects by picking that you don't want to stretch or move.

AutoCAD then displays the Specify base point or displacement: prompt. This step is just like moving objects. You can respond in two ways.

Pick a base point. At the Specify second point of displacement: prompt, pick a second point. Object snap and polar snap are helpful for picking these points.

Type a displacement, without using the @ sign. For example, to lengthen the objects by 6 feet in the 0-degree direction, type **6'<0**. Then press Enter at the Specify second point of displacement: prompt.

When specifying a displacement by typing at the keyboard, you can use both positive and negative distances. For example, 6'<180 is the same as -6'<0. Both would stretch the objects 6 feet to the left.



### Rotating objects

AutoCAD enables you to easily rotate an object or objects around a base point that you specify. The base point is usually an object snap point on the object. To indicate the rotation, specify an angle of rotation. Zero degrees is generally to the right, and degrees increase counterclockwise, although you can change this convention. By specifying a negative angle, you can turn objects clockwise.

To rotate an object, choose Rotate from the Modify toolbar and select an object. Alternatively, select an object and then choose Rotate from the Modify toolbar. At the Specify base point: prompt, indicate the point around which you want to rotate. At the Specify rotation angle or [Reference]: prompt, type an angle at the command line.

The Reference option lets you specify the angle with reference to another angle or an object. At the Specify the reference angle <0>: prompt, you type in an angle or (more likely) specify an angle by picking two points. These are often object snap points on the object that specify the object's current angle. At the Specify the new angle: prompt type or pick a new angle. This new angle can also be indicated by picking an object snap on another object in the drawing. You can use the Reference option to align the object with another object in your drawing.

### Scaling objects

*Scaling*, or resizing, objects is another common editing task in AutoCAD. As with rotating objects, you specify a base point, usually an object snap on the object. The base point is the one point on the object that does not move or change as you scale the object. The most common way to resize an object is to specify a scale factor. The current object has a scale factor of 1. Therefore, to increase the size of the object, type in a number greater than 1. For example, a scale factor of 2 doubles the size of the object. To decrease the size of the object, type in a number less than 1. A scale factor of 0.25 creates an object one-quarter of its previous size.

As with the ROTATE command, you can scale using the Reference option. You specify the reference length, usually the current length of the object, by typing it in or using object snaps on the object. At the Specify new length: prompt, you can type a new length or pick a point. AutoCAD measures this point from the base point you specified to determine the new length.

To scale an object, choose Scale from the Modify toolbar and select the object. Alternatively, select the object and choose Scale from the Modify toolbar.

### Breaking objects

It is often much easier to draw a long line and then break it into two or more shorter lines. A common use for BREAK is to break a wall at a door or a window in an architectural floor plan. You specify two points on the object, and AutoCAD erases whatever is between those two points. Typically, you use object snaps to specify the points. Sometimes, you can use TRIM to break an object, but if there is no convenient cutting edge, you may find BREAK more efficient.

You can break lines, polylines, splines, xlines, rays, circles, arcs, and ellipses.

To break a line, choose Break from the Modify toolbar. You cannot select the object first and then the BREAK command. AutoCAD responds with the Select object: prompt. (Notice that you can only select one object to break.) At this prompt, you have two choices:



- Select the object at one of the break points you want to create. AutoCAD then responds with the Specify second break point or [First point]: prompt. Because you have already specified the first point, you can now specify the second point. AutoCAD breaks the object between the two points.
- Select the object using any method of object selection. AutoCAD then responds with the Specify second break point or [First point]: prompt. Right-click and choose First point. At the Specify first break point: prompt, pick the first break point. At the Specify second break point: prompt, pick the second break point. AutoCAD breaks the object between the two points.

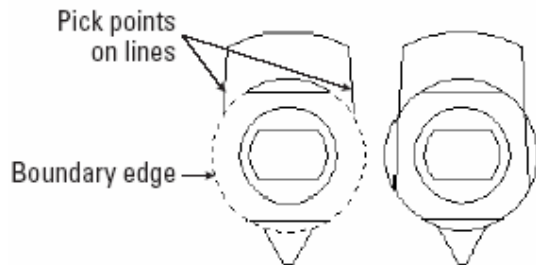
*Note: Sometimes you may want to break an object into two pieces at a point, without erasing any part of the object. AutoCAD 2002 has a new button on the Modify toolbar to help you easily break an object at a point. After selecting the object, pick where you want to break the object at the Specify second break point or [First point]: prompt. The two new objects look the same as before on the screen until you select one of the objects. To do this, AutoCAD uses @, which always signifies the last point entered, to specify the second break point.*

You can use BREAK to shorten an object. Pick one point on the object where you want the new endpoint to be. Pick the other point past its current endpoint. AutoCAD cuts off the object at the point you picked on the object.

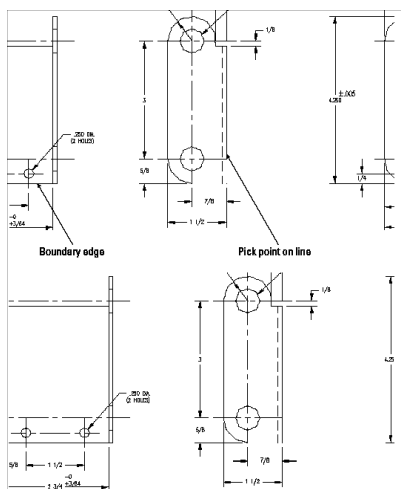
### Extending objects

The EXTEND command has the same prompts as the TRIM command, but instead of trimming objects to a cutting edge, it extends them to a boundary edge. As with TRIM, when you select an object to extend, you must pick the object on the side that you want extended (not on the side that you want left as is).

The object you want to extend does not have to actually intersect the boundary edge after its extension. AutoCAD can extend an object to a boundary edge that would intersect the extended object if it were longer. This is called extending to an implied intersection.



Extending two lines using an arc as the boundary edge



Extending a line to an implied intersection

You can extend arcs, elliptical arcs, lines, open polylines, and rays. You can use polylines, arcs, circles, ellipses, lines, rays, regions, splines, text, or xlines as boundary edges. An object can be used as both a boundary edge and an object to be extended in the same extending process.

While using the **EXTEND** command, you can now easily switch to trimming objects by pressing the **Shift** key as you select objects to trim.



To extend an object, choose **Extend** from the **Modify** toolbar. You cannot select objects before the **EXTEND** command. AutoCAD displays the Current settings: **Projection=UCS, Edge=Extend Select boundary edges ... Select objects: prompt.** AutoCAD lets you know the values of the two settings that affect extending. **Projection** is used only for 3D models. **Edge** is used for implied intersections.

When **Edge** is set to **Extend**, AutoCAD extends to the implied intersection of the boundary edge and the object to be extended. At this prompt, pick the object(s) you want to use as the boundary edge(s). Press **Enter** to end object selection. You can extend to an actual or implied intersection:

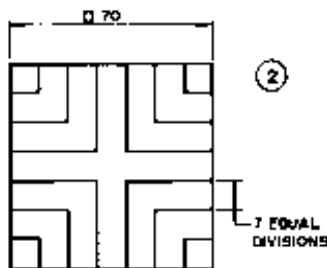
If the extension will result in an actual intersection, at the **Select object to extend or shift-select to trim or [Project/Edge/Undo]: prompt**, select objects to extend. Be sure to pick each object at the end you want to extend. Press **Enter** to end object selection. AutoCAD extends the object(s).

If you want to extend to an implied intersection, at the prompt, right-click and choose **Edge**. AutoCAD responds with the **Enter an implied edge extension mode [Extend/No extend] <Extend>: prompt.** Right-click and choose **Extend**. Then select the objects you want to extend at the **Select object to extend or shift-select to trim or [Project/Edge/Undo]: prompt.** Be sure to pick each object at the end you want to extend. Press **Enter** to end object selection. AutoCAD extends the object(s).

Use the **Undo** option if the results of the extension are not what you want. You can then continue to select objects to extend.

You can use the **Fence** object selection method to select objects to extend. AutoCAD extends the side of the object that the fence line crosses.

### Assignment:




### Direct Distance Entry

As mentioned earlier in the lessons, there are many ways to do things in AutoCAD. To enter distances, you have been shown **Absolute**, **Relative** and **Polar Co-ordinates**. If you had to type in coordinates every time you needed to enter a point, creating drawings would take a very long time. To make selecting points easier, AutoCAD lets you select them with your mouse. Now you will be shown two more ways to tell AutoCAD where to locate the point you are drawing to.

### DO NOT FORGET THE BASICS OF MANUAL ENTRY

**Direct Distance Entry** (or **DDE**) is a way of bypassing the usual ways of entering in co-ordinates and just entering the distance. The method is quite easy.

Make sure that you have set **Ortho** (locking your input to vertical and horizontal) by pressing the **F8** button and checking to see that the "Ortho" button is depressed on the status bar like this:  Your **F8** key will toggle **Ortho** on and off.

Let's say you want to draw a line from one point directly to the right 10 units.

Start the **Line** command and click on the screen anywhere, then move your cursor to the right of that point. If **Ortho** is turned on, the line should only point directly to the right and not at an angle. Now

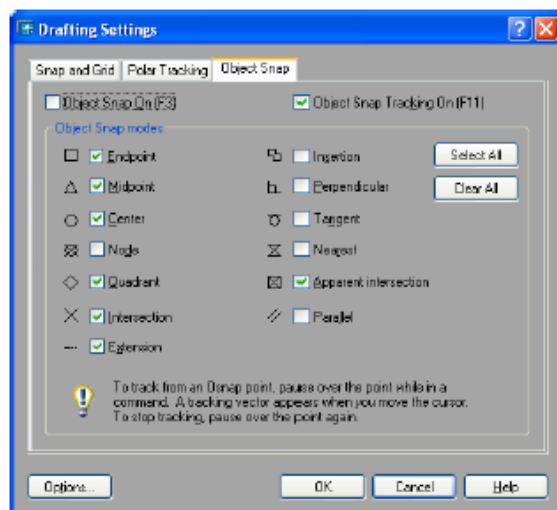
type in **10** <ENTER>. Press <ENTER> again to exit the command. You now have line that is 10 units long.




That is a very easy way to draw line segments. If Ortho is not set, your drawing will get messed up very quickly and will not be easy to fix. I have seen too many students take this easy route and wreck projects.

Now wouldn't it be cool to draw angled lines (like the short ones in the above image)? Well you can, but first you have to make change in your settings.

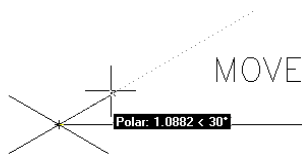
Type in **DSETTINGS** and you get the Drafting Settings dialog box:



Pick on the Polar Tracking tab if it isn't open. Then make sure that Polar Tracking is On (press your F10 key to toggle it on and off) and then select the increment angle. 

### Polar Tracking

In the drawing image above, I made 4 lines 1 unit long using Polar Tracking. See if you can duplicate this on the end of the 10 unit line you just drew. The process is the same as you did for the DDE of the previous line. Make sure your increment angle is 30°.



You can not have Ortho and Polar Tracking on at the same time. As you start to draw more, you will see that these two features are great time savers. There will be times, though, when you have to use absolute and relative co-ordinate entry (especially in 3D).

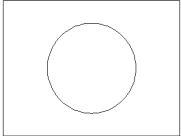
Now I'm going to repeat this one last time:

**Make sure you have either Ortho or Polar tracking on and don't forget how to manually enter points!**

## Object Snap Tracking

Now you've just seen how you can find distances from points and measure accurately, but what if you want to find specific points based on previously drawn objects? You can use Osnaps, but they don't always find the point you need.

Consider that you want to draw a circle in the middle of a rectangle like the example below:



In the olden days, you would have drawn a line from the corners of the rectangle to find the middle, then snapped to the midpoint of that line for the circle's center point, drawn the circle and then erased the reference line.

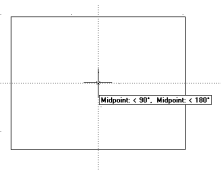
Now you can use **object snap tracking**. This is a way of finding reference points and drawing from them. To do this exercise, make sure that your Midpoint Osnap is on.

Draw a **RECTANGLE** from 0,0 to 4,3

Make sure that the Object Snap Tracking **OTRACK** and Osnap button are depressed on the status bar.

Start the **CIRCLE** command.

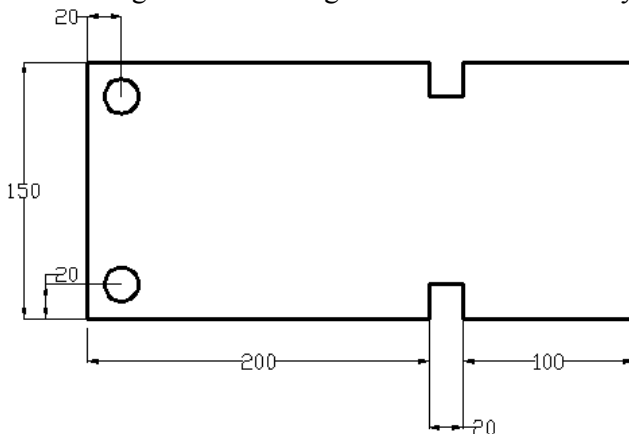
Move your cursor over the middle of the bottom line of the rectangle. You should see a light dotted line project vertically through your cursor. Now move your cursor to the middle of the right vertical line of the rectangle. As you move your cursor towards the center of the rectangle, you should see the 2 dotted lines cross. It should look like this:



Once you see both lines, you can pick and the center point will be exact center of the rectangle. Give your circle a radius of 1".

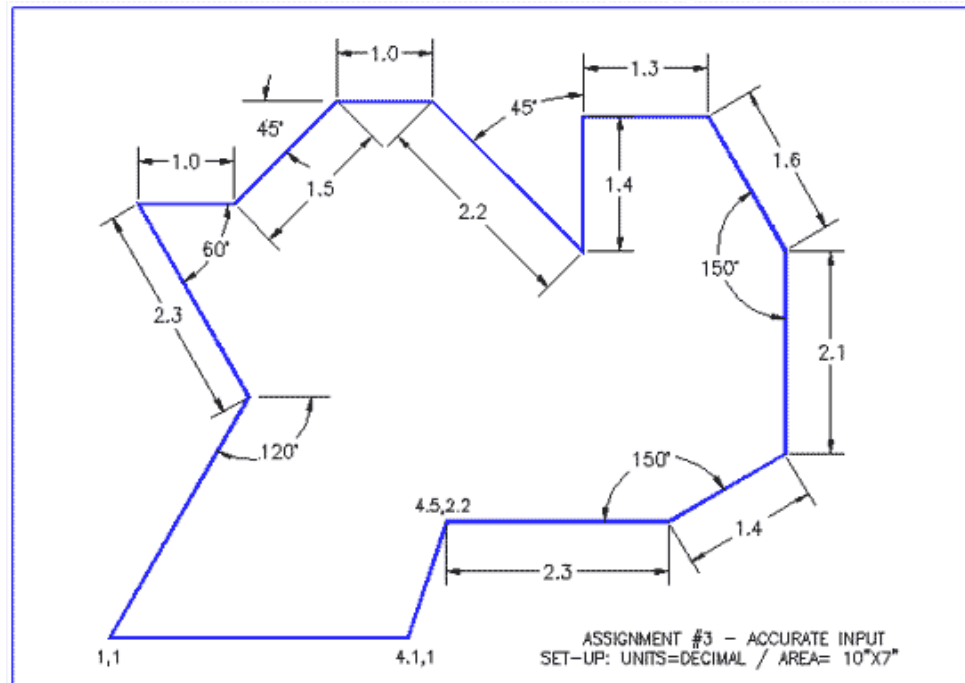
***This is a very simple example, but as you start drawing more complex shapes, this will be a handy to tool to master. Object tracking will work with any Osnap that is invoked.***

Draw this figure below using the direct distance entry.



### Accurate Input

Now it's time to see if you were paying attention in the first lesson. In this lesson you will complete the drawing that will test your knowledge of drawing accurately using different types of precision input. Start at the bottom left corner. Use the drawing tools you have learned in the previous lessons. You will have to use a combination of absolute, relative, and polar co-ordinate input.




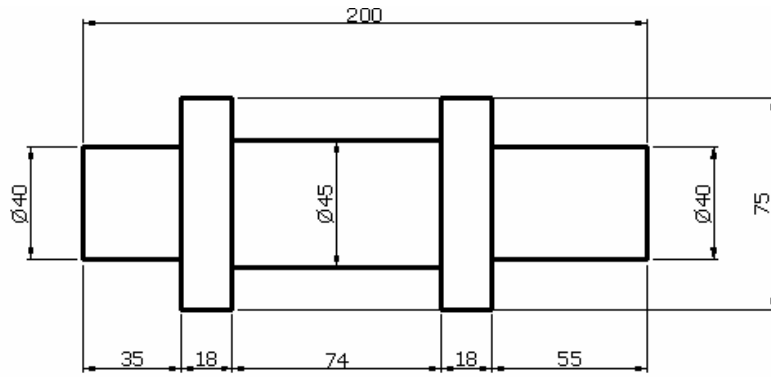
1. Open AutoCAD and set up the drawing as you have for the previous assignments.
2. Complete the assignment by drawing it accurately using the dimensions provided.
3. If you make an error along the way, remember that you can use your **ENDPOINT** Osnap to begin where you left off.
4. Save your drawing.

### From

The From feature enables you to create a new object starting at a known distance and direction from an existing object. It's like creating one or more invisible lines between the existing object and the new object, helping you start the new object in the proper place. Use From when no object snaps are available to help you determine a new point.

Here's how to use the From feature:

1. Start a command to draw an object, such as LINE.
2.  Choose Snap From on the Object Snap flyout of the Standard toolbar (or press Shift and right-click and choose From on the shortcut menu). You can also type **from** on the command line.
3. AutoCAD prompts you for a base point, which you usually provide using an object snap, such as endpoint.
4. AutoCAD prompts you for an Offset, which you provide using relative or polar coordinates.
5. Continue the command you started (in Step 1).

**Assignment:**

## Lesson 3

### Drawing Circles



**Circle**

**Draw**  **Circle**  [options]

You can create circles in several ways. The default method is to specify a center and radius. AutoCAD also provides 3 more methods to draw circles which are discussed below.

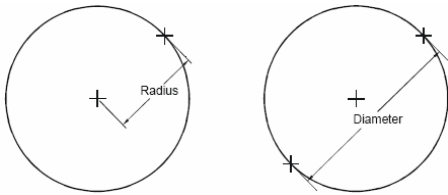
Command: `_circle`

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *Use one of the point fixing methods or enter an option*

#### Center, Radius/Center, Diameter

This allows drawing a circle based on the center point and radius or diameter. The radius or diameter can be specified by fixing a point or by entering a value. The distance between the specified point and the center, is the radius or diameter, as the case may be.

Specify radius of circle or [Diameter]: *Enter radius value or enter D for Diameter option*



#### 3Points

A circle can be fixed by specifying any 3 circumferential points using this option.

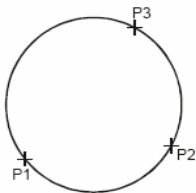
Command: `_circle`

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: `_3p`

Specify first point on circle: *Use one of the point fixing methods*

Specify second point on circle: *Use one of the point fixing methods*

Specify third point on circle: *Use one of the point fixing methods*



#### 2Points

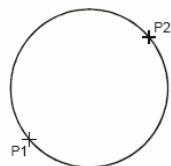
Specify 2 diametrically opposite points to draw a circle using this method.

Command: `_circle`

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: `_2p`

Specify first end point of circle's diameter: *Use one of the point fixing methods*

Specify second end point of circle's diameter: *Use one of the point fixing methods*



### Tangent, Tangent, Radius

Select this to draw a circle of given radius. The circle will be fixed tangential to both the selected objects.

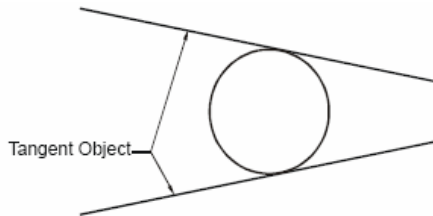
Command: `_circle`

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: `_ttr`

Specify point on object for first tangent of circle: *Select an object*

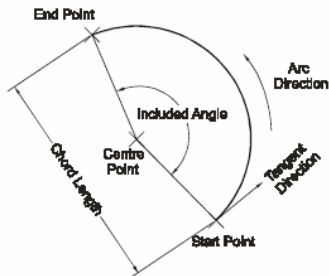
Specify point on object for second tangent of circle: *Select an object*

Specify radius of circle <1.0000>: *Enter a value.*

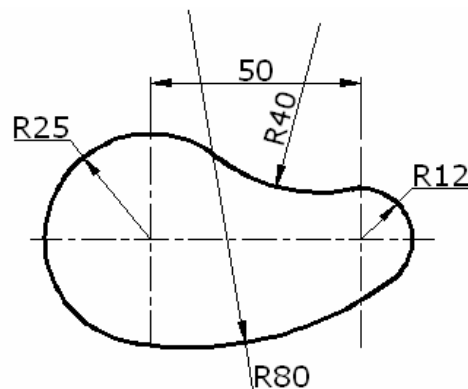
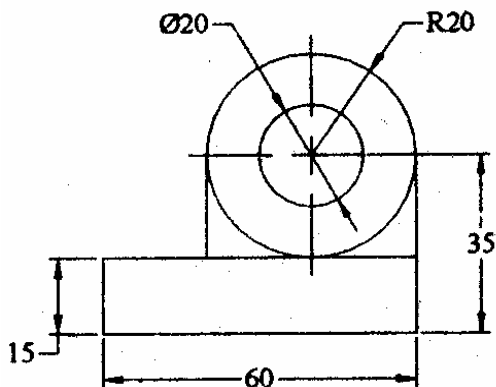


### Arc Draw `□ Arc □ [options]`

ARC command allows you to create an arc segment. Arcs created by AutoCAD are circular arcs which form a part of a circle. There are different methods of creating an arc, as illustrated below. The default method is to specify the circumference with three points.



Let us have a recap of the tools what we have learned so far. As you do these Hand On exercises, you will familiarize yourself with the Basic 2D Drafting commands and File handling features of AutoCAD. Our objective is to draw the simple models shown here.






## Zoom and Pan Options

### Zoom and Pan with scroll mouse


#### *Scrolling, Panning, & Zooming*

The user can see other parts of the drawing by **scrolling** or **panning**. Scrolling is done via the scroll bar arrows.

To *pan* means to move the display without changing the magnification. The word refers to the expression of panning a camera across a scene or view. You pan to view a different part of your drawing. You can pan any amount and in any direction using the PAN command. You can pan vertically and horizontally using the scroll bars. To leave pan mode, press Esc or Enter or start any command via a menu or toolbar.


To pan across the drawing, choose the Pan Realtime  tool from the Standard toolbar. Notice how the cursor changes from an arrow into a hand. Click and hold down the left mouse button, and move the cursor in the desired direction. The drawing follows the movement of the hand cursor. Pan can be used transparently, while you are in the middle of another command.


**Zooming** allows you to change drawing magnification by zooming in or out.


**Zoom All**  tool (in the **Standard** toolbar) returns the drawing view to its original position.


**Zoom Out**  reduces the apparent drawing size in the graphic window by 50%.


**Zoom In**  enlarges by 200%.

**Zoom Window**  tool allows you to specify a rectangular area as an area of the drawing to fill the window.

**Zoom Realtime**  tool followed by a click anywhere in the graphics window, then moving the cursor in and out of the graphics window gives dynamic magnification. Pressing **Escape** or **Enter** exits the command.

**Zoom Previous**  tool returns you to the previous view of the drawing ... you can use it repeatedly to switch through up to 10 previous views.

**Zoom Extents**  tool zooms to the edges of the drawing.

**Zoom Scale**  Type a scale factor followed by X. AutoCAD's default zoom—e.g., type z, then **Enter**. Typing 2x, then enter makes the drawing appear twice as large on the screen. It zooms "in". Typing .5x, then enter makes the drawing appear half as large on the screen. It zooms "out".

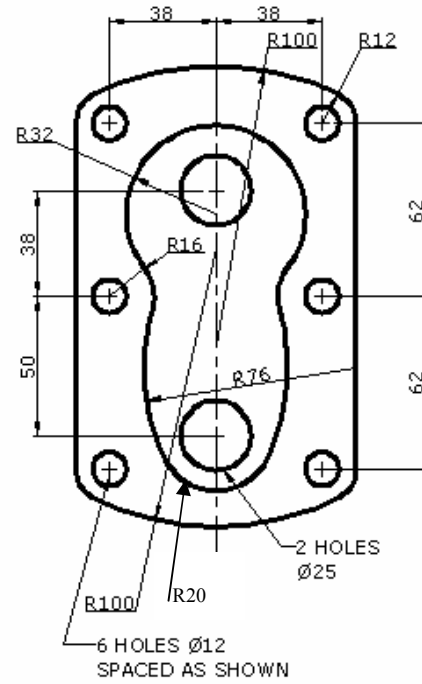
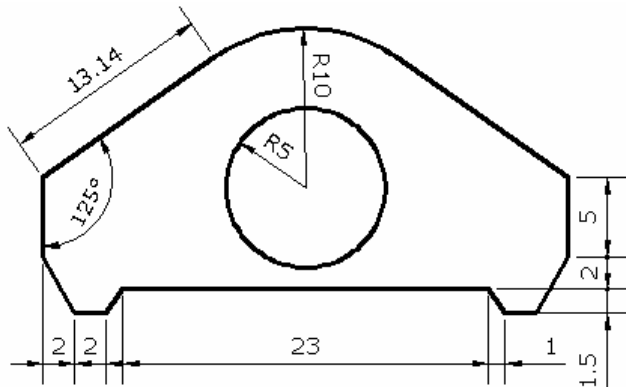
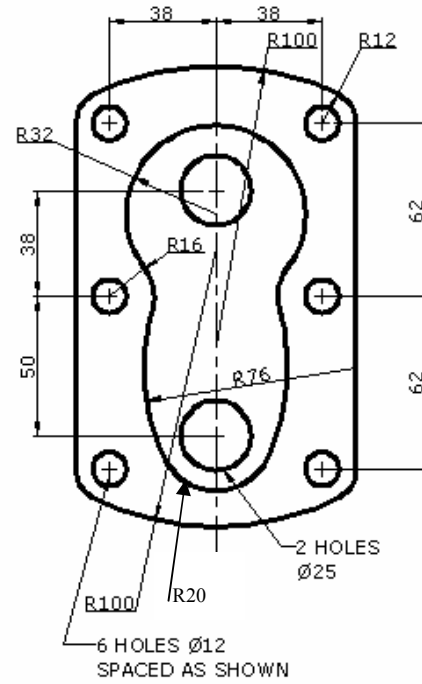
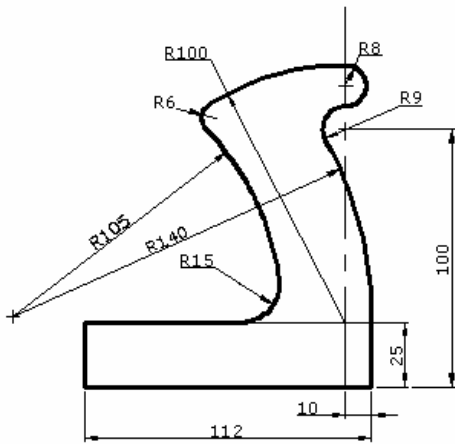
## Redraw and Regen

You use the REDRAW command to remove blips or to quickly refresh the screen. (The REDRAWALL command redraws the display in all viewports.) Use the REGEN command whenever you want AutoCAD to recalculate the entire drawing. AutoCAD users use the word "regen" to refer to the REGEN command and also as shorthand for regenerate or regeneration. However, in most cases, you want to redraw rather than regen because it is quicker. To redraw the screen, choose View Redraw (which actually executes the REDRAWALL command).

To regenerate the entire drawing, type regen at the command line and press enter. (REGENALL regenerates all viewports.) Certain zooms and pans may require a regeneration, which AutoCAD does automatically.

In recent releases, AutoCAD has significantly improved the mechanics behind the display of your drawing. It is now rarely necessary to regenerate a drawing.

Assignments:



## Lesson 4

### User Coordinate Systems

By default, a drawing is set up using a World Coordinate System. This sets the origin of the X,Y points at 0,0 and the angles using the familiar East (right-facing) equals 0 degrees system. The UCS icon in its default display is shown below.



You can easily create your own UCS and even save it for future use in the drawing. Creating a UCS is essential for 3D drawing but often useful for 2D drawing as well. To define a UCS in a 2D drawing, you indicate the angle of the X and Y axes and an origin point. The origin point then becomes the new 0,0 coordinate. AutoCAD offers several options for specifying the UCS.

#### Understanding UCS options

To create a UCS, choose Tools → New UCS. (You can also choose a button on the UCS flyout of the Standard toolbar.)

#### Saving and restoring a custom UCS

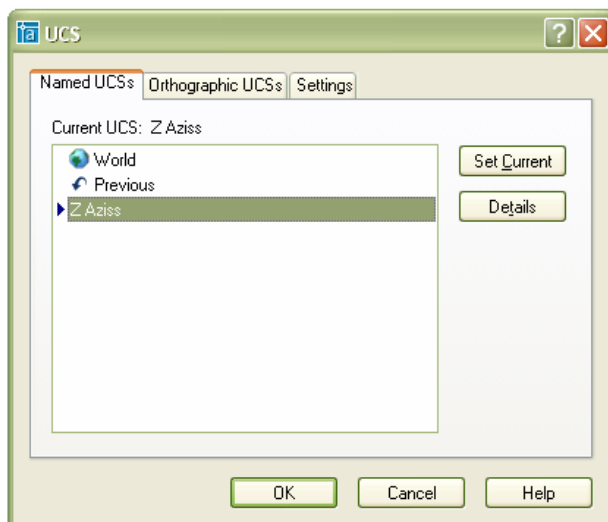
Once you create a UCS, you can save it so you can easily switch back and forth between the World Coordinate System and your UCS. To save a UCS, follow these steps:

1. Specify the UCS as described in the previous section.
2. Choose Tools → Named UCS. AutoCAD opens the UCS dialog box with the Named UCSs tab on top. If your current UCS is not named, it is listed as Unnamed, and highlighted. Previously named UCSs are also listed. You can also use this dialog box to return to the previous UCS or the World Coordinate System (WCS).
3. Click Unnamed. Type a name for the UCS and press Enter. The name can be up to 255 characters and can include spaces.
4. Click OK.

To restore a saved UCS, choose Tools → Named UCS. (You can also choose Display UCS Dialog from the UCS flyout of the Standard toolbar.) Choose the UCS you want to use and click Set Current. Then click OK.

You can use the same dialog box to return to the previous UCS you used. But a quicker way is to choose UCS Previous from the UCS flyout on the Standard toolbar. Using this button, you can quickly switch back and forth between two UCSs.

To delete a saved UCS, select it in the UCS dialog box and press the Del key.





## Controlling the UCS icon




When you start a new drawing, the UCS icon is at the bottom-left of your drawing at 0,0. By default, the icon remains at 0,0 even if you pan around the drawing. The icon may therefore end up in the middle of your drawing. If 0,0 is not displayed, then the icon reverts to the bottom-left of your drawing.

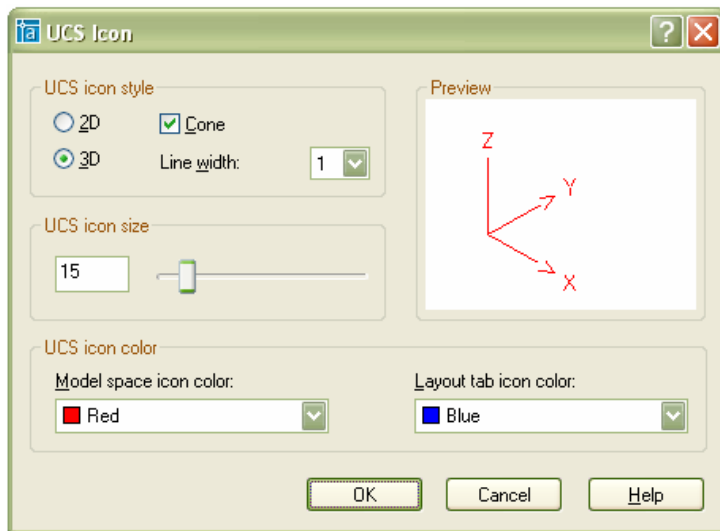
You can turn off the UCS icon completely. If you are not working with customized UCSs, you often have no reason to see the UCS in a 2D drawing. If you want the UCS icon on, you can also tell AutoCAD to display it only at the bottom-left of your drawing. The value of this is obvious—it keeps the icon out of the way so it does not obstruct your drawing.

If you create a new UCS, keeping the UCS icon at the 0,0 point (the origin) of your new UCS helps you get your bearings. AutoCAD places a plus sign in the icon to indicate the origin. However, if the origin is out of the current display or so close to the edge that the icon won't fit, AutoCAD places the UCS icon at the lower-left corner of your drawing anyway.

To control the UCS icon, choose View  Display  UCS Icon. (Don't confuse this with Tools  New UCS, which creates a custom UCS.) A submenu opens with the following items:

- Icon: Toggles the display of the UCS icon on and off.
- Icon Origin: Toggles the placement of the UCS icon at the origin on and off.

To customize the look of the UCS icon, choose View  Display  UCS Icon  Properties (or type **ucsicon** and choose the Properties option) to open the UCS Icon dialog box, shown below. As you make changes, you can see the result in the preview box.



Choose between 2D and 3D styles. The 3D style shows the Z axis, but you won't see it when you are looking at your model from the top (called plan view). If you choose the 3D style, you can check the Cone box to display a cone effect for the arrows of the X and Y axes. Either way, the Z axis has no arrow, so you can distinguish it from the X and Y axes. You can also choose a line width from 1 to 3, 1 being the default.

You can change the size of the icon. Just drag the slider up and down to get the result you like. Finally, you can change the color of the icon both in model space and in paper space layouts. Click OK to close the dialog box.

## Using a custom UCS

Although it seems confusing at first, once you try creating your own UCS it's easier to understand how the process works.

## Inquiry Commands

Calculating distances

You can easily calculate the distance between any two points by using the DIST command.

Choose Tools ⇨ Inquiry ⇨ Distance. The command prompts you for two points. You can use any means of specifying a point, although object snaps or snap mode are useful if you want to be sure of which point you're specifying.



Inquiry toolbar:

You can use this information to check dimensions or to make calculations that you need for drawing.

### Finding coordinates

Finding a coordinate is even easier than calculating a distance. Choose Tools ⇨ Inquiry ⇨ ID Point (or just type **id**). The ID command prompts you for a point. You can use any means of specifying a point, although object snaps or snap mode are useful.

Here is a typical listing: X = 6'1'-5 1/8" Y = 32'-4 5/8" Z = 0'-0"

You can use this information to find an absolute coordinate.

### Calculating area and perimeter

You can also calculate the area and perimeter of any area. To start the AREA command, choose Tools ⇨ Inquiry ⇨ Area.

You can specify points bounding the area that you want to calculate. The points don't have to be on an object. The command calculates the area and perimeter as if you had drawn lines between all of the points. The command automatically closes the area if you don't close the area by assuming that the last point is the same as the first point. This option is limited to areas with straight sides.

Instead of picking points, you can use the Object option to calculate the area of objects, which gives you the flexibility to use areas that include curves. Acceptable objects are circles, ellipses, splines, polylines, polygons, regions, and objects created with the 2D SOLID command. This option calculates an area, but adds a length instead of a perimeter. For a circle, you get a circumference.

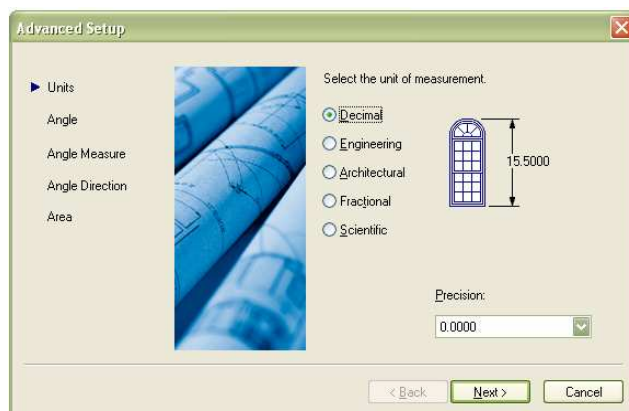
Although a polyline may have the exact same shape as an arc, you cannot calculate the area of an arc. Polylines and splines can be open. In this case the area is calculated as if you had drawn a line from the endpoint to the start point, but it does not include that last imaginary line when calculating the length.

To calculate the area of irregular shapes, you can keep a running total by using the Add option. Start by specifying the first area. Then use the Add option and specify a second area.

The result is the sum of the two areas. You can continue to add areas. You can also use the Subtract option to subtract areas.

### Units

The coordinates you use in AutoCAD are measured in units that can represent any real world measurement, such as inches or millimeters. A surveyor or city planner might even use miles as the base unit. However, different disciplines customarily express units differently, and you should use the unit type appropriate for the type of drawing you are creating. AutoCAD offers five types of units, as shown in Table below. The sample measurement column shows how a line 32.5 inches long would be displayed in the various unit types.



<b>Unit Type</b>	<b>Unit Types Sample Measurement</b>	<b>Description</b>
Decimal	32.50	Number of units, partial units in decimals
Engineering	2'-8.50"	Feet and inches, partial inches in decimals
Architectural	2'-8 1/2"	Feet and inches, partial inches in fractions
Fractional	32 1/2	Number of units, partial units in fractions
Scientific	3.25E+01	Base number + exponent

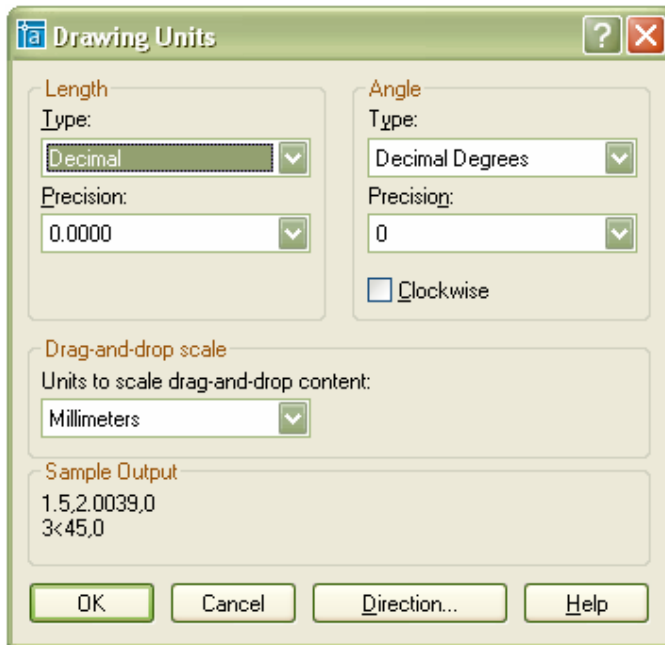
Notice how the engineering and architectural units translate a line of 32.5 units into feet and inches. Engineering and architectural units assume a unit of one inch, unlike the other unit types, which can represent any measurement.

The unit type affects how coordinates are shown on the status bar and how AutoCAD lists information about objects. You generally input coordinates using the type of units you have chosen, although AutoCAD in some cases accepts coordinates in another unit type.

### Setting the drawing units

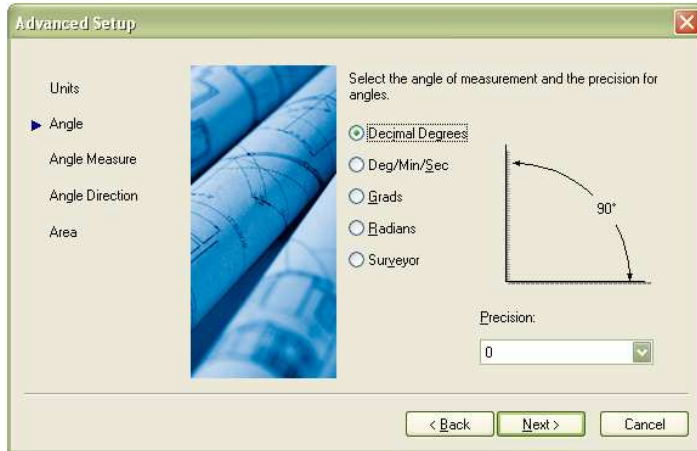
To set the units, choose the type of units you want in the Units step of the Advanced Setup Wizard. Choose a precision in the Precision drop-down list box and then click Next.

To set drawing units without the wizard, choose Format → Units to open the Drawing Units dialog box, shown in the figure below. The left side of the Drawing Units dialog box enables you to choose which unit type you want to use. In the Precision box in the Length section, click the arrow and a list of precision options drops down. Click the one you want. You can also set the units that AutoCAD uses when inserting the drawing into another drawing using the DesignCenter.



### Setting the angle type

The second step of the Advanced Setup Wizard enables you to set the angle type.



As with units, your choice of angle type depends on your profession and work environment. Decimal Degrees is the default.

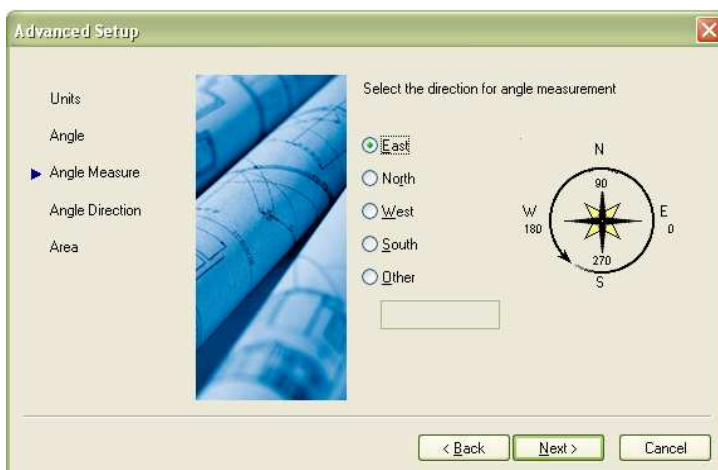
<b>Angle Type Name</b>	<b>Angle Types Sample Measurement</b>	<b>Description</b>
Decimal Degrees	32.5	Degrees, partial degrees in decimals
Deg/Min/Sec	32°30'0"	Degrees, minutes, and seconds
Grads	36.1111g	Grads
Radians	0.5672r	Radians
Surveyor	N 57d30' E	Surveyor (directional) units

**Note:**

A minute is 1/60 degree and a second is 1/60 minute. Grads and radians are simply alternate ways of measuring angles. A grad is a metric measurement equal to 1/100 of a right angle. Radians measure an angle by placing a length, equal to the radius, along the circle's circumference. Radians range from 0 to 2  $\pi$  instead of from 0 to 360 as degrees do. A radian is approximately 57.30 degrees. Surveyor units measure angles in directions, starting with north or south and adding an angle in a degrees, minutes, seconds format that shows how far the angle is from north or south and in which direction (east or west).

Choose the type of angle measurement you want. In the Precision box, click the arrow and a list of precision options drops down. Click the one you want.

Once you have chosen the angle type and the precision, click Next. As with the unit type, you can set the angle type without the wizard by choosing Format  $\leftrightarrow$  Units to open the Drawing Units dialog box



**Setting the angle measure and direction**

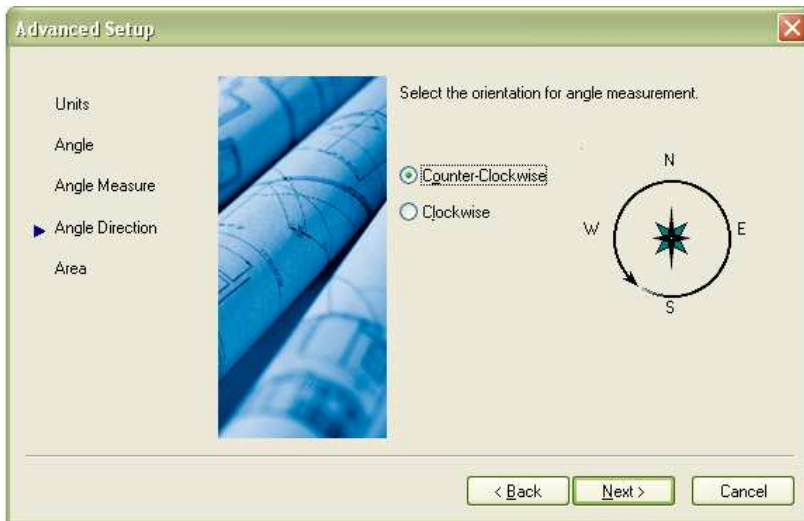
In the next step of the Advanced Setup Wizard, you set the Angle Measure.

According to standard convention, you measure angles so that 0 degrees starts to the right, in the East direction. To change this convention, choose the direction that you want for 0 degrees, and then click Next.

To set the angle measure without the wizard, choose Format  $\rightarrow$  Units. In the Drawing Units dialog box, choose Direction to open the Direction Control dialog box.

Here you can choose to have 0 degrees start in a direction other than East. You can also type any other angle or click the Pick an angle button to pick two points on your screen that specify an angle. Choose OK twice to return to your drawing.

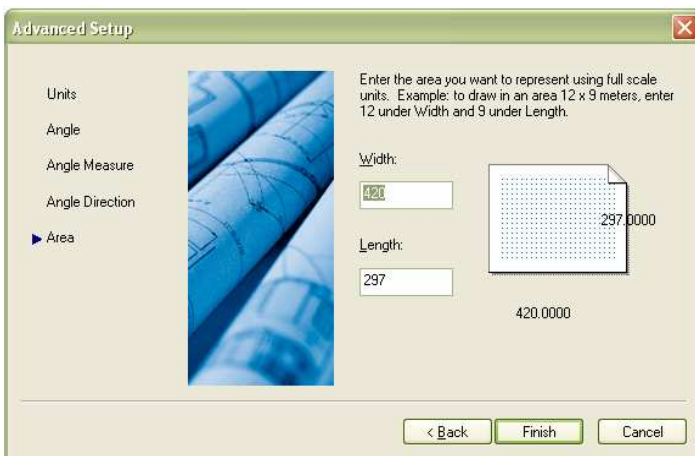
The next step of the wizard, Angle Direction is shown below.



By standard convention, degrees increase in a counterclockwise direction. To change the direction, choose Clockwise. Otherwise, leave the direction as Counter- Clockwise and click Next.

### Drawing Limits

The last step of the Advanced Setup Wizard, shown in below, enables you to specify the area of your drawing, also called the *limits*. The drawing limits are the outer edges of the drawing, specified in X,Y units. You need to set only the width and length of the drawing. Together, these two measurements create an invisible bounding rectangle for your drawing.



Almost universally, the lower-left limit is 0,0, which is the default. Therefore, the upper-right corner really defines the drawing size. The Setup Wizard assumes that you want to leave the lower-left corner at 0,0 and asks you only to specify the upper-right corner, using the Width and Length settings. Remember that you typically draw at life size (full scale) in AutoCAD. Therefore, the size of your drawing should be equal to the size of the outer extents of what you are drawing plus a margin for a



title block and perhaps for annotation and dimensioning. If you want to show more than one view of an object, as is common in both architectural and mechanical drawings, you need to take this into account.

To decide on the upper-right corner of your drawing *limits* (the width and length of your drawing); you need to consider what the drawing units mean for you.

Generally, the smallest commonly used unit is used, often inches or millimeters.

Therefore, if you are drawing a plan view of a house that is approximately 40-feet across (in the X direction) by 30-feet deep (in the Y direction), this translates to a top-right corner of 480,360. Adding room for a title block brings you to about 500,380.

**Note:** *You can draw outside the drawing limits. However, the drawing limits setting affects the size of the grid if you turn it on. This can help you visualize the size of your drawing if you don't have a title block. The ZOOM command with the All option also uses the drawing limits to resize the display, although it displays the entire drawing if the drawing is outside the limits. Setting the limits makes ZOOM All more useful.*

### **Setting drawing limits**

To set the drawing limits, type a width and a length in the Area step of the wizard.

AutoCAD shows the result in the picture to the right. Then click Finish to start drawing.

To set the drawing limits without the wizard, choose Format ⇨ Drawing Limits to start the LIMITS command. Press Enter to accept the lower-left corner default of 0,0 that appears on the command line. Then type the upper-right corner coordinate that you want and press Enter.

## **Scales**

### **Understanding scales**

You need to consider the fact that your drawing will most likely be plotted onto a standard paper (sheet) size. The standard orientation for drafting (and the default for most plotters) is landscape orientation, meaning that as you look at the drawing, the paper is wider than it is tall. These conventions have carried over from the days of hand drafting. (In a computer program, this is not really necessary, as you can rotate the drawing when you plot it.) To scale a drawing onto a piece of paper in a pleasing manner requires a rectangular shape that somewhat resembles the proportions of standard paper sizes.

In addition, although you specify the scale at plotting time, it helps to be aware of the scale you will use when plotting your drawing at the outset. The scales used for GIS (Geographic Information Systems), where you might be drawing an entire county, will be different from those used when drawing a house. The scales used in mechanical drafting are again totally different. In fact, in mechanical drafting, if you are drawing a very small object, such as a 2-inch screw-plate, you might scale up, that is, enlarge the drawing when plotting.

One important reason for establishing the scale at the beginning is to ensure that text, whether annotations or dimensions, is readable in its final plotted form. Applying a scale makes it possible to ensure that text remains a reasonable size even as the rest of the drawing is scaled up or down. In AutoCAD, scale also affects dotted and dashed lines as well as line widths.

Some drawings are not scaled. Examples are electrical or electronic schematics, piping diagrams, and railroad schematics. These drawings are representations of electrical or electronic connections but do not resemble the actual physical object that will eventually be created from the drawing. These drawings can be any size as long as they are clear and organized.

When determining your scale to try to fit a drawing on a sheet of paper, be aware that a plotter cannot print on the entire sheet. A certain amount of the margin around the edge is not available for drawing. Plotters differ in this respect. The plotter's manual can let you know the width of this unprintable margin. On average, you can assume a half-inch margin on each side; thus you should subtract one inch from both the length and width sheet measurements to determine the actual drawing space.

### **The MVSETUP Command**

The MVSETUP command is used in two different ways—to set up a drawing and to create viewports for paper space layouts.

Paper space layouts are a way of laying out your drawing in preparation for printing or plotting.

When you use MVSETUP to set up a drawing, this command simply provides a routine to walk you through some of the basic setup functions discussed in this lesson.

It is similar to the Quick Setup Wizard, but you type your responses on the command line. You can use MVSETUP when you start to customize AutoCAD to set up a drawing from a script file or AutoLISP program (topics covered in Parts VI and VII of this book). To use MVSETUP, type **mvsetup** on the command line and press enter. AutoCAD responds with the following prompt:

Enable paper space? [No/Yes] <Y>:

Type **n**; press enter to use MVSETUP without entering paper space.

The next prompt lets you enter the units type:

Enter units type: [Scientific/Decimal/Engineering/Architectural/Metric]:

Choose the option you want. Then AutoCAD displays a list of scale factors appropriate to the units option you chose. At the Enter the scale factor: prompt, type in a scale factor.

Finally, AutoCAD prompts you to set the drawing limits with the following two prompts:

Enter the paper width:

Enter the paper height:

After each prompt, enter a number based on the size of the paper you plan to plot on.

AutoCAD draws a rectangle of the size you indicated for the drawing limits.

## Preparing to Print

### Plot Devices

- Setting up a layout
- Scale and Size

### Plotting a Drawing

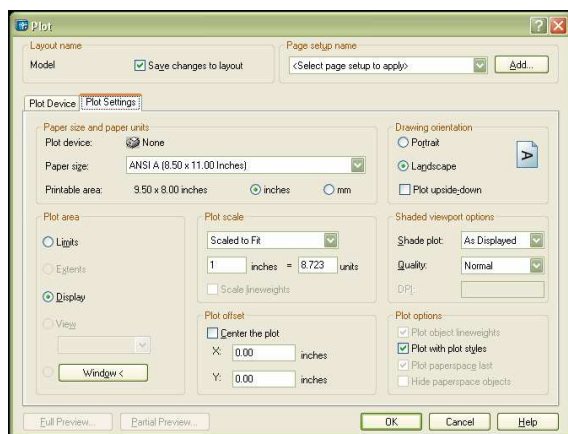
Once you have laid out your drawing, you are ready to plot it. The first step is to check the plotter or printer—it should be on, connected to your computer, and have the appropriate paper in it. Your drawing should appear on the screen.



To start plotting, choose Plot from the Standard toolbar to open the Plot dialog box, shown below with the Plot Settings tab on top. As you can see, this tab is almost identical to the Layout Settings tab of the Page Setup dialog box.

If you set the Layout Settings in the Page Setup dialog box and clicked OK, those settings were saved with the tab that was current at the time. You can usually just click OK in the Plot dialog box and plot immediately. In AutoCAD 2002, plot settings are saved in your drawing.

If you saved a page setup, you can choose it from the Page setup name drop-down list. If you want to make changes for one plot and not save them to the layout, clear the Save changes to layout checkbox. Then next time you plot, your earlier settings will be restored. If the box is checked, the settings you make now are saved to the layout.



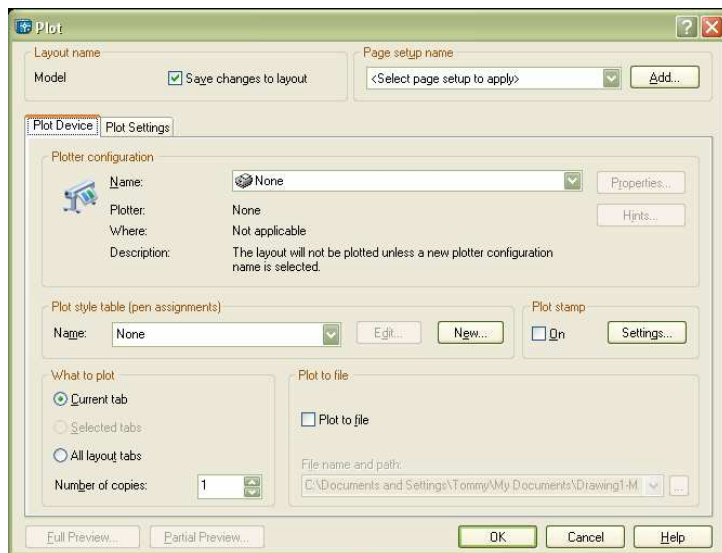
The Plot Settings tab of the Plot dialog box

## Choosing a plotter/printer

Choose the Plot Device tab, as shown below, to select a plotter or printer. Choose the plotter/printer you want from the list, and then click OK. A plotter must be either a Windows system printer or a configured plotter. To add a system printer, choose Start → Settings → Printers and double-click Add a Printer. To add a configured plotter, which has a driver especially to optimize the functioning of that plotter, use the Add Plotter Wizard. (Choose Tools → Wizards → Add Plotter.)

To avoid inadvertently plotting to a Windows system printer when you should be plotting to a plotter, you can hide the display of Windows system printers in the Plot and Page Setup dialog boxes. Because these printers won't appear on the list of plotters, you can't plot to them. To hide system printers, choose Tools → Options and click the Plotting tab. At the bottom-left corner of the dialog box, check Hide System Printers and click OK.

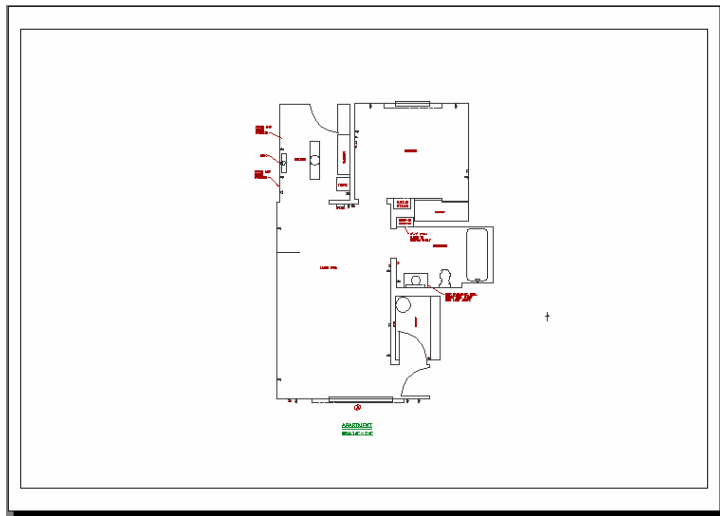
If you want to use a plot style table, check that it appears in the Plot Style Table drop-down list. In the "What to plot" section, you can choose to plot all layout tabs or selected tabs. By default, you plot the current tab, either the Model tab or one of the layout tabs. To choose how many copies you want to plot, change the number in the Number of copies text box of the "What to plot" section.



The Plot Device tab of the Plot dialog box

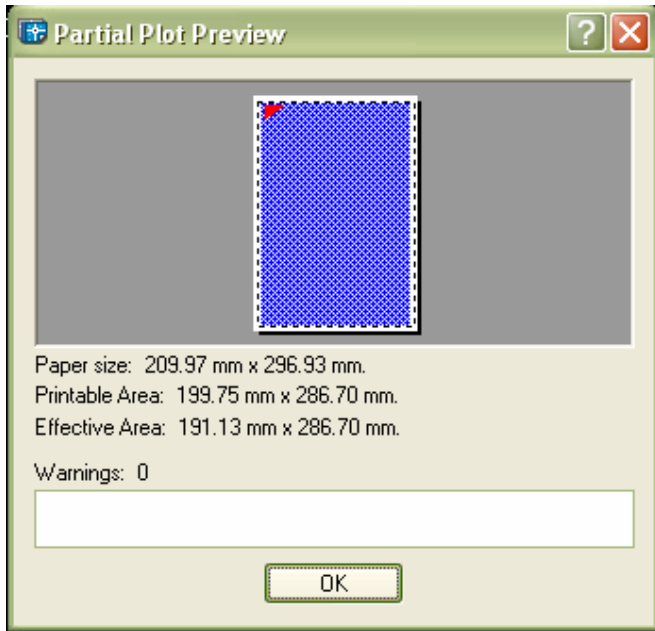
### Previewing your plot

You should preview your drawing before you plot. Choose Full Preview to see exactly how your drawing will plot. Here is an example of a full preview.



You can also choose Partial Preview, The solid area indicates the area of the paper that the plot will cover. The dashed line indicates the printable area of the paper. While the full preview takes more time, it is much more helpful.

The Print Preview button on the Standard toolbar is just like the one you have in your Windows word processor. When you choose Print Preview, you see a preview of your plot using current plot parameters. Right-click to open the shortcut menu that lets you plot, zoom, pan, or exit the preview.



A partial plot preview

## Plotting in Model Space

Figure 1 shows the new Plot dialog box. The area enclosed in red rectangle contains the new features.

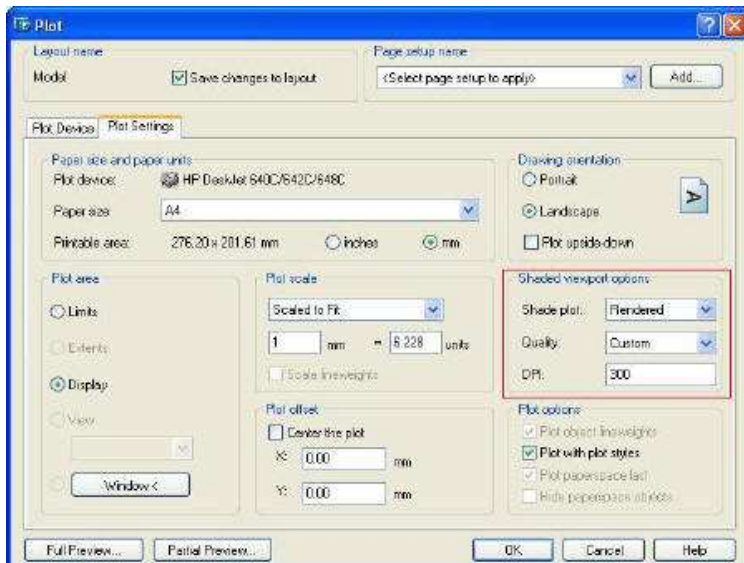


Figure 1

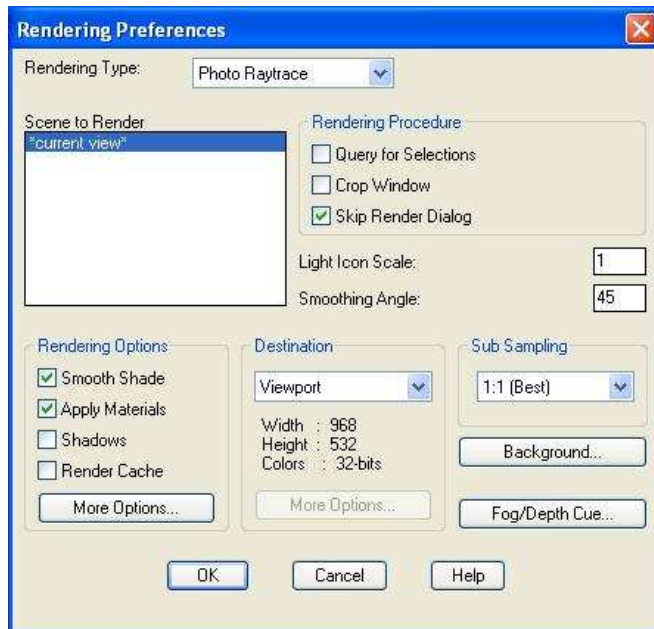
**When you are plotting a 3D model you can select one of the following options in Shade plot drop-down list [Figure 2]:**

- ▶ AsDisplayed
- ▶ Wireframe
- ▶ Hidden
- ▶ Rendered



**Figure 2**

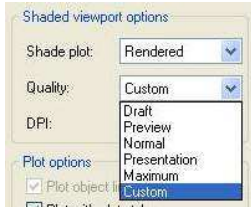
If the objects are shaded in the drawing area, selecting the first option results in a shaded plot. Select the Rendered option to plot objects as if they are rendered. The result depends on the render settings. You can set rendering options in Rendering Preferences dialog box [Figure 3]. Invoke RPREF command to view this dialog box.



**Figure 3**

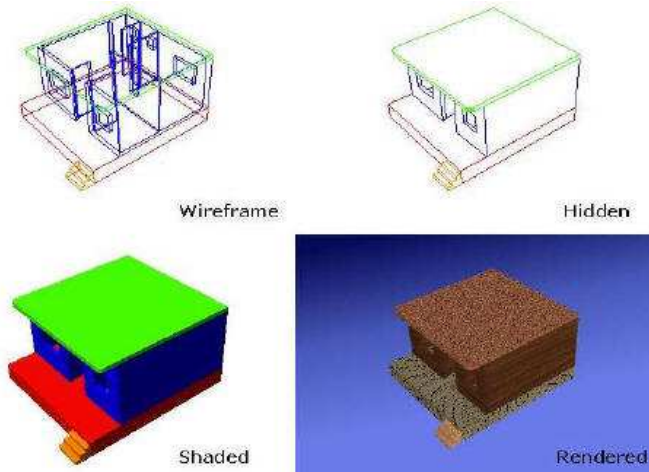
If you select either As Displayed or Rendered you have the chance to set the quality by one of the following options [Figure 4].

- ▶ Draft: Plots the models as Wireframe objects, regardless of the selected option.
- ▶ Preview: The quality of rendered or shaded plot will be up to 150 dpi.
- ▶ Normal: The quality of rendered or shaded plot will be up to 300 dpi.
- ▶ Presentation: The quality of rendered or shaded plot will be up to 600 dpi.
- ▶ Maximum: In spite of its name sets the quality to the current settings of the plotter not to the maximum capabilities.
- ▶ Custom: You can set the quality by enter a number in DPI edit box. The valid numbers that you can enter depend on the printer/plotter that is in use.



**Figure 4**

Figure 5 compares different plotting outputs for the same model.



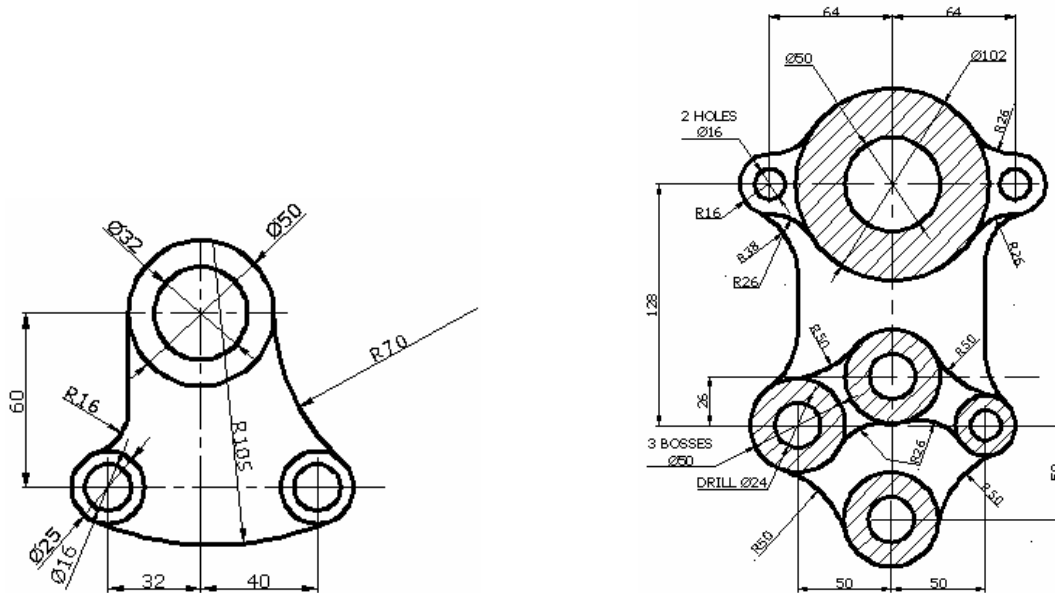
**Figure 5**

### Creating the plot

To start the plotting process, simply click OK in the Plot dialog box.

Assignments:

- 1.
- 2.



## Lesson 5

### Further Editing Commands

#### Editing and Modifying with grips

By now you should be familiar with grips. These are the 'blue squares' that get highlighted when you select an object outside of a command. Grips are handles that allow you to manipulate the object without entering a command directly. Be aware that grip editing can lead to undesired results if you are not careful.

The look of your grips can be changed by entering the **DDGRIPS** command.

Notice that you can enable or disable grips, as well as control whether they are enabled within blocks or not.

You can change the colors so that they are more visible depending on the particular drawing you are working in. You can also change the size of the grip if you need.

Notice that there are two different colors for grips, one when they are unselected (but visible) and another color when they are selected, or hot.

Usually these settings do not need to be changed.

When you first start working with grip editing, I recommend that you use it for modifying your dimensions. This is an easy way to get started, as it won't affect your geometry.

Once you have mastered grip editing with dimensions, try editing your geometry and you'll find that in many cases this can be quicker and easier than using the regular AutoCAD commands.

When you first select a grip and make it hot, it turns red and you are able to stretch it to where you need it to go. Always read your command line when grip editing.

Let us learn other object snaps that AutoCAD supports by doing another exercise. Our objective is to draw the model shown below.

#### Double-Clicking to Edit Objects

You can double-click objects to edit them. What happens after you double-click depends on the type of object. In most cases, double-clicking an object just opens the Properties window where you can change the object's properties. For example, double-clicking a polyline does not start the PEDIT command.

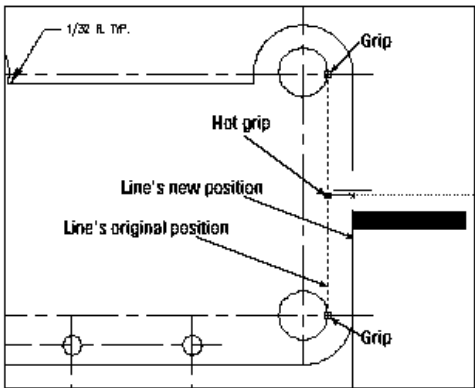
#### Grips

Grips offer a whole new way to edit objects—without choosing commands. By using grips, you can quickly stretch, move, rotate, scale, copy, and mirror objects. When you select an object without first choosing a command, the object appears highlighted with *grips*—small boxes at preset object snap points. (If you don't see grips, they may be turned off. See the "Customizing grips" section later in this chapter to learn how to turn them back on.) You can continue to select more objects in this way.

Note: You can deselect an object by pressing Shift and clicking it. (The grips remain unless you click it once more.) Other objects with grips continue to remain selected.

You then activate a grip by clicking it and use it to manipulate the object. When the grip is activated, it is filled in red (by default). An activated grip is also called a *hot grip*, as shown below. In some cases, you activate more than one grip at a time. To activate more than one grip, hold down Shift and then click the grips. If you activate a grip in error, click it again to deactivate it. Grips are so called because you can "hold on to" the object by dragging the grips with the mouse.

Once you activate a grip, right-click with the mouse to open the Grip shortcut menu listing all the grip options.



Moving a line. Grips appear at preset object snaps. A hot grip is used to manipulate an object.

You can also press the spacebar or Enter to cycle through five possible commands on the command line. It takes a little time to get used to working with grips, but once you do, they are a quick way to make certain changes. As long as you are familiar with the STRETCH, MOVE, ROTATE, SCALE, and MIRROR commands, you can easily learn how to accomplish the same edits using grips because the prompts are so similar. Once you complete the edit, the object remains highlighted and the grips remain, so you can further edit the object. If you want to edit another object, press Esc once to remove the grips. Then select another object or objects, or choose another command.

### Stretching with grips

Stretching with grips involves understanding how the grip points relate to the object. For example, you cannot stretch a line from its midpoint—if you think about it, there's no way to define in which direction to stretch the line. Also, you cannot stretch a circle. (You can only scale it.) Aside from these types of limitations, anything goes.

### Stretching one line

You can stretch one line. The result is similar to using the CHANGE command to change a line's endpoint. To stretch a line, select it. Click the grip at the endpoint you want to stretch. AutoCAD responds with **\*\* STRETCH \*\***

Specify stretch point or [Base point/Copy/Undo/eXit]: STRETCH is the first grip editing command on the command line. Simply specify the new endpoint for the line, using any method of specifying a coordinate. AutoCAD stretches the line. The other options work as follows:

- ❖ Base point lets you define a base point—other than the activated grip—and a second point. Right-click to open the Grip shortcut menu and choose Base Point. AutoCAD displays the Specify base point: prompt. Define a base point. AutoCAD responds with the original Specify stretch point or [Base point/Copy/Undo/eXit]: prompt. Define the second stretch point to stretch the line.
- ❖ Copy puts you in Multiple mode. Right-click to open the Grip shortcut menu and choose Copy. AutoCAD responds with the original Specify stretch point or [Base point/Copy/Undo/eXit]: prompt. AutoCAD keeps the original line and creates a new line stretched as you specify. You can then continue to create new stretched lines.
- ❖ Undo undoes the last edit. Right-click to open the Grip shortcut menu and choose Undo.
- ❖ eXit returns you to the Command prompt. Right-click to open the Grip shortcut menu and choose Exit. Esc also returns you to the Command prompt.

### Stretching multiple lines

Stretching more than one line at a time is similar to the most common use of the STRETCH command. However, it can also be somewhat confusing.

As explained earlier in this chapter for the STRETCH command, objects that cross the crossing window are stretched while objects entirely within the crossing window are moved. When you stretch multiple lines, you should activate endpoint grips to stretch lines and activate midpoint grips to move lines. Picking all those grips accurately can be difficult and time-consuming. Also, small objects close together create a lot of overlapping grips that are hard to select. For this reason, stretching multiple lines works best with simple models.

To stretch multiple lines, follow these steps:



1. Choose the objects you want to stretch. The objects are highlighted and display grips. You can use any method of choosing objects—you are not limited to crossing windows.
2. Hold down Shift and pick each grip that you want to stretch. If there are internal objects that you want to move with the stretch, select their grips, too—the midpoints of the lines and arcs, and the centers of the circles.
3. Release Shift and pick a grip to use as a base point. AutoCAD responds:  
**\*\* STRETCH \*\***  
Specify stretch point or [Base point/Copy/Undo/eXit]:
4. Specify a new stretch point. You can also use any of the other options.

At the end of this section on grips, you have the opportunity to try them out in an exercise.

### **Moving with grips**

Moving objects is easy using grips. Choose all the objects you want to move. Click any grip to activate it. This becomes the base point. Right-click to open the Grip shortcut menu and choose Move. You can also press the spacebar to cycle through the grip editing modes. For example, to move an object, press the spacebar once.

Once you have chosen Move editing mode, AutoCAD responds:

**\*\* MOVE \*\***

Specify move point or [Base point/Copy/Undo/eXit]:

Use any method to specify the second point. Be sure to use @ if you are typing in relative coordinates. AutoCAD moves the selected objects. The other options work as follows:

- ❖ Base point lets you define a base point—other than the activated grip. Rightclick to open the Grip shortcut menu and choose Base Point. AutoCAD displays the Specify base point: prompt. Define a base point. AutoCAD responds with the original Specify move point or [Base point/Copy/Undo/eXit]: prompt. Define the second move point to move the objects.
- ❖ Copy puts you in Multiple mode and lets you copy objects. Right-click to open the Grip shortcut menu and choose Copy (or type **c** ∞). AutoCAD responds with the original Specify move point or [Base point/Copy/Undo/ eXit]: prompt. AutoCAD keeps the original object and creates a new object where you specify. You can then continue to create new objects.
- ❖ Undo undoes the last edit. Right-click to open the Grip shortcut menu and choose Undo.
- ❖ eXit returns you to the Command prompt. Right-click to open the Grip shortcut menu and choose Exit. Esc also returns you to the Command prompt.

### **Rotating with grips**

Rotating with grips is very similar to using the ROTATE command. Choose all the objects you want to rotate. Click any grip to activate it. This becomes the base point. Right-click to open the Grip shortcut menu and choose Rotate. AutoCAD responds:

**\*\* ROTATE \*\***

Specify rotation angle or [Base point/Copy/Undo/Reference/eXit]:

Type in a rotation angle or pick a point. AutoCAD rotates the objects. The other options work as follows:

- ❖ Base point lets you define a base point—other than the activated grip. Rightclick to open the Grip shortcut menu and choose Base Point. AutoCAD displays the Specify base point: prompt. Define a base point. AutoCAD responds with the original Specify rotation angle or [Base point/Copy/Undo/Reference/eXit]: prompt. Define the rotation angle to rotate the objects.
- ❖ Copy puts you in Multiple mode and lets you copy objects. Right-click to open the Grip shortcut menu and choose Copy. AutoCAD responds with the original Specify rotation angle or [Base point/Copy/Undo/Reference/ eXit]: prompt. AutoCAD keeps the original object and creates a new rotated object where you specify. You can then continue to create new objects.
- ❖ Undo undoes the last edit. Right-click to open the Grip shortcut menu and choose Undo.
- ❖ Reference lets you specify a reference angle and a new angle. Right-click to open the Grip shortcut menu and choose Reference. AutoCAD responds with the Reference angle <0>: prompt. Type an angle or pick two points to specify an angle. AutoCAD displays the Specify new angle or [Base point/Copy/Undo/Reference/eXit]: prompt. Type an angle or pick a point. This works just like the Reference option for the ROTATE command.
- ❖ eXit returns you to the Command prompt. Right-click to open the Grip shortcut menu and choose Exit. Esc also returns you to the Command prompt.

### Scaling with grips

Scaling with grips is very similar to using the SCALE command. Choose all the objects you want to scale. Click any grip to activate it. This becomes the base point. Right-click to open the Grip shortcut menu and choose Scale. AutoCAD responds:

\*\* SCALE \*\*

Specify scale factor or [Base point/Copy/Undo/Reference/eXit]:

Type a scale factor. AutoCAD scales the objects. The other options work as follows:

- Base point lets you define a base point—other than the activated grip. Rightclick to open the Grip shortcut menu and choose Base Point. AutoCAD displays the Specify base point: prompt. Define a base point. AutoCAD responds with the original Specify scale factor or [Base point/Copy/ Undo/Reference/eXit]: prompt. Define the scale factor to scale the objects.
- Copy puts you in Multiple mode and lets you copy objects. Right-click to open the Grip shortcut menu and choose Copy. AutoCAD responds with the original Specify scale factor or [Base point/Copy/Undo/Reference/eXit]: prompt. AutoCAD keeps the original object and creates a new scaled object where you specify. You can then continue to create new scaled objects.
- Undo undoes the last edit. Right-click to open the Grip shortcut menu and choose Undo.
- Reference lets you specify a reference length and a new scale. Right-click to open the Grip shortcut menu and choose Reference. AutoCAD responds with the Reference length <0>: prompt. Type a length or pick two points to specify a length. AutoCAD displays the Specify new length or [Base point/Copy/Undo/Reference/eXit]: prompt. Type a length or pick a point. This works just like the Reference option for the SCALE command.
- eXit returns you to the Command prompt. Right-click to open the Grip shortcut menu and choose Exit. Esc also returns you to the Command prompt.

### Mirroring with grips

Mirroring with grips is very similar to using the MIRROR command. Choose all the objects you want to mirror. Click any grip to activate it. This becomes the first point of the mirror line. Right-click to open the Grip shortcut menu. Choose Mirror. AutoCAD responds:

\*\* MIRROR \*\*

Specify second point or [Base point/Copy/Undo/eXit]:

Specify the second point of the mirror line. AutoCAD mirrors the objects.

By default, AutoCAD erases the original objects. To keep the original objects, you must use the Copy option. This feature is the opposite of the MIRROR command, where the default is to keep the original objects.

The other options work as follows:

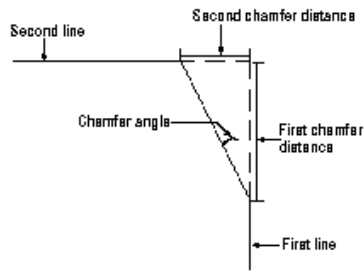
- Base point lets you define a base point—other than the activated grip—and a second point. Right-click to open the Grip shortcut menu and choose Base Point. AutoCAD displays the Specify base point: prompt. Define a base point—that is, the first point of the mirror line. AutoCAD responds with the original Specify second point or [Base point/Copy/Undo/eXit]: prompt. Define the second point of the mirror line, and AutoCAD mirrors the objects.
- Copy puts you in Multiple mode and lets you keep the original objects. Rightclick to open the Grip shortcut menu and choose Copy. AutoCAD responds with the original Specify second point or [Base point/Copy /Undo/ eXit]: prompt. AutoCAD keeps the original objects and creates new mirrored objects. You can then continue to create new mirrored objects.
- Undo undoes the last edit. Right-click to open the Grip shortcut menu and choose Undo.

eXit returns you to the Command prompt. Right-click to open the Grip shortcut menu and choose Exit. Esc also returns you to the Command prompt.

### Fillet and Chamfer

#### Creating chamfered corners

The CHAMFER command creates corners from two nonparallel lines. You can also chamfer xlines, rays, and polylines. You can simply extend the lines to meet at an intersection (a square corner), or create a beveled edge. If you create a beveled edge, you define the edge by either two distances or one distance and an angle relative to the first line you are chamfering. This figure shows the elements of a chamfered corner.



A chamfered corner

Chamfering is a two-step process. First you define how you want to chamfer the corner, specifying either two distances from the corner or a distance and an angle.

Then you select the two lines you want to chamfer. AutoCAD chamfers them using the information you specified.

To chamfer, choose Chamfer from the Modify toolbar. You cannot select objects before the CHAMFER command. AutoCAD responds with the (TRIM mode) Current chamfer Dist1 = 0.5000, Dist2 = 0.5000 Select first line or [Polyline/Distance/Angle/Trim/Method]: prompt. AutoCAD starts by listing the current settings. You can define two distances from a corner or one distance and an angle:

- To define two distances from the corner, right-click and choose Distance. At the Specify first chamfer distance <0.5000>: prompt, type the first chamfer distance or press Enter to accept the default (which is the last distance you defined). At the Specify second chamfer distance <0.5000>: prompt, type the second distance. The default for this is always the first chamfer distance because equal chamfer distances are so common.
- To define a distance (from the corner) and an angle, right-click and choose Angle. At the Specify chamfer length on the first line <1.0000>: prompt, enter a distance. This is the same as the first chamfer distance. At the Specify chamfer angle from the first line <0>: prompt, type the angle between the first line and the chamfer line.

To extend two nonparallel lines to make a corner, set the chamfer distances to zero and then chamfer the lines. AutoCAD extends them to meet. If they already intersect, AutoCAD trims them to create a corner. The pick points on intersecting lines should be on the part of the lines you want to keep (not on the part of the lines you want to trim off).

Now that you've specified the settings you want, you're ready to chamfer. Your distances or distance and angle are displayed as you just specified them. AutoCAD repeats the Select first line or [Polyline/Distance/Angle/Trim/

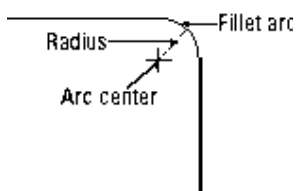
Method]: prompt. Select the first line. Unless you are creating a chamfer with equal distances, the order in which you select the lines is important. AutoCAD trims the first line selected by the first distance and the second line selected based on either the second distance or the angle. At the Select second line: prompt, select the second line. AutoCAD chamfers the lines.

By default, CHAMFER trims the original lines that it chamfers. If you want to keep the full original lines when you add the chamfer line, choose the Trim option and choose No Trim.

### Creating rounded corners

The FILLET command creates rounded corners, replacing part of two lines with an arc. Fillets are often used in mechanical drawings. In certain cases, you can use FILLET instead of the ARC command to create arcs. As with CHAMFER, you can fillet lines, xlines, rays, and polylines—they can even be parallel. You can also fillet circles, arcs and ellipses.

The FILLET command defines the fillet arc by its radius, as shown in this figure.



Like chamfering, filleting is a two-step process. First you define the radius of the fillet arc. Then you select the two lines you want to fillet. AutoCAD fillets them using the information you specified.



To specify the fillet information, choose Fillet from the Modify toolbar. You cannot select objects before the command with Fillet. AutoCAD responds with the Current settings: Mode = TRIM, Radius = 0.5000 Select first

object or [Polyline/Radius/Trim]: prompt. Right-click and choose Radius. At the Specify fillet radius <0.5000>: prompt, type the radius you want. The default is either 0.5000 or the last radius you specified.

AutoCAD repeats the Select first object or [Polyline/Radius/Trim]: prompt. Select the first object you want to fillet. At the Select second object: prompt, select the second object you want to fillet. AutoCAD creates the fillet.

By default, FILLET trims the original lines that it fillets, but the FILLET command recalls the last setting you used. If you want to keep the full original lines when you create a fillet, right-click and choose the Trim option, and then choose No Trim.

Filleting with a zero radius gives the same results as chamfering with distances set to zero.

The order in which you select the two objects to be filleted is not important. However, where you pick the objects is quite important. If two objects intersect, AutoCAD keeps the objects on the same side of the intersection as your pick point and fillets them. Those parts of the objects on the far side of the intersection are erased.

When you fillet arcs and lines, if more than one fillet is possible, FILLET connects the endpoints closest to your pick points. Filleting circles and lines can result in unexpected results. Sometimes you need to experiment to find the proper pick points.

## **Divide and Measure**

### **Measurement Commands**

AutoCAD can make calculations on objects for you. The DIVIDE command divides an object into equally spaced sections. The MEASURE command divides an object into sections of a specified length. These commands are useful in many fields—you might need to space bolt holes evenly around the edge of a bushing, or place fence studs along the edge of a plot every five feet.

### **Dividing objects**

The DIVIDE command divides an object into equally spaced sections. DIVIDE does not break the object—it simply places point objects along the object. You can then use the Node object snap if you wish to draw from those points.

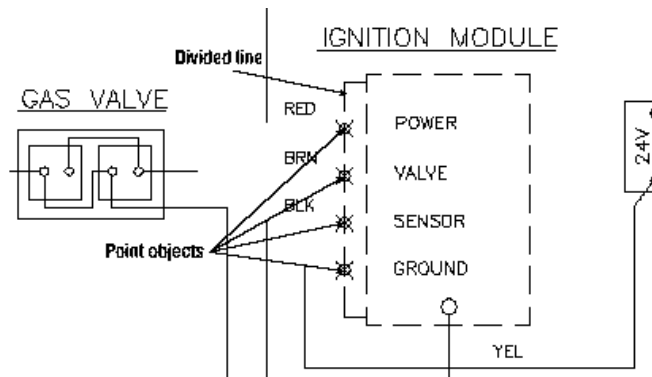
To divide an object, choose Draw ⇨ Point ⇨ Divide. Select the object you want to divide. AutoCAD responds with the Enter the number of segments or [Block]: prompt. Type in the number of segments you want to create. AutoCAD places the point objects and ends the command.

Remember that you can set the point display by choosing Format ⇨ Point Style. An easy-to-see point style is especially useful for the DIVIDE command. Specify the point style before using the command.

Remember that to create eight segments, for example, AutoCAD places seven point objects. If you have in your mind the number of point objects you want, add one when specifying the number of segments.

You can use the Block option to place a block of your choice along the object instead of a point object. If you choose the Block option (right-click and choose Block), AutoCAD responds with the Enter name of block to insert: prompt. Type the name of the block. AutoCAD asks, Align block with object? [Yes/No] <Y>. Answer Y or N depending on whether you want to align the block with the object.

AutoCAD asks for the number of segments you want, and you type that number, as described earlier. The figure below shows an electrical schematic. Here you want to divide a line so you can evenly space wires entering the ignition module. Four wires need to come in so the line was divided into five segments, using an easy-to-see point object.



### Measuring objects

The MEASURE command is similar to the DIVIDE command except that you specify the distance between point objects instead of the total number of segments.

AutoCAD starts measuring from the endpoint closest to where you pick the object. MEASURE does not break the object—it simply places point objects along the object. You can then use the Node object snap if you wish to draw from those points.

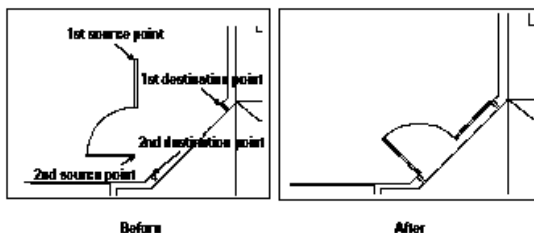
To measure an object, choose Draw  $\rightarrow$  Point  $\rightarrow$  Measure. Select the object you want to measure. AutoCAD responds with the Specify length of segment or [Block]: prompt. Type the segment length you want. AutoCAD places the point objects and ends the command.

Remember that you can set the point display by choosing Format  $\rightarrow$  Point Style. An easy-to-see point style is especially useful for the MEASURE command. Specify the point style before using the command.

As with the DIVIDE command, you can place a block along the object using the Block option. AutoCAD prompts you for the name of the block and lets you choose if you want to align the block with the object. AutoCAD asks for the segment length, and you type in the lengths between the points, as described earlier.

### Aligning objects

The ALIGN command lets you move and rotate an object or objects in one procedure. It is especially useful in 3D work. By specifying which points on an object move where, you can align the object with other objects in your drawing, as shown below:



Aligning requires several steps. Even so, it can save time when you need to both move and rotate at the same time, especially if you don't know the rotation angle you need. To align an object, choose Modify 3D  $\rightarrow$  Operation  $\rightarrow$  Align and select an object or objects. Alternatively, select the object or objects first and then choose Modify  $\rightarrow$  3D Operation  $\rightarrow$  Align.

Then follow these steps:

AutoCAD asks for the first source point. Specify a point, usually an object snap on the object you want to move.

AutoCAD asks for the first destination point. Specify the point where you want the first source point to end up.

AutoCAD asks for the second source point. If you press Enter, AutoCAD simply moves the selected objects. To continue to align, specify another point, usually an object snap on the object you want to move.

AutoCAD asks for the second destination point. Specify the point where you want the second source point to end up.

AutoCAD asks for the third source point. This is used for 3D alignment, to specify how you want to rotate the object in the third dimension. For 2D alignment, press Enter to continue the command.

AutoCAD displays the Scale objects based on alignment points? [Yes/No] <N>: prompt. If the distances between the source and destination points are not the same, type **y** if you want to scale the original object so the source and destination points match exactly.

### Resizing commands

There are four additional commands that resize objects. The TRIM and EXTEND commands bring the endpoint of an object to another object. LENGTHEN lets you lengthen or shorten a line, polyline, arc, or elliptical arc. STRETCH is used to stretch (larger or shorter) a group of objects, letting you change their direction at the same time.

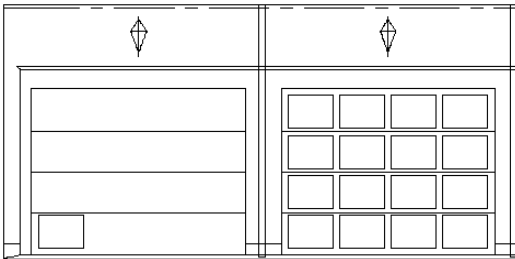
### Rectangular and Polar Arrays


#### Using the ARRAY command

The ARRAY command creates a rectangular or circular (polar) pattern by copying the object(s) you select as many times as you specify. The ARRAY command is a powerful drawing tool. It can quickly create large numbers of objects—saving a huge amount of time and effort.

#### Rectangular arrays

A rectangular array creates a grid of rows and columns of one or more objects. The figure below shows an example of a rectangular array.



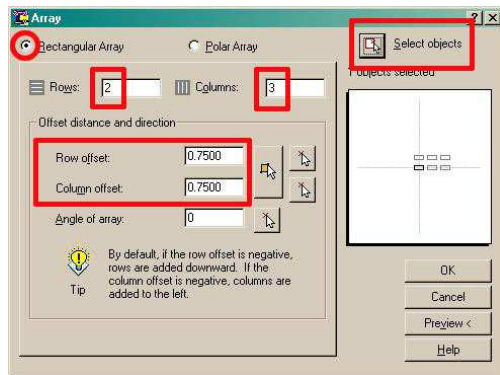
- 1)  Select the object or objects and choose Array from the Modify toolbar. Alternatively, choose Array from the Modify toolbar and select the object or objects. AutoCAD opens the Array dialog box, shown.
- 2) Click Rectangular Array at the upper-left corner of the dialog box (if it is not already selected).
- 3) If you have already selected one or more objects, the dialog box indicates the number of selected objects. If you have not selected any object, click Select Objects to return to your drawing and select objects. Press Enter to end object selection and return to the dialog box.
- 4) Type the number of rows and columns you want in the Rows and Columns text boxes. Press Tab after the last number you type to see the new result in the preview panel.
- 5) Type the row offset (the distance between the rows). The preview panel doesn't display any change when you change the row or column offsets.
- 6) Type the column offset (the distance between the columns).
- 7) If you want to specify the offsets by picking points on your screen, click one of the following buttons:

Pick Both Offsets  Pick Row Offset  Pick Column Offset 

- 8) If you want to change the angle of the array, type an angle. The preview panel displays the results of this value. To specify the angle by picking, click the Pick Angle of Array button.
- 9) To preview the array, click Preview. Then choose Accept to create the array, Modify to return to the Array dialog box, or Cancel to end the command.
- 10) To create the array, click OK.

If you change the Snap angle or the UCS, AutoCAD creates the rectangular array at the angle of the snap or UCS.

If you need to create a number of copies of an object along a straight path, use a one-column or one-row array instead of the COPY command. It's faster and easier.





The Array dialog box

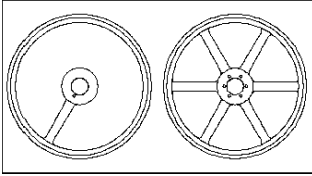
### Polar (circular) arrays

A polar array creates copies of one or more objects arrayed in a circle around a center point. An example of a polar array is shown below.

To create a polar array, follow these steps:

1.  Select the object or objects and choose Array from the Modify toolbar. Alternatively, choose Array from the Modify toolbar and select the object or objects. AutoCAD opens the Array dialog box.
2. Click Rectangular Array at the upper-left corner of the dialog box (if it is not already selected).
3. If you have already selected one or more objects, the dialog box indicates the number of selected objects. If you have not selected any object, click Select Objects to return to your drawing and select objects. Press Enter to end object selection and return to the dialog box.
4.  Specify the center point by typing X and Y coordinates or click the Pick Center Point button. If you selected an object first, check that the center point displayed is what you want.
5. Select the two items you want to specify from the Method drop-down box. You can choose any two from the three choices:
  - **Total Number of Items:** Sets the total number of items in the resulting array, including the one you are arraying.
  - **Angle to Fill:** Sets the number of degrees the polar array covers. For example, to array around half a circle, specify 180°.
  - **Angle Between Items:** Specifies the number of degrees between each item in the polar array.
6. Complete the values of the two items you specified. You can click the buttons to pick the angle to fill or the angle between items on the screen.
7. Check the Rotate Items as Copied checkbox to rotate the objects you are arraying. Uncheck the box to leave them unrotated.
8. Click Preview to preview the array. Then choose Accept to create the array, Modify to return to the dialog box, or Cancel to cancel the command.
9. Click OK to create the array.

You can specify which point on the last object selected AutoCAD uses to array the objects. AutoCAD makes a calculation of the distance from the center point of the array to a base point on the last object selected. By default, AutoCAD uses a default point based on the type of object selected. Table 10-1 lists the default base points. If you are arraying more than one object, you may not get the result you want without specifying the base point. Even for one object, you may want to change the base point used.

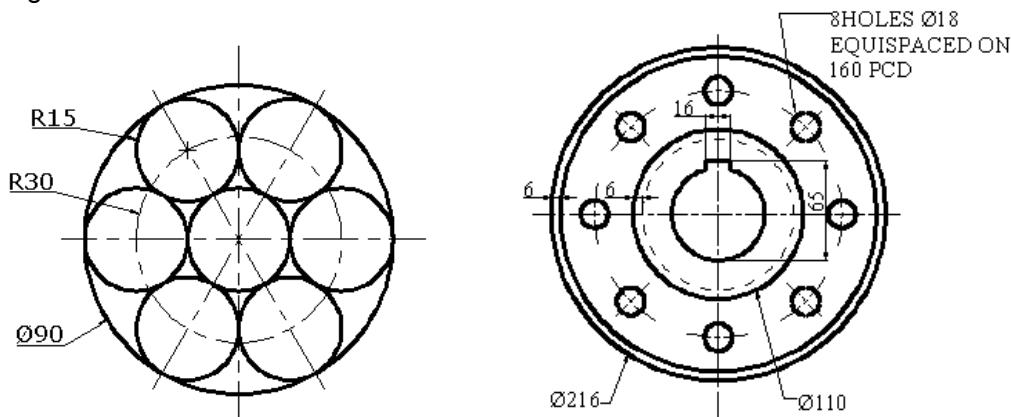


To specify the base point, click the More button at the bottom of the dialog box. The dialog box opens up to display the Object Base Point section. Uncheck Set to Object's Default and either type in coordinates or, more likely, click the Pick Base Point button and use an object snap to pick the desired point on an object. The preview box adjusts according to your choice but, because the preview only shows rectangles in the place of your object, the display may not be very helpful.

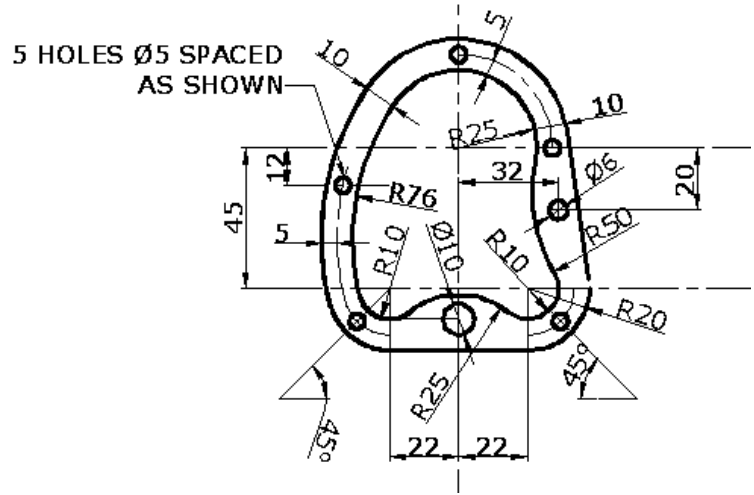
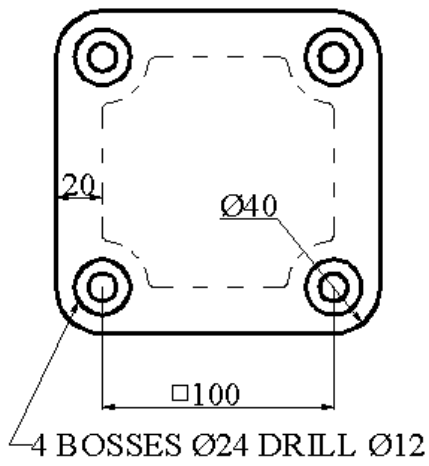
### Object Base Points for Arrays

Type of Object	Default Base
Arc, circle, or ellipse	Center point
Polygon or rectangle	First corner
Line, polyline (2D or 3D), ray, spline, or donut	Starting point
Text (single-line or paragraph), block	Insertion point
Construction line (xline)	Midpoint
Region	Grip point

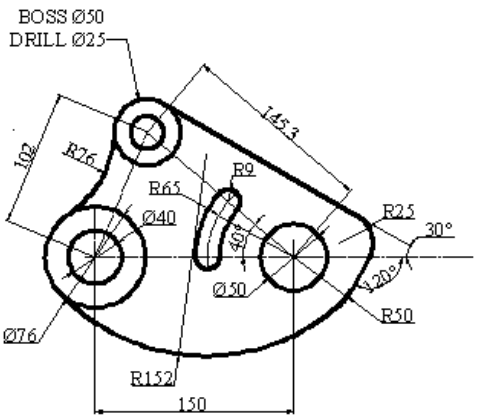
Assignments:



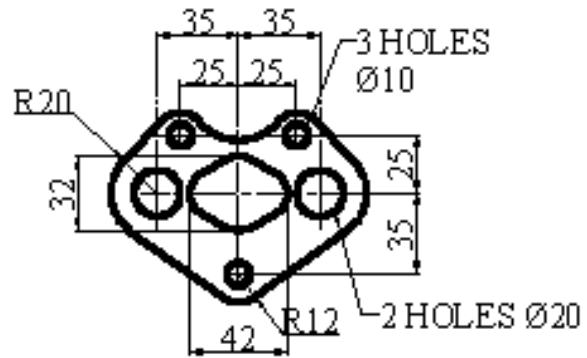




2.



3.



## Lesson 6

### Text

#### Single Line Text

Choose Draw → Text → Single Line Text. Follow the prompts:  
 Current text style: "ROMANS" Text height: 0'-4 1/2"  
 Specify start point of text or [Justify/Style]: *Right-click and choose Justify.*  
 Enter an option  
 [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:  
*Right-click and choose BC.*  
 Specify bottom-center point of text: *Use the Midpoint running object snap to pick a point on your drawing*  
 Specify rotation angle of text <0>: *Pick the endpoint.*  
 Enter text: **2-0** ↵  
 Enter text: ↵

Start the DTEXT command again. Follow the prompts:  
 Specify start point of text or [Justify/Style]: *Right-click and choose Justify.*  
 Enter an option  
 [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:  
*Right-click and choose Middle.*  
 Specify middle point of text: *Pick another point. (This point doesn't have to be exact.)*  
 Specify rotation angle of text <45>: **0** ↵  
 Enter text: **%%UMASTER BATH** ↵  
 Enter text: ↵

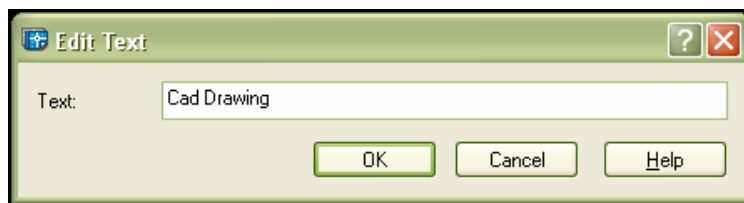
#### Using the TEXT command

The TEXT command was AutoCAD's original command for creating text. DTEXT was an update, showing the characters on your screen as you typed.

#### Editing single-line text

As with any drawing object, the need often arises to edit your text. You have two ways to edit single-line text.

You can use the DDEDIT command. Choose Modify → Objects → Text → Edit. At the Select an annotation object or [Undo]: prompt, select any text. Remember that each line of text created with DTEXT or TEXT is a separate object. AutoCAD opens the Edit Text dialog box with your text highlighted in an edit box, as shown below. You can start typing to completely replace the text or click where you want to change part of the text and use standard Windows techniques to edit the text. Click OK or press Enter to make the changes and return to your drawing.

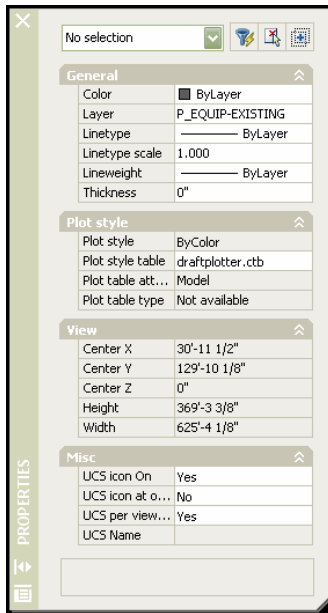


The advantage of this method is that DDEDIT continues to prompt you for additional annotation objects to edit, making it efficient for editing several lines of text at once. Press Enter to end the command.



You can also change text using the Properties window. Select any text object and click Properties on the Standard toolbar to open the Properties window, as shown. Here you can edit not only the text content but also every other conceivable property, including layer, linetype, linewidth, color, insertion point, justification, rotation angle, and several other properties that I

cover in the next section on text styles. You can only edit the contents of one text object at a time this way.



### Scaling text

You can now change the scale of selected text (either one text object or several at once) without moving the text insertion point.

To use SCALETEXT, follow these steps:

1. Choose Modify - Object - Text - Scale.
2. Select the text objects to scale.
3. At the Enter a base point option for scaling Existing/Left/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Existing>: prompt, press Enter to use the existing insertion point of the selected text or choose a new base point. (Your last choice for this prompt becomes the new default, so if you used another option, type **e** ↵.) These options are the same as the Justify options described earlier in this lesson.
4. At the Specify new height or [Match object/Scale factor] <3/32>: prompt right-click and choose Scale factor to specify a scale factor, just as you would for the SCALE command. You can also type a new height or use the Match object option to match the height of the selected text objects to another existing text object. AutoCAD prompts you to select an object with the desired height.
5. If you chose the Scale factor option, type the factor you want at the Specify scale factor or [Reference] <2>: prompt.
6. If you want to specify the scale factor with reference to existing text or a value, use the Reference option. At the Specify reference length <1>: prompt, type a length or specify two points that measure the reference length.
7. At the Specify new length: prompt, type a value or pick two points to indicate the new length.

### Justifying text

The new JUSTIFYTEXT command lets you change the justification of selected text objects without moving the text.

To use JUSTIFYTEXT, Choose Modify - Object - Text - Justify. Then select the text objects you want to modify. At the Enter a justification option

[Left/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Left>: prompt, right-click and choose the justification you want.

### Understanding Text Styles

You certainly do not always want to use AutoCAD's default font.

AutoCAD lets you create text styles that give you full creative control over the font, font style (bold, italic, or both), character width, obliquing angle, and text height. You can even design backward,

upside-down, and vertical text. (Vertical text is like the text you occasionally see on the spine of a book. It goes down instead of to the right.)

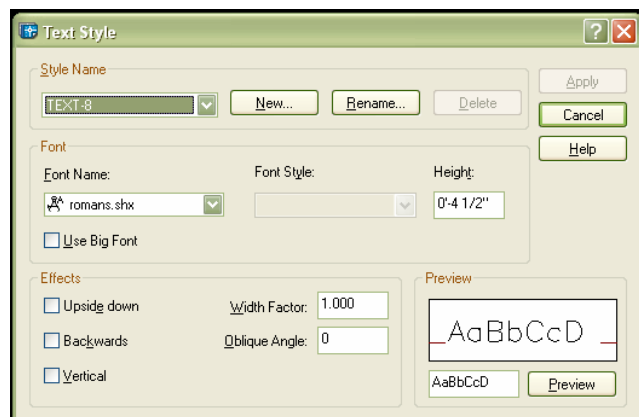
Text styles are similar to layers. Like a layer, each text style:

- Has a name and several properties.
- Is saved with the drawing.
- Can be made current when you want to use it.
- Can be renamed and deleted.

Creating text styles is part of the typical drawing setup procedure. You should include text styles in your drawing templates.

### Creating a new text style

To create a new text style, choose Format Text Style. This starts the STYLE command and opens the Text Style dialog box, shown below.



Choose New to open the New Text Style dialog box. Type the name of your new text style and click OK. Text style names can be up to 255 characters and can include spaces. AutoCAD returns you to the Text Style dialog box where you define the new text style.



### Font

In the Font section of the Text Style dialog box, you specify the font, font style, and text height. Click the Font Name drop-down list arrow to see the list of fonts. Fonts with the double-T icon are TrueType fonts. The other fonts are AutoCAD's own fonts. AutoCAD fonts are defined in a shape file that has the file name extension *.shp* and are compiled into a file with the extension *.shx* for faster access.

Click a font to choose it. You can see a preview of the font in the Preview section of the dialog box. The Preview section also has an edit box. Type some characters and click Preview to see the preview of those characters.

If the font you have chosen supports different styles such as bold or italic, you can choose one of them in the Font Style drop-down box. None of the AutoCAD fonts supports font styles, but many of the TrueType fonts do.

Type in the height you want for your font. Remember to take into account the scale factor if necessary.

You can leave the height at zero if you want to be able to vary the text height within that one style. If the height is zero, AutoCAD prompts you for a height whenever you use the DTEXT or TEXT command.

### Effects

In the Effects section, you specify the orientation, width factor, and oblique angle of the text style.

The default width of characters is set to 1. You can specify a smaller number to compress text and a larger number to expand it, as shown.

Width = 1.5 Bearing Housing  
 Width = .8 Bearing Housing

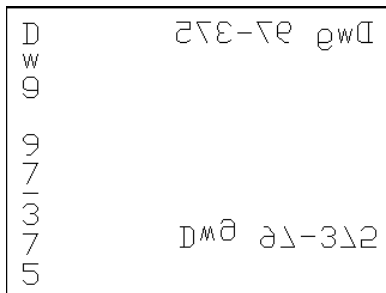
The oblique angle refers to the angle of the individual letters. It is generally used to create an effect such as italic text. Of course, you don't need to use an oblique angle if you are using a TrueType font that supports italic text.

The angle definition used to define oblique text is different from the angle definition used for other objects. Up and down text—that is, normal text—is a zero oblique angle. A positive angle slants the text to the right—typical for italic text. A negative angle slants the text to the left. Below shows text with a positive and negative oblique angle.

Oblique angle = -10 Bearing Housing

Oblique angle = 10 Bearing Housing

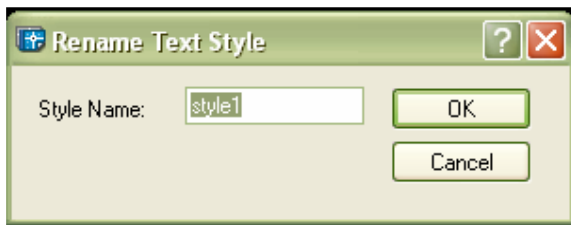
You can create text that is backward (like a mirror image) or upside-down. Some fonts let you create vertical text. Figure below shows an example of each kind of text. Check the appropriate checkbox to create the effect you want.



When you have completed defining your text style, click Apply to make it current. Click Close to return to your drawing.

### Renaming and deleting text styles

You can rename and delete text styles easily. To rename a text style, start the STYLE command to open the Text Style dialog box. Choose Rename to open the Rename Text Style dialog box, as shown below. It works just like the New Text Style dialog box.



To delete a text style, choose it from the Style Name drop-down list of the Text Style dialog box and click Delete. AutoCAD asks you to confirm. Click Yes to delete the text style. You cannot delete a text style that is being used.

### Changing a text style

To change a style, choose Format - Style. From the Style Name drop-down list, choose the text style you want to change. Make changes the same way you did when creating the style. Choose Apply, and then Close. AutoCAD regenerates the drawing and changes all text that uses the style you changed. This is a powerful way to control the look of text in your drawing.

To change existing text to another text style, choose Properties from the Object

Properties toolbar and select the text. In the Properties window, choose a new text style in the Text Style drop-down list.

### **Making a style current**

You choose the current style when you use one of the text commands. If you use DTEXT or TEXT, AutoCAD displays the Specify start point of text or [Justify/Style]: prompt. Right-click and choose Style. (AutoCAD displays the current style and height before the prompt.)

If you know the name of the style you want to use, type it and press Enter. AutoCAD repeats the Specify start point of text or [Justify/Style]: prompt. You can choose the Justify option or pick a start point to continue the command.

If you don't know the name of the style you want to use right-click and choose? from the shortcut menu. AutoCAD responds with the Enter text style(s) to list <\*>: prompt. Press Enter and AutoCAD lists all the text styles with their properties and then repeats the Specify start point of text or [Justify/Style]: prompt. Here is a sample listing:

Text styles:

Style name: "Notes" Font files: romans.shx  
 Height: 0'-0 1/8" Width factor: 0.950000 Obliquing angle: -10  
 Generation: Normal  
 Style name: "ROMANS" Font files:  
 C:\ACADWIN\FONTS\ROMANS.SHX  
 Height: 0'-0 1/8" Width factor: 1.000000 Obliquing angle: 0  
 Generation: Normal  
 Style name: "SIMPLEX" Font files: simplex.shx  
 Height: 0'-0 1/8" Width factor: 1.000000 Obliquing angle: 0  
 Generation: Normal  
 Style name: "STANDARD" Font files: txt  
 Height: 0'-0 13/64" Width factor: 1.000000 Obliquing angle: 0  
 Generation: Normal

To change the current style, right-click and choose Style again and type the style name from the list you just displayed. Then continue to use the Justify option or to pick a start point.

If you use MTEXT, the Multiline Editor opens, as explained in the next section. Choose the Properties tab and choose the text style you want from the Style drop-down list.

### **Importing a text style**

You can use the AutoCAD DesignCenter to import features from other drawings. To import a text style, follow these steps:

1. Choose AutoCAD DesignCenter from the Standard toolbar to open the DesignCenter.
2. In the left pane, navigate to the drawing that has the text style you want.
3. Double-click the drawing icon or click its plus sign.
4. To see the list of the text styles, double-click the text styles icon in either the left or right pane.
5. Double-click the text style's icon to import it into your drawing.
6. Click the DesignCenter's Close button to close the DesignCenter.

### **Creating Multiline Text**

Single-line text is awkward when you want to type quite a bit of text. The main disadvantage is that single-line text does not use word wrap, a feature that wraps text to the next line to keep a neat right margin. Multiline text (also called paragraph text and not to be confused with multilines) solves this problem and also offers many more formatting options compared to single-line text. The entire paragraph of multiline text is one object.

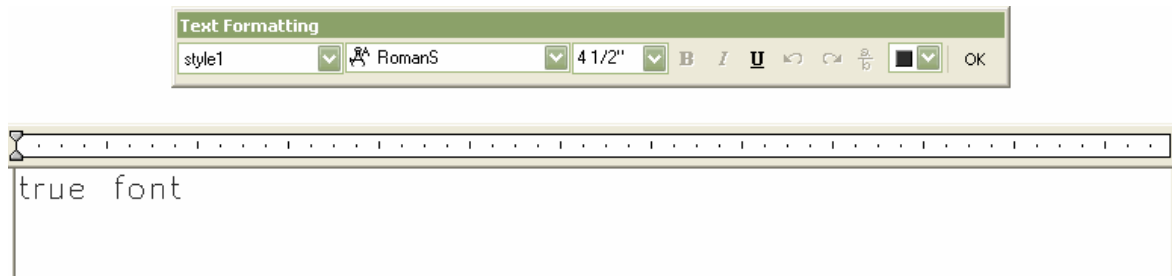
The edit box you use to create multiline text resembles Windows word processors and is resizable. You use this box both to create and also to edit text and its properties.

### Using the Multiline Text Editor

To create paragraph text, choose Multiline Text from the Draw toolbar. This starts the MTEXT command. AutoCAD tells you the current style and text height. For example:

Current text style: ROMANS. Text height: 4 1/2" of a bounding box to specify where to place the text. At the Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width]: prompt, specify the diagonally opposite corner of the bounding box. You can also choose one of the other options to specify the text properties before you type in the text. However, these options (explained later in this chapter) are also available in the Multiline Text Editor, which opens once you have specified the bounding box.

The Multiline Text Editor is shown in below.



Type your text in the large edit box. The Multiline Text Editor wraps the text to the next line when AutoCAD senses that the text has met the right side of the bounding box you specified. Although you have created a bounding box with four sides, AutoCAD only limits the text by the paragraph width, that is, the left and right margins.

If you type too much text for the bounding box, AutoCAD scrolls the text in the Text Editor. You can also enlarge the Text Editor by dragging on its bottom edge.

### Creating stacked fractions automatically

You can create automatic stacked fractions and tolerances as you type, using a system similar to those described earlier for creating special characters with DTEXT/TEXT.

You can also type unstacked fractions (as in **1/2**), select the fraction text, and then click Stack/Unstack on the Character tab. You can still stack fractions in this manner.

## Lesson 7

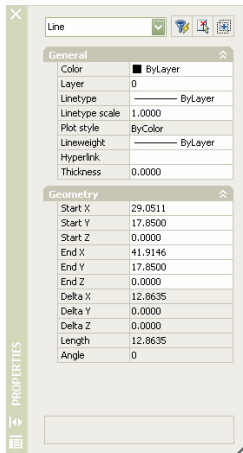
### Object Properties Manager

#### Editing with the Object Property Manager

The Object Property Manager is a window that displays the properties of selected objects. You can use the Object Property Manager (which can also be called the Properties window or OPM) to change the properties of objects as well.



To open the Object Property Manager window, click Properties on the Standard toolbar. The Properties window opens on your screen.



The Properties window can be used to directly edit objects—called Geometry in the Properties window—but can be used for editing other object properties as well:

- You can change the layer, color, linetype, linetype scale, and lineweight of objects.
- You can edit text and text properties.
- You can edit plot styles.
- You can edit hyperlinks.

The Properties window contains two tabs. The Categorized tab organizes the types of properties you can edit by category. In the properties window above, you see the result of opening the Properties window with one line selected. Use the Geometry category to edit the line. The Alphabetic tab contains the same information organized alphabetically.

To change either endpoint of the selected line, just type new coordinates in the Properties window and press Enter. Of course, this method of editing a line is only useful when you know the absolute coordinates. When you select a geometric property, such as Start X, the X coordinate of the line's start point, an arrow button appears. You can click this button and then pick a new start point directly on your screen. The new point becomes the new start point of the line, changing both the X and Y coordinates of the start point if your pick point requires it.

*Note: The Properties window has its own undo function. Right-click in the Properties window and choose Undo. Multiple levels of undo are available. You can dock the window if Allow docking is checked on the shortcut menu (right-click in the Properties window). Then drag the window to the left or right of your screen. You can resize the window by pointing to its edge until you see the double-headed arrow cursor, and then clicking and dragging.*

The information you see in the Properties window depends on whether objects are selected:

- If you open the Object Properties Manager with no object selected, you see only properties that apply to the entire drawing, such as UCS, current layer, and viewport data. You can select an object while the window is open to display data on that object.
- If one object is selected, you see both general information and geometrical information on that object.
- If more than one object is selected, you see general information such as layer, color, and linetype data, but no geometry data.



- If more than one object is selected and you choose one type of object from the drop-down list at the top of the window, you see information applying only to that type of object. This drop-down list provides a quick way to filter the type of objects you can edit.

Click the filter button at the top-right corner of the window to open the Quick Select dialog box. The Quick Select dialog box lets you create a selection set according to specified criteria—for example, all green lines. You can then change the properties of these green lines.

The Properties window has buttons at its top-right corner.

The Toggle Value of PICKADD Sysvar button turns PICKADD on and off. PICKADD changes how you select more than one object. When it's on (set to 1), the default, you can continue to select objects and they are added to the selection set. When it's off (set to 0), you need to press Shift to add objects to the selection set; otherwise, selecting an object deselects previously selected objects.

When PICKADD is off, each time you pick another object previous objects are deselected.

However, if the previously picked objects have grips, the grips remain.

Toggle buttons can be confusing. Here's how to change the PICKADD variable:



When you see the 1, PICKADD is off. Click the button to turn PICKADD on. When you see the plus, PICKADD is on. Click the button to turn PICKADD off.



The Select Objects button lets you select objects. If you're going to pick objects or use implied windowing, the Select Objects button is no advantage.

You can just select the objects and apply changes in the Properties window.

However, if you want to use a fence or polygon method of selection, the Select

Objects button is helpful. Follow these steps:

1. Click the Select Objects button.
2. On the command line, type the selection method you want (such as **f** for fence) as you would if a command were active or you had started the SELECT command.
3. Select the objects you want.
4. Press Enter to end object selection.

Use the Properties window to make the desired changes. (You can also start an editing command to use with the selected objects.)

Click the Object Property Manager's close box to close the window.

## Lesson 8

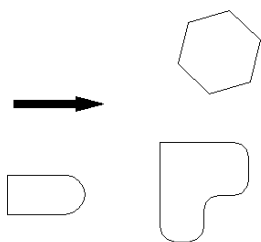
### Creating and Modifying Polylines

#### Creating and Editing Polylines

Polylines are single objects that combine line segments and arcs. In certain situations, it is useful to be able to edit an entire set of lines and arcs as one object. Polylines can have a width, which can vary from the start point to the endpoint. Polylines ensure that all the vertices of a closed area actually meet, which is very helpful if you want to hatch the area. They are also very useful for 3D drawing. In short, polylines are a neat, clean way to draw. (Technically, polylines are called lightweight polylines to distinguish them from the type of polylines used in Release 13 and earlier.)

The RECTANG and POLYGON commands create polylines.

Some examples of polylines:



#### Using the PLINE command

To draw a polyline, choose Polyline from the Draw toolbar. This starts the PLINE command. AutoCAD responds with the Specify start point: prompt. Specify the start point. Then AutoCAD responds with the Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: prompt. It offers the following options:

- **Arc:** Enables you to draw arcs. This option opens up a set of arc suboptions, which are explained next, after this list.
- **Close:** Closes a polyline by drawing a line from the endpoint of the last line segment to the start point of the polyline. This option appears only after you have picked a second point.
- **Halfwidth:** Defines half the width of the polyline—the distance from the center of the polyline to its edge. AutoCAD asks you for the starting halfwidth and the ending halfwidth, enabling you to create polylines that are tapered.
- **Length:** Specifies the length of the next line segment. AutoCAD draws the line segment in the same direction as the last line segment or tangent to the last arc.
- **Undo:** Undoes the last line segment.
- **Width:** Defines the width of the polyline. AutoCAD asks you for the starting width and the ending width.
- **Specify next point:** Enables you to create a line segment. This is the default option.

Like the LINE command, PLINE continues to prompt you for more points, repeating the entire prompt each time. When you are done, press Enter to end the command.

If you choose Arc, AutoCAD responds with the Specify endpoint of arc or [Angle/Center/Close/Direction/Halfwidth/Line/Radius/Second pt/ Undo/Width]: prompt. While this may seem overwhelming, most of the options are similar to the ARC command options. The arc options are as follows:

- ❖ **Angle:** Specifies the included angle.
- ❖ **Center:** Specifies the arc's center.
- ❖ **Close:** Closes the polyline by drawing a line from the endpoint of the last arc to the start point of the polyline.
- ❖ **nDirection:** Specifies the direction of the arc from the start point.
- ❖ **Halfwidth:** Defines half the width of the polyline—the distance from the center of the polyline to its edge. AutoCAD asks you for the starting halfwidth and the ending halfwidth.
- ❖ **Line:** Returns you to the main polyline prompt so you can draw line segments.

- ❖ **Radius:** Specifies the arc's radius.
- ❖ **Second pt:** Specifies the second point of the arc.
- ❖ **Undo:** Undoes the last arc.
- ❖ **Width:** Defines the width of the polyline. AutoCAD asks you for the starting width and the ending width.
- ❖ **Specify endpoint of arc:** Specifies the endpoint of the arc. This is the default.

AutoCAD creates an arc tangent to the previous arc (continuing in the same direction). PLINE continues to display the arc submenu until you use the Line suboption or end the command by pressing Enter

### Editing polylines with the PEDIT command

Because polylines can be quite complex, AutoCAD has a special command to edit them, PEDIT. To edit a polyline, choose Modify ⇨ Object ⇨ Polyline. AutoCAD responds with the Select polyline or [Multiple]: prompt. Select a polyline and AutoCAD responds with the Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: prompt. The options are as follows:

- **Close:** Closes an open polyline. If necessary, it adds a segment to connect the endpoint to the start point. If the polyline is already closed, this prompt becomes Open. Open creates a break between the first and last segments of the polyline.
- **Join:** Joins touching lines, arcs, or other polylines to the polyline.
- **Width:** Enables you to specify one width for the entire polyline.
- **Edit Vertex:** Provides a set of suboptions for editing vertices. These are explained after this list.
- **Fit:** Turns the polyline into a curve that passes through the vertices.
- **Spline:** Creates a curve using the vertices as control points. The curve does not usually actually pass through the vertices. This is not the mathematically exact spline that the SPLINE command produces.
- **Decurve:** Returns a Fit or Spline curve to its original vertices.
- **Ltype gen:** Turns on continuous linetype generation for the selected polyline.
- **Undo:** Undoes the most recent edit.

You can change any line or arc into a polyline. Start PEDIT and choose a line or arc. AutoCAD responds: Object selected is not a polyline. Do you want to turn it into one? <Y>. Press Enter to accept the default. AutoCAD turns the object into a polyline. You can use this technique to turn a series of connected lines and arcs into a polyline. First, turn one of the objects into a polyline as I just explained. Then use the Join option and select the other objects individually or by a selection window. When you finish object selection, AutoCAD tells you how many segments were added to the polyline. In order to create a polyline in this way, the individual lines and arcs must connect exactly end to end.

When you choose the Edit Vertex option, AutoCAD offers a new set of suboptions, with the Enter a vertex editing option [Next/Previous/Break/Insert/ Move/Regen/Straighten/Tangent/Width/eXit] <N>: prompt. You see an x at one of the vertices. This is the current vertex, which you can edit. The suboptions are as follows:

- **Next:** Moves you to the next vertex so you can edit it.
- **Previous:** Moves you to the previous vertex.
- **Break:** Enables you to break the polyline. You can choose the Go suboption to break the polyline into two (although you can't see the break). You can move to another vertex using the Next or Previous suboptions and then choose Go. AutoCAD breaks the polyline between the original vertex and the vertex you moved to. Use the eXit suboption to return to the previous prompt. (You can also use the BREAK command.)
- **Insert:** Enables you to insert another vertex. AutoCAD prompts you for its location.
- **Move:** Enables you to move the vertex. AutoCAD prompts you for its location.
- **Regen:** Regenerates the polyline.
- **Straighten:** Deletes vertices. This works like the Break option with the same Next, Previous, Go, and eXit suboptions.

Once you move to a new vertex, AutoCAD draws a straight line between it and the original vertex. If you don't move to a new vertex, this option affects only an arc by changing it to a straight line segment.

- **Tangent:** Specifies a direction from the vertex. AutoCAD uses this if you choose the Fit option.
- **Width:** Enables you to specify a starting and ending width of the segment starting with the current vertex.
- **eXit:** Exits this group of suboptions.

You can make many changes during the PEDIT session. If you return to the command line and use the U or UNDO command, the entire session is undone. If you want to undo only part of the session, use the Undo options of the PEDIT command.

You can now edit more than one polyline at a time, including 2D and 3D polylines, using the PEDIT command. To edit multiple polylines, start the PEDIT command and use the Multiple option (type **m** or right-click and choose Multiple from the shortcut menu) at the first prompt. You see the Enter an option [Close/Open/Join/Width/Fit/ Spline/Decurve/Ltype gen/Undo]: prompt. Then choose the option you want. For example, you can change the width of all the selected polylines or apply the Spline option to them.

You can also now join two polylines that are not touching—if you use the Multiple option first. Select the polylines and then choose the Join option. You then need to specify two suboptions:

- **Fuzz distance:** The maximum distance that the endpoints of the polylines can be from each other. In other words, in order for the join to work, the fuzz distance must be greater than the distance of the endpoints. If you want to join the endpoints no matter what, type in a very large number.
- **Jointype:** The method of joining polylines. You can use the Extend method, which extends (or trims) the segments to the nearest endpoints, or the Add method, which adds a straight segment between the two nearest endpoints.

You can choose the Both suboption, which tries to extend or trim, but, if it can't, it adds a segment.

### Editing polylines using the Properties window

You can also edit polylines in the Properties window. Double-click a polyline, and AutoCAD opens the Properties window for Polylines, shown in the figure below.



*The Properties window with a polyline selected*

Notice that you can choose a vertex by clicking Vertex in the Geometry section of the dialog box. An **x** appears on the polyline to let you know which vertex you have chosen. You can then edit the vertex's coordinates by typing them in, or by clicking the Pick a Point button that appears and picking a point in the drawing. You can change the start and ending width and specify a global width for the entire polyline.

Notice that you can close and open a polyline in the Misc section. You can also turn on continuous linetype generation for the selected polyline by choosing Linetype generation and choosing Enabled from the drop-down list that appears. Of course, you can change the layer, color, linetype, lineweight, and linetype scale as well.

## Drawing Rectangles

Use the RECTANG command to create a rectangle by specifying the two diagonal corners. Choose Rectangle from the Draw toolbar

AutoCAD prompts you for the two corners and creates the rectangle. You can specify the two corners using any method of specifying coordinates. For example, if you know the rectangle should be 6 inches wide and 3 inches high, you can specify the second point as @6,3.

*Once you specify the first corner, you can now specify the length and the width of the rectangle instead of the second corner.*

Here are the prompts for specifying the length and width of a rectangle:

Specify other corner point or [Dimensions]: *Choose the Dimension option.*

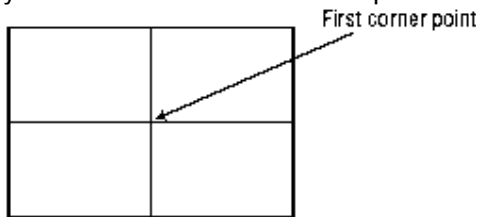
Specify length for rectangles <0.0000>: *Type the length and press Enter.*

Specify width for rectangles <0.0000>: *Type the width and press Enter.*

Specify other corner point or [Dimensions]: *Pick a point to specify where you want the rectangle.*

Once you have specified the length and width, four possible rectangles are possible, as shown in the figure. As you move your mouse cursor around the first corner you specified, AutoCAD displays these rectangles. Click when you see the one you want.

Once you set the dimensions, they remain as defaults for future rectangles that you draw. As a result, you can use the Dimensions option to quickly draw a number of identical rectangles.



When you specify the first corner, a length, and a width, choose which of four possible rectangles you want.

## Drawing Polygons



AutoCAD's POLYGON command enables you to draw multisided closed figures with equal side lengths. AutoCAD can draw polygons with anything from 3 to 1,024 sides. To draw a polygon, choose Polygon from the Draw toolbar.

### Creates an equilateral closed polygon

POLYGON command creates a regular polygon. AutoCAD constructs a polygon using the object type called *Polyline*. The concept of polyline is discussed later in this lesson.

Command: `_polygon` Enter number of sides <4>: *Enter a positive integer*

### Center, Inscribed/Circumscribed

Specify center of polygon or [Edge]: *Use one of the point fixing methods*

Defines the center point of an imaginary circle, which passes through the edges or endpoints of the polygon.

Enter an option [Inscribed in circle/Circumscribed about circle] <1>: *I Specify radius of circle: Specify radius of the imaginary circle*

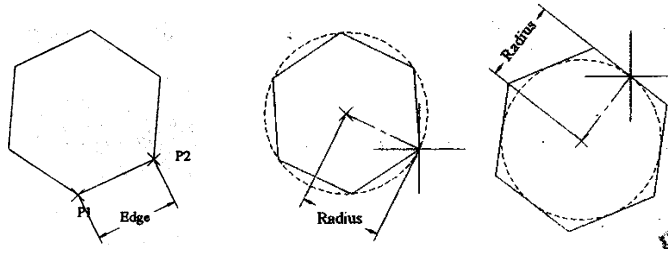
Using inscribed option AutoCAD draws the polygon, such that the vertices of the polygon lie on the circumference of an imaginary circle constructed by AutoCAD with radius and center point specified.

Enter an option [Inscribed in circle/Circumscribed about circle] <1>: *C Specify radius of circle: Specify radius of the imaginary circle*

Using circumscribed option AutoCAD constructs the polygon such that the side of base is tangential to all imaginary circle constructed by AutoCAD with radius and center point specified.

## Edge

Using the edge method to create the polygon, you define the side of base by fixing two points and AutoCAD constructs the rest of polygon, here the user has control over the exact size and orientation of the polygon.



Command: `_poly`

Command: `_polygon` Enter number of sides <6>: *Enter a positive integer* Specify center of polygon or [Edge]: *E*

Specify first endpoint of edge: *Use one of the point fixing methods* Specify second endpoint of edge: *Use one of the point fixing methods*

If you type a number for the radius, the bottom edge of the polygon is horizontal. However, if you pick a point for the radius with your mouse, you can specify the orientation of the polygon. Rotate the mouse cursor around the center, and you see the polygon rotate. Pick when you like what you see. If you type a number for the radius, the bottom edge of the polygon is horizontal. However, if you pick a point for the radius with your mouse, you can specify the orientation of the polygon. Rotate the mouse cursor around the center, and you see the polygon rotate. Pick when you like what you see.

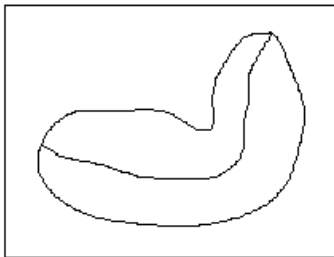
## Exploding Polylines

You can explode rectangles into their component parts. You may need to do this to edit a block. If you wish, you can then redefine the block. To explode a block, choose Explode from the Modify toolbar. (You can select objects before or after choosing the command.) You can also explode polylines, dimensions, hatches, regions, multilines, and certain 3D objects (bodies, 3D meshes, 3D solids, polyface meshes, and polygon meshes) into simpler types of objects. When you explode a block with nested blocks, AutoCAD only explodes the top level block. You need to use EXPLODE again to explode the next level of blocks.

## Creating and Editing Spline Curves

### Drawing and Editing Splines

The SPLINE command draws a NURBS, which stands for nonuniform rational B-spline. Not to get too technical, a spline is a smooth curve that is defined by a series of points. The SPLINE command provides a more precise representation of a spline than the Spline option of the PLINE command. By default, the curve passes through each point you define. Figure below shows a bean bag chair created with two splines. Both the chair and the grips are on all the originally specified points.



A bean bag chair created with two splines

### Creating splines

To create a spline, choose Spline from the Draw menu. AutoCAD responds with the Specify first point or [Object]: prompt. Use the Object option to convert a polyline that you have created with PEDIT's Spline option into a true spline. (It won't look any different, but its internal definition changes.) Otherwise, specify the first point for the spline.

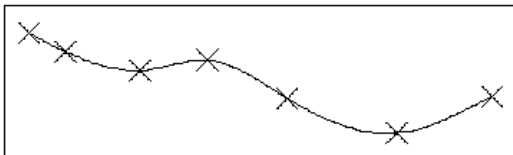
If you choose a point, AutoCAD displays the Specify next point: prompt so you can pick a second point. Once you do so, AutoCAD responds with the Specify next point or [Close/Fit tolerance] <start tangent>: prompt. Use these options as follows:

- **Close:** Closes the spline by connecting the last point with the first in a continuous (tangent) curve. AutoCAD asks for a tangent direction. You can specify a direction by picking a point (watch the spline image change as you move the cursor) or pressing Enter to accept the default tangent direction.
- **Fit Tolerance:** Specifies how closely the spline comes to the points you pick. The default, 0, creates a spline that passes through each point. If you want the curve to have a latitude of 0.5 units from the points, set the tolerance to 0.5.
- **Specify next point:** The default is to continue to enter points. Press Enter to end point selection.
- **Start tangent:** When you press Enter to complete point selection, AutoCAD prompts you for start and ending tangent directions. You can press Enter at both prompts to accept the default tangents based on the curve's current shape. You can see the effect of other tangent points by moving the cursor and watching the image change.

## Editing splines

Like polylines, splines have their own editing command. To understand how to edit splines, you need to understand how AutoCAD stores them. When you pick points to create the spline, AutoCAD stores these points as fit points (or data points). If the tolerance is zero, the curve lies on these fit points. When you select a spline at the command line, grips appear at these fit points. However, when you use the SPLINEDIT command, to see the fit points as grips, you must choose the Fit Data option first. Only then can you edit the fit points.

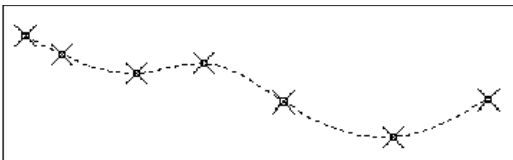
AutoCAD calculates control points based on the fit points. The spline is then calculated based on the control points, not the fit points. Most of the control points are not on the spline. When you use SPLINEDIT to edit a spline, and use the Move vertex option described later, AutoCAD displays the control points as grips, and you can move the control points. The figure below shows a spline created using point objects as the fit points—these were the points picked when creating the spline. Note that the spline passes through each fit point.



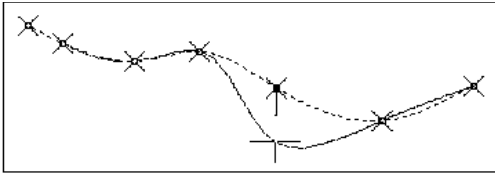
The point objects were used as the pick points.  
These are the fit points of the spline.

The next figure shows the same spline as in figure above as it appears when selected with no command active. Notice that the grips are exactly on the fit points. Because AutoCAD calculates the spline based on the control points, it doesn't need the fit points to generate the spline. In fact, if you use the Move vertex or Refine options to move or edit a control point, AutoCAD now has to use only the control points to generate the spline and discards the fit point information so that you can no longer edit it. The Fit Data option also disappears from the prompt.

You can edit the fit points using grips, as shown in below.



When you select a spline with no command active, you see grips on the fit points.

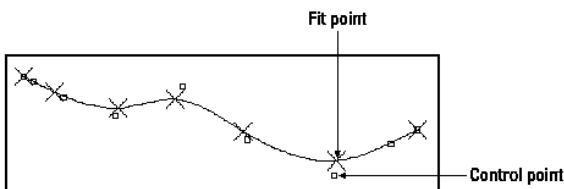


Editing fit points of a spline using grips

To edit a spline, choose Modify  $\rightarrow$  Object  $\rightarrow$  Spline to start the SPLINEDIT command. After you select the spline, AutoCAD responds with the Enter an option [Fit data/Close/Move vertex/Refine/rEverse/Undo]: prompt. Here's how to use these options:

- **Fit Data:** Fit data means the points you've chosen, their tolerance, and the tangents. Use this option to edit fit data. This option has its own list of suboptions that are explained after this list.
- **Close/Open:** If the spline is open, this option closes it by adding a continuous (tangent) curve from the last point to the start point. If the spline is closed, this option appears as Open. If the spline was originally closed, the Open option removes the connection between the last and first points, although the spline looks the same. If the spline was originally open and you closed it, when you use the Open option, AutoCAD erases the curve that it added when you closed it.
- **Move Vertex:** This works like the Edit vertex option of PEDIT, except that here AutoCAD displays the points as grips and highlights them. You can use Next and Previous suboptions, select any point to move, and pick a new location for the highlighted vertex.
- **Refine:** Enables you to refine the spline in three ways:
  - Add control points. This doesn't change the shape of the spline, but AutoCAD adjusts nearby control points slightly.
  - Elevate the order of the spline, which adds control points throughout the spline (but once you go up, you can't go down).
  - Change the weight of any control point. This is like the gravity the control point exerts on the spline. Watch the spline inch toward the control point as you increase its weight.
- **rEverse:** Reverses the direction of the spline so that the start point becomes the endpoint and vice versa.
- **Undo:** Undoes the most recent edit operation.

When you use the SPLINEDIT command and select a spline, you see control points, not fit points. In the figure below, the grips indicate the control points.



When you choose the SPLINEDIT command and select a spline, the grips indicate the control points, which are not generally on the spline.

If you choose the Fit Data option, AutoCAD responds with the Enter a fit data option [Add/Close/Delete/Move/Purge/Tangents/tolerance/eXit]

<eXit>: prompt. Descriptions of these options follow:

- **Add:** Adds fit data points to the curve. AutoCAD asks you to select a point and then automatically selects the next point as well, shown with highlighted grips. AutoCAD then prompts you for a new point, which must be between the two highlighted points. AutoCAD reshapes the spline accordingly.
- **Open/Close:** Opens or closes the spline using the fit points.
- **Delete:** Deletes a selected fit point.
- **Move:** Moves a fit point. You can use the Next or Previous suboptions or the Select point option to select the point you want to move. Selected points appear as highlighted grips. AutoCAD prompts you for the new fit point location. You can also use grips to edit fit points.
- **Purge:** Deletes fit point information.



- **Tangents:** Enables you to specify start and end tangents of open splines, or one tangent for closed splines. You can let AutoCAD calculate a default tangent.
- **toLerance:** Enables you to specify the tolerance, which determines how closely the spline comes to the fit points.
- **eXit:** Exits the suboption menu.

You can also use SPLINEDIT to edit 3D splines.

When you use SPLINEDIT and choose the Fit Data option, the grips appear on the fit points so you can edit them, as shown in the next figure.



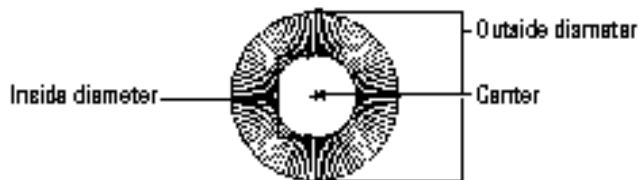
When you use the Fit Data option of the SPLINEDIT command, the grips appear on the fit points.

You can edit a spline in the Properties window. It works similarly to editing a polyline. You can choose the fits points or the control points and change them.

### Making Donuts

A donut is a wide polyline that looks like two concentric circles. You define a diameter for the inner and outer circles. AutoCAD accepts both DONUT and DOUGHNUT if you type in the command. Donuts are often used in electrical drawings and to create symbols. If the inner circle's radius is zero, you create a filled-in circle.

The setting of the FILL command determines whether AutoCAD fills in the donut. Type **fill** and type **on** or **off**. FILL is on by default. Turning FILL off displays a radial pattern of lines, as shown below. Type **regen** or choose View  Regen to update existing donuts to the new FILL setting.



### Understanding DONUT options

The DONUT command has the following prompts:

- Specify inside diameter of donut <0.5000>: Type the diameter of the inside circle or pick two points to specify the inside diameter. The number in brackets is the diameter of the last inside circle you defined.
- Specify outside diameter of donut <1.0000>: Type the diameter of the outside circle or pick two points to specify the outside diameter. The outside diameter must be larger than the inside diameter. The number in brackets is the diameter of the last outside circle you defined.
- Specify second point: If you define the inside or outside diameter by picking a point, AutoCAD asks for a second point. Use this technique if you are using object snaps to define the diameters.
- Specify center of donut or <exit>: Specify the center of the donut. Pressing Enter ends the command.

### Drawing donuts

Drawing donuts is relatively easy in AutoCAD. Choose Draw  Donut. Then specify the inner and outer diameters and the center. AutoCAD continues to prompt for centers so you can place additional donuts. Press Enter to end the command.

## Drawing ellipses

To draw an ellipse, choose Ellipse from the Draw toolbar. In addition to the information AutoCAD explicitly requests in the prompts, you need to know the angle of the first axis you define. Not all ellipses are horizontal or vertical. You control this when you stipulate the second point of the first axis. The second axis is automatically perpendicular to the first axis. The Ellipse Arc button is new on the Draw toolbar. It simply executes the ELLIPSE command with the Arc option.

To draw an elliptical arc, choose Ellipse Arc from the Draw toolbar. When you draw an elliptical arc, AutoCAD introduces a helpful but sometimes confusing feature: While you are defining the arc angles, AutoCAD redefines 0 degrees along the major axis. This helps you define the included angle in an orientation that relates to the ellipse, rather than the usual orientation where 0 degrees is to the right.

## Draw » Ellipse

### Creates an ellipse or an elliptical arc

AutoCAD helps drawing a perfect ellipse or elliptical curve using this command. There are two different ways in defining an ellipse: (a) you define the full length of one axis and half-length of the other axis, (b) define the center point of the ellipse (the point where the axes intersect with each other) and half-length of other two axes.

Command: `_ellipse`

Specify axis endpoint of ellipse or [Arc/Center]: Use *one of the point fixing methods or enter an option*

### Axis Endpoints

#### Draw > Ellipse > Axis, End

This is the first step towards defining the first axis. The two end points of the axis are specified. When the first point is specified in the above option, it prompts for the second point.

Specify other endpoint of axis: Use *one of the point fixing methods*

Here, there is not distinction of major or minor axes of an ellipse. Depending on the distance you specify one could be the major axis of an ellipse and the other could be the minor axis. AutoCAD doesn't make any difference between these two.

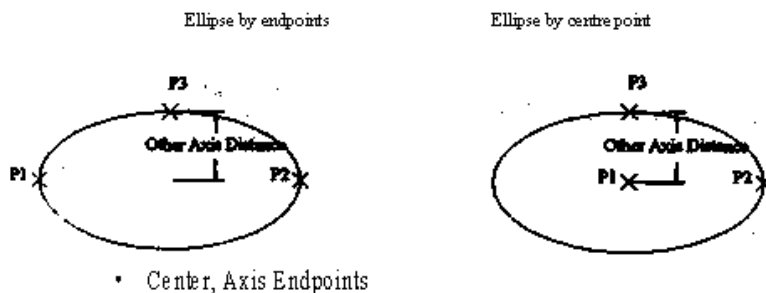
After the first axis of the ellipse is defined, AutoCAD gives you two options to define the length of the other axis. The first one is to define half-length of the other axis

Specify distance to other axis -or- [rotation]: Specify *a distance*

This defines the second axis as the distance from the center of the ellipse (midpoint of the first axis that you defined earlier) in a direction perpendicular to the first axis.

The *Rotation* option defines the major to minor axis ratio of the ellipse by rotating an imaginary circle about the first axis you defined by the given rotation angle. The higher the value, the greater is the ratio of minor to major axis. Entering 0 defines a circle. Rotation around major axis: Specify a point or enter a value (0-89.4)

Specify distance to other axis or [Rotation]: *R* Specify rotation around major axis: Specify *an angle*



#### Draw > Ellipse > Center

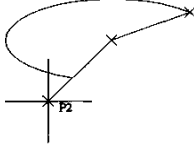
You can start from the center point of an ellipse. After the definition of the center point, AutoCAD proceeds in a similar way we learned above.

Command: `_ellipse`  
 Specify axis endpoint of ellipse or [Arc/Center]: C  
 Specify center of ellipse: *Specify a distance*  
 Specify endpoint of axis: *Specify a distance*  
 Specify distance to other axis or [Rotation]: *Specify a distance or angle*

## Draw > Ellipse > Arc

Creates an elliptical arc. AutoCAD constructs an imaginary ellipse using the methods we discussed above. Then the parameters about the arc are prompted for user input. The following sequence shows how a simple elliptical arc could be constructed.

PI



Elliptical arc by start and end angles

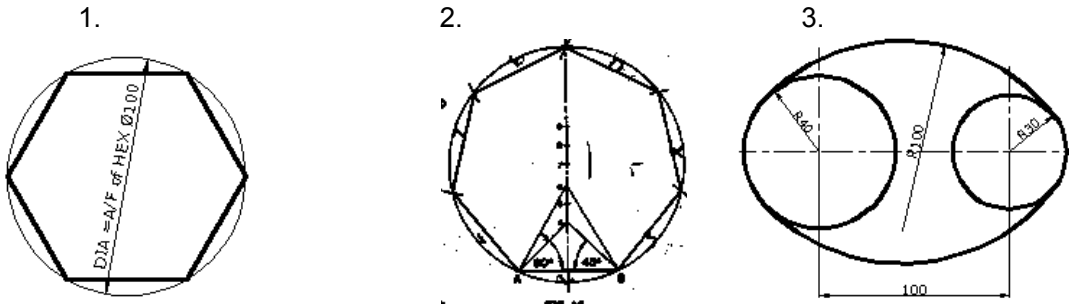
Command: `_ellipse`  
 Specify axis endpoint of ellipse or [Arc/Center]: a  
 Specify axis endpoint of elliptical arc or Center: *Use one of the point fixing methods*  
 Specify other endpoint of axis: *Use one of point fixing methods*  
 Specify distance to other axis or [Rotation]: *Specify a distance*  
 Specify start angle or [Parameter]: *Specify an angle or enter an option*  
 Specify end angle or [Parameter/Included angle]: *Specify an angle or enter an option*

To specify start angle, you could enter a numeric value, in which case, AutoCAD measures the angle from the angle of the first axis you defined.

Parameter requires the same input as start angle.

Specify start angle or [Parameter]: p  
 Specify start parameter or [Angle]: *Enter a value or specify an angle*  
 Specify end parameter or [Angle/Included angle]: *Enter a value or specify an angle*  
 Specify end parameter or [Angle/Included angle]: *Enter a value or specify an angle*

## Exercises



DIA. = A/C OF POLIGONS = 100

## Lesson 9

### Hatching Drawings

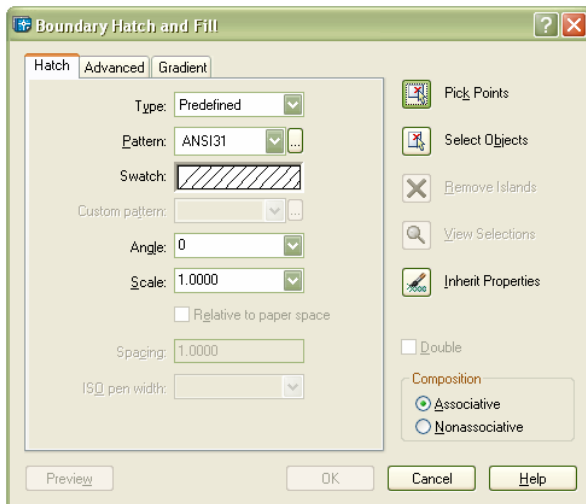
#### HATCHING

Hatching in AutoCAD is a way of filling in areas of your drawing with a pre-formatted pattern to represent certain materials. It is usually used in sectional views. Starting with AutoCAD release 14, you can now use a solid fill to completely fill in areas such as walls in a floor plan.

There are two types of hatching you can use. Generally you will want use the BOUNDARY HATCH command. Start by typing in **H**.

Command	Keystroke	Icon	Menu	Result
Boundary Hatch	<b>Bhatch / H</b>		Draw > Hatch	Covers an area with a predefined pattern.

When you start the command, you will this dialog box appear:



There are few things that you have to do to get the hatch that you require. First you must select the pattern that you want to use. Pick the **PATTERN** button to do this. Next you have to pick the internal point that you need the hatch to fill. Use the **PICK POINTS** button to do this. You can pick more than one internal point. The important thing to remember is that **the entire boundary must be visible on the screen**, or AutoCAD will not be able to hatch it. Once you have picked your pattern and points, pick on the **PREVIEW HATCH** button to see how the hatch will appear. If everything is OK, press the **APPLY** button. IF you need to change anything, you can adjust the scale and angle of the hatch until it looks the way you want it to.

If you are using a solid hatch, make the hatch WHITE so that it will print out as a solid black, any other color will appear grey (with lines) when printed.

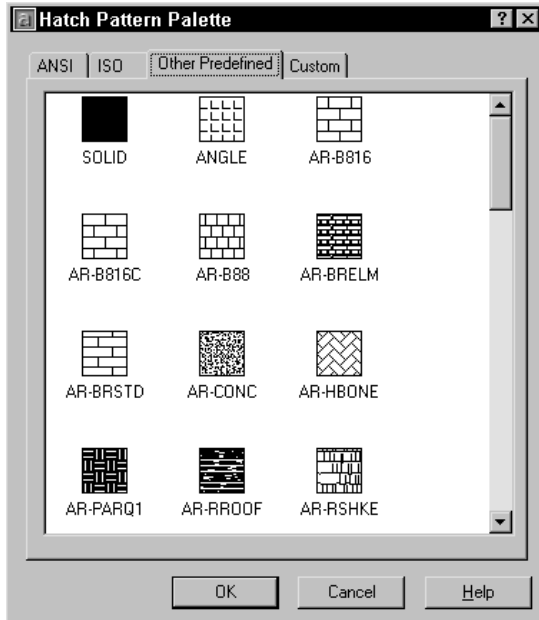
AutoCAD can detect 'islands' within a boundary area and not hatch them.

#### Defining a hatch

To hatch an area, choose Hatch from the Draw toolbar. This starts the BHATCH command. AutoCAD opens the Boundary Hatch dialog box, shown in below. Use this dialog box to define your hatch. From the Type drop-down box, choose one of the three options:

- **Predefined:** Enables you to select one of AutoCAD's hatch patterns.
  - **User-defined:** Enables you to define your own hatch pattern by specifying the angle and spacing, using the current linetype.
- Custom:** Enables you to choose a pattern you have created in your own .pat file.

Click the Pattern drop-down box to choose the hatch patterns from a list. Or click the box marked with an ellipsis to open the Hatch Pattern Palette, which enables you to choose from image tiles, as shown below:



In the palette, click the tabs to see the different types of hatches. Click the image tile to choose a hatch pattern and click OK. The palette is simply another method of choosing the hatch pattern—you can also choose the pattern using the Pattern drop-down list.

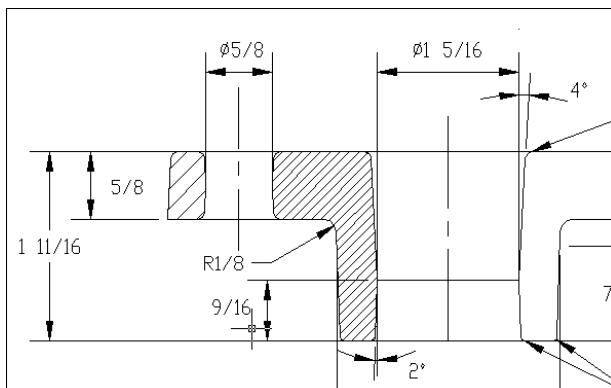
The swatch shows you how your chosen hatch will look.

The Custom pattern drop-down box is available only if you have chosen Custom as the pattern type. Here you choose the name of your custom hatch pattern.

Use the Angle text box to rotate the angle of the hatch pattern. You can choose from the drop-down list or type an angle. Watch out here—many of the patterns are already defined at an angle. The hatch pattern in the next figure uses a 0-degree angle because ANSI31 is defined as diagonal lines.

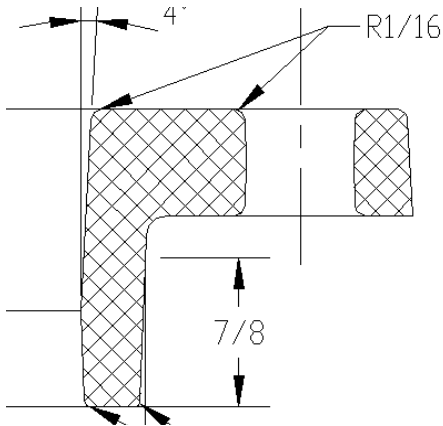
The Scale box determines the scale of the hatch pattern. You can choose from the drop-down list or type a scale. AutoCAD scales the hatch pattern proportionately to its definition. A scale of 1 (the default) creates the hatch as defined. A scale of 0.5 shrinks it by one-half. The second figure shows two hatch patterns using the ANSI31 pattern.

The left one uses a scale of 1, and the right one uses a scale of 0.5.



You can scale the hatch pattern to your needs.

The Spacing box and the Double checkbox are available if you choose a userdefined hatch pattern. A user-defined hatch uses the current linetype and creates a hatch based on the spacing and angle you specify. This figure shows a userdefined double hatch with an angle of 45 degrees and 0.1 unit spacing.



Use the following steps to define a user-defined hatch pattern:

1. Choose Hatch from the Draw toolbar to open the Boundary Hatch dialog box.
2. In the Type drop-down box, choose User-defined.
3. In the Angle box, type an angle, or choose one from the drop-down list. When creating a user-defined hatch, you need to specify the actual angle you want to see.
4. In the Spacing box, enter the spacing between the lines. If you type .5, AutoCAD creates a hatch pattern with lines 0.5 units apart.
5. If you want to cross-hatch so that the parallel lines are crossed by an equal number of perpendicular lines, check Double.

The ISO pen width box is available only for ISO predefined hatch patterns. AutoCAD adjusts the scale of the pattern according to the pen width you specify. Notice that when you choose a pen width from the drop-down box, the scale shown in the Scale box automatically changes to be equal to the pen width. Note that you still have to separately set the width of your plotter pens when you plot your drawing.

### Determining the hatch boundary

The hardest part of hatching is placing the hatch, not defining it. Hatching an entire object is the simplest way to place a hatch. But often the area you want to hatch is fairly complex, and AutoCAD needs to do some calculations to determine it.

The Boundary Hatch dialog box offers two ways to specify the hatch boundary—you can pick points inside an area and let AutoCAD try to find an enclosed boundary, or you can select objects. If you want to hatch an entire object, choose Select Objects. AutoCAD returns you temporarily to your drawing. Select all the objects you want to hatch. You can use all the standard object selection options to select objects. Remove and Add are especially helpful. When you are done, press Enter to end object selection and return to the dialog box.

If the area you want to hatch does not neatly fit into one or more objects, choose Pick Points. AutoCAD handles this task in the same way it handles the BOUNDARY command, also covered in this chapter. AutoCAD temporarily returns you to your drawing and displays the following:

Select internal point: Selecting everything...

Selecting everything visible...

Analyzing the selected data...

Select internal point:

AutoCAD is determining the boundary set, which is simply everything visible on the screen. At the Select internal point: prompt, pick a point that is inside the boundary you want to hatch. You can continue to pick internal points to hatch adjoining areas. AutoCAD helpfully highlights the boundaries it finds. Press Enter to return to the dialog box.

While you are in your drawing, either before or after picking points or selecting objects, you can right-click to open a very useful shortcut menu.

Choose from the following options on the shortcut menu:

- **Enter:** Returns you to the dialog box.

- **Undo Last Select/Pick:** Undoes your most recent object selection or point pick.
- **Clear All:** Undoes all your picks/object selections.
- **Pick Internal Point:** Switches to picking of internal points.
- **Select Objects:** Switches to selection of objects.
- **Normal Island Detection:** Sets island detection to normal mode.
- **Outer Island Detection:** Sets island detection to outer mode.
- **Ignore Island Detection:** Sets island detection to ignore mode.
- **Preview:** Previews the hatch.

This shortcut menu enables you to manage the hatch boundary without returning to the dialog box. It's always a good idea to preview the hatch before applying it. Click OK to create the hatch. AutoCAD creates the hatch and closes the dialog box.

The dialog box has several other options:

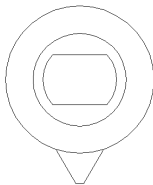
- Click Preview if you want to try it before you apply it.
- Click View Selection if you want to temporarily return to your drawing and check which objects you have selected.
- Choose Inherit Properties to use the hatch type, pattern, angle, scale, and/or spacing of an existing hatch. AutoCAD returns you to your drawing, and you pick a hatch pattern. You then return to the Boundary Hatch dialog box.
- Click Nonassociative if you want to create a hatch that is not associated with its object.

### Islands

Islands are enclosed areas entirely inside a hatch boundary. Islands make hatching more difficult because sometimes you don't want to hatch the inside of the island.

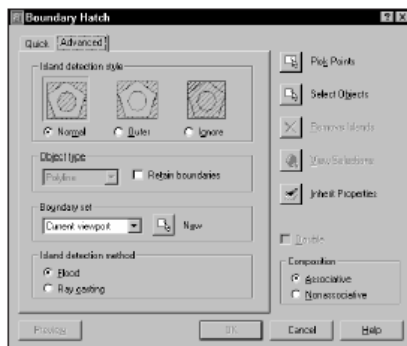
Below is a top view of a 3-inch operating nut, which is a good example of enclosed areas with islands.

Text is counted as an island, enabling you to hatch areas that contain text without hatching over the words.



### Managing islands when you select objects to hatch

When you choose the hatching boundary by selecting objects, you must also select the islands. If you can select the entire area by window, you automatically include the internal islands. If you need to pick individual objects, you must also pick the islands individually. If you later erase an island, AutoCAD retains hatch associativity and regenerates the hatch so it covers the entire outer boundary. The resulting hatch depends on the boundary style. To specify the boundary style, in the Boundary Hatch dialog box, click the Advanced tab, shown in the figure below.

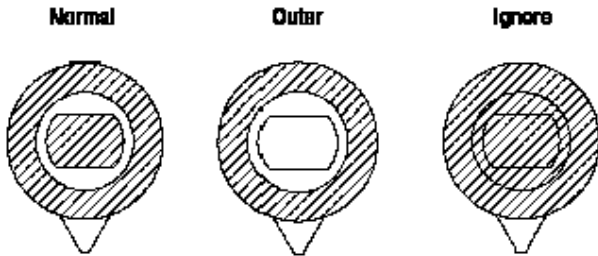


The Advanced tab is used to refine the hatching process.

Three boundary styles affect how islands are hatched:

- **Normal:** Hatches alternating areas so that the outer area is hatched, the next inner island is not hatched, the next inner island is hatched, and so on.
- **Outer:** Hatches only the outer area and does not hatch any inner islands.
- **Ignore:** Ignores islands and hatches everything from the outside in.

The figure below shows three copies of the nut hatched in the three styles. To hatch this model, I selected the entire model, except the spout at the bottom, with a window.



Hatching islands using the three boundary styles

### Managing islands when you pick points to specify the hatch

When you pick points instead of selecting objects, you do not need to select the islands. AutoCAD detects islands by default. Once you pick points, the Remove Islands button becomes available, and you can select the islands to remove them from consideration. For example, if you remove all the islands shown in the figure above, the result is the same as using the Ignore style—everything inside the outside boundary is hatched.

If you don't want AutoCAD to detect islands when you pick points, click Advanced.

In the Advanced Options dialog box, uncheck Island Detection.

### Other advanced options

On the Advanced tab you can also specify a smaller boundary set than the objects visible onscreen. Use this only when you are picking points and have such a complex drawing that AutoCAD takes a long time to analyze the visible objects.

When you pick points to determine the hatch boundary, AutoCAD uses the same mechanism as the BOUNDARY command to temporarily create a boundary for hatching. Check Retain Boundaries on the Advanced tab if you want to keep the boundary and specify if you want it to be a region or a polyline. Otherwise, AutoCAD discards the boundary once it completes the hatch.

You can choose whether you want to include islands in the boundaries that AutoCAD creates by choosing the Island detection method. The Flood method includes islands as boundary objects. The Ray casting method excludes islands as boundary objects.

### Dragging and dropping hatch patterns

You can open a *.pat* (hatch pattern) file from the AutoCAD DesignCenter, preview its hatch patterns, and drag any hatch pattern into any closed object in your current drawing.

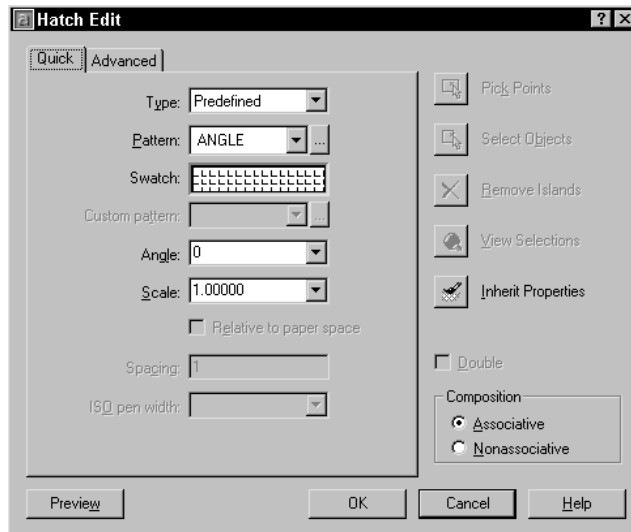
Here's how to drag a hatch pattern from the DesignCenter:

1. Click AutoCAD DesignCenter on the Standard toolbar (or press Ctrl+2) to open the DesignCenter.
2. Use the Tree View to navigate to the *\AutoCAD 2004\Support* folder. If necessary, click the Desktop button and navigate from there.
3. Click the plus (+) sign next to the *Support* folder and select a hatch pattern (*.pat*) file. The main hatch pattern file is *acad.pat*. (Of course, if the *.pat* file you want is in another location, navigate accordingly.)
4. On the right pane of the DesignCenter, drag the hatch pattern you want onto a closed object in your drawing. If you need more options, right-click the pattern and choose BHATCH to open the Boundary Hatch dialog box and specify the hatch parameters in the usual way.

### Editing hatches

To edit a hatch pattern, double-click it. AutoCAD opens the Hatch Edit dialog box, shown in the figure below. You can also choose Modify ⇄ Object ⇄ Hatch and then select a hatch object.





The Hatch Edit dialog box

As you can see, this dialog box is exactly the same as the Boundary Hatch dialog box, except that not all of the options are available. You can use this dialog box to change any of the hatch properties, however. You can change the boundary style on the Advanced tab. You can also preview the hatch. When you have made your changes, click Apply to return to your drawing.

Because hatches are associative (unless you explode them or chose to create them as nonassociative), when you edit their boundaries, they adjust to fit the new boundary. However, if the new boundary is no longer closed, the hatch may lose its associativity. AutoCAD warns you with the “Hatch boundary associativity removed” message.

To move or stretch a hatched boundary, you can select the hatch along with the boundary. However, perhaps you want to change the boundary’s layer. You may find it difficult to select the boundary without the hatch, especially if the hatch is tightly spaced. If you are at a Select objects: prompt, use a window to select the boundary and the hatch together, and then use Remove to pick the hatch. Another option is to turn off the hatch’s layer. Finally, you can zoom in enough so that you can pick the boundary without the hatch lines.

Note: When you create hatches—solid fill or lines—AutoCAD displays them if FILL is on, which it is by default. To turn FILL off, type **fill** ↵ and **off** ↵. You must regenerate the drawing to see the effect.

You may find it difficult to select solid fill hatches. In some locations, you can pick the solid hatch, while at other points you get the Other corner: prompt, meaning that AutoCAD didn’t find anything. Try a crossing window. If necessary, try a crossing window at the edge of the hatch. This always selects the solid hatch but also selects the boundary. Hatches have a grip at their center. If you can find the grip and include it in the window, you can easily select the solid hatch.

### Using the SOLID command

AutoCAD also has a command, SOLID, that creates solidly filled areas. (It is not at all related to 3D solids.) In general, BHATCH is much more flexible. However, although the SOLID command is a 2D command, it is sometimes used in 3D drawing. When you create a 2D solid and give it thickness, it creates surfaces with tops and bottoms.

SOLID is used to create straight-edged shapes. If FILL is on, AutoCAD fills in the shape with a solid fill (that’s why it’s called SOLID).

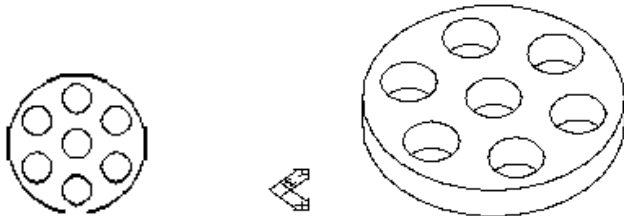
To draw a solid, type **solid** ↵. AutoCAD prompts you for first, second ↵, third, and fourth points. You must specify these points in zigzag order, not around the perimeter of the shape. After the fourth point, AutoCAD continues to prompt you for third and fourth points, which you can use to add to the solid. Press Enter to end the command.

### Creating Regions

Regions are two-dimensional surfaces. They look like closed polylines, but AutoCAD can calculate more information from regions than from polylines, such as the centroid, moments of inertia, and other

properties relating to mass. You can also create complex shapes by combining, subtracting, and intersecting regions. While these commands are most often used for 3D drawing, they are often used in 2D drawing as a preparation for 3D drawing. You create a region from other objects. You can use closed polylines, closed splines, circles, ellipses, and combinations of lines, arcs, and elliptical arcs that create a closed shape. The shape cannot intersect itself like a figure 8.

A complex region is shown below. Although it looks like a circle with seven circles inside it, it is actually a circular surface with seven holes in it. When you select it, you can see that it is one object. The real proof of the pudding is when you try to extrude it to create a 3D object out of it. You can then view it at an angle, hide background lines, and clearly see the holes, as shown in the second figure.



To create a region, choose Region from the Draw toolbar. AutoCAD asks you to select objects. Select all the objects and press Enter to end object selection. If all the objects create a closed, nonintersecting shape, AutoCAD tells you:

1 loop extracted.  
 1 Region created.

The original objects are deleted. If your objects aren't perfectly end to end, AutoCAD merely states:  
 0 loops extracted.  
 0 Regions created.

In order to draw your original objects end to end, remember to use Endpoint object snaps. Also, don't forget that you can start a line or arc at the end of the last point drawn by pressing Enter at the first prompt. You can also use TRIM and EXTEND to clean up extraneous endpoints. The BOUNDARY command (see the next section) offers a way to create regions in situations where objects are not neatly drawn end to end.

If you had a hatch inside the objects, you lose hatch associativity. You can re Hatch the region if you wish. When you create a region that is hatched, AutoCAD has no problem creating the region but doesn't quite know what to say about the hatch.

Here's the response:


1 closed, degenerate or unsupported object rejected.

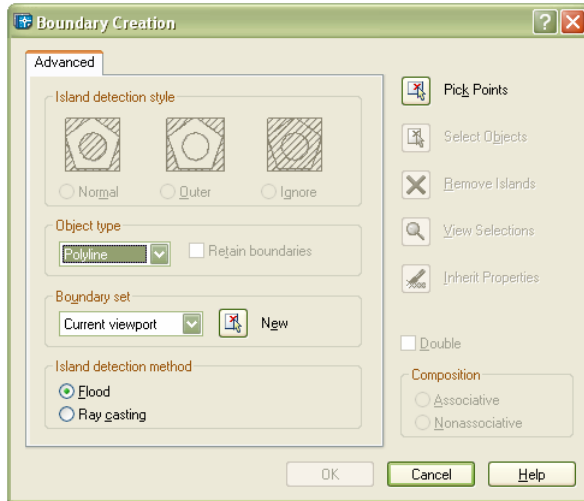
As mentioned earlier, you can combine, subtract, and intersect regions to create complex objects. The three commands to accomplish these functions are UNION, SUBTRACT, and INTERSECT. These commands are most often used in 3D drafting.

### Creating Boundaries

The BOUNDARY command creates either polylines or regions from an enclosed area. This command has the capability of analyzing an area and ignoring intersecting lines that give the REGION command so much trouble. However, no spaces between objects are allowed. You can use the BOUNDARY command to calculate the area of a closed space.

Use the BOUNDARY command whenever you need to create a closed complex area.

To create a boundary, choose Draw  Boundary to open the Boundary Creation dialog box. This dialog box is related to the Boundary Hatch dialog box. Most of the items in the dialog box are not active unless you are creating a hatch.



From the Object Type drop-down list, choose either Region or Polyline. Next, choose the boundary set, which is the area AutoCAD analyzes to create the boundary.

Usually, you can accept the default of Current Viewport. However, if you have a very complex drawing, choose New to temporarily return to your drawing. Specify a window around the area you want for the boundary set. AutoCAD returns you to the dialog box. Now choose Pick Points. AutoCAD returns you to your drawing with the Select internal point: prompt. Pick any point inside the closed area you want for your boundary. AutoCAD starts to think as follows:

Selecting everything...Selecting everything visible...Analyzing the selected data...Analyzing internal islands...

AutoCAD then prompts you for another internal point. If you want to create other boundaries, continue to pick internal points. Press Enter to end point selection.

AutoCAD informs you how many regions or polylines it created and ends the command.

When BOUNDARY creates a region or polyline, the original objects are not deleted.

You end up with a region or polyline on top of your original objects.

### Creating Construction Lines

Sometimes you want to create a line that is used solely for the purpose of reference.

For example, you might want to do the following:

- Draw two lines from the midpoints of two perpendicular lines so that you can use their intersection as the center for a circle.
- Draw a line from one object to another to visually indicate the relationship between the two objects.
- Show the relationship between equivalent parts of a model shown in front and right-side views.
- Draw a line through the center of an object shown in cross-section so that you can show dimensions from the center line to the edge of an object.

You could use regular lines for these purposes. However, construction lines (also known as *xlines*) are unique in that they extend infinitely in both directions. This makes them especially useful for seeing the relationships among various objects in your drawing.

Of course, construction lines are not actually infinite. However, they extend to the edge of the drawing area on your screen, and if you zoom out to see more of your drawing, they expand so that they always extend to the edge of the screen. AutoCAD's object snap tracking sometimes eliminates the need for construction lines; nevertheless, sometimes you can work more easily having a line visible for several commands and then erasing it.

If you zoom to show the extents of your drawing, AutoCAD ignores the xlines and shows you just the extents of the regular objects in your drawing.



The XLINE command offers several ways to create construction lines. Start the command by choosing Construction Line from the Draw toolbar. You see the following prompt: Specify a point or [Hor/Ver/Ang/Bisect/Offset]:

Below are lists the possible options. AutoCAD continues to prompt you for more points so that you can continue to draw construction lines—much like the LINE command. Press Enter to end the command.

### **XLINE Command Options**

<b>Option</b>	<b>Description</b>
Specify a point	This option enables you to define the xline with two points. At the first prompt, specify a point. At the Specify through point: prompt, specify another point. The first point becomes the base point for subsequent construction lines that you can draw by specifying other through points.
Hor	To draw a construction line parallel to the X axis, type <b>h</b> to specify the Horizontal option. AutoCAD responds with the Specify through point: prompt. Specify one point. Useful for drawing a series of horizontal construction lines.
Ver	To draw a construction line parallel to the Y axis, type <b>v</b> to specify the Vertical option. AutoCAD responds with the Specify through point: prompt. Specify one point.
Ang	Type <b>a</b> (for Angle). AutoCAD responds with the Enter angle of xline (0) or [Reference]: prompt. If you enter an angle, AutoCAD asks for a through point.  Or you can type <b>r</b> and select a line as a reference, and then provide an angle and a through point. AutoCAD then calculates the angle of the construction line from the angle of the reference line. Useful for drawing a series of construction lines at a specified angle.
Bisect	To draw a construction line that bisects (divides in half) an angle, type <b>b</b> . AutoCAD responds with the Specify angle vertex point: prompt. Choose any point that you want the construction line to pass through. Then, at the Specify angle start point: prompt, choose a point that defines the base of the angle. At the Specify angle end point: prompt, choose a point that defines the end of the angle.
Offset	To draw a construction line parallel to a line, type <b>o</b> . You can specify the offset distance by typing in a number or use the Through option to pick a point through which the construction line should pass. Either way, the next step is to select a line. If you specified an offset distance, AutoCAD displays the Specify side to offset: prompt. Respond by picking a point on the side of the selected line on which you want the construction line to appear.

### **Creating Rays**

Rays are similar to construction lines, except that they start at a specific point and extend to infinity in one direction only. If you need a line to extend only in one direction, using a ray may be less confusing.

You can use most object snaps with construction lines and rays. (You can't use endpoint for construction lines or midpoint for rays.) Construction lines and rays can be edited like any other object.

To draw a ray, choose Draw - Ray. At the Specify start point: prompt, specify the start point for the ray. At the Specify through point: prompt, specify another point. AutoCAD continues to ask for through points. Press Enter to end the command.

## Lesson 10

### Isometric Drawings


An *isometric* drawing is a 2D drawing drawn to look like a 3D drawing. Every child learns how to draw a box that looks three-dimensional. By drawing parallelograms instead of squares, the drawing gives the impression of being in three dimensions. AutoCAD enables you to do the same thing.

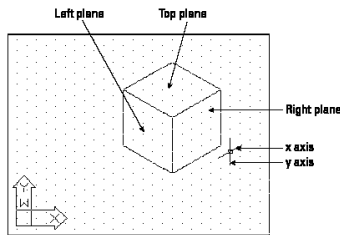
### Understanding isometric planes

AutoCAD uses the ISOPLANE (short for isometric plane) command to rotate the crosshairs to the special angles required for isometric drawing. You then toggle the ISOPLANE setting from left to right to top to draw on each of the three “planes.” As you do so, AutoCAD changes the angles of the crosshairs, snap, and grid to the appropriate angles.

These angles are 30 degrees for the X axis, 90 degrees for the Z axis, and 150 degrees for the Y axis. As you toggle among the planes, you see the crosshairs take on various configurations of these angles. The figure below shows the standard isometric cube. You can see three sides—left, right, and top. In the figure, the crosshairs are set to the right isometric plane.


Isometric drawing is not often used for precise drawing because it can be difficult to specify the exact points you need. Also, true 3D drawing has mostly supplanted isometric drawing. It is, however, used for piping work as well as for illustrations.

Note: Use snap points and object snaps as much as possible in an isometric drawing. Also, set the cursor to 100 percent of the screen to better visualize the isometric planes. (Choose Tools  Options and click the Display tab.) The grid is also a helpful aid.

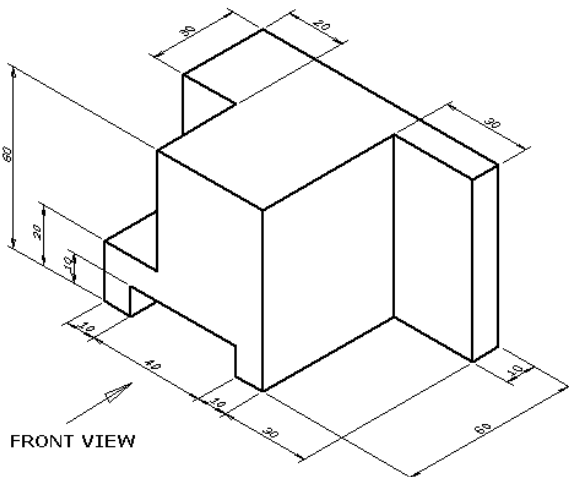


The isometric cube

### Drawing in isometric mode

To start isometric mode, choose Tools  Drafting Settings to open the Drafting Settings dialog box. On the Snap and Grid tab, in the Snap type & style section, choose Grid snap (if not already selected), and then Isometric snap. While you're there, turn on Snap and Grid if you want them on. Click OK. Once you are in isometric mode, press F5 to toggle from plane to plane.

#### 1. MACHINE BLOCK

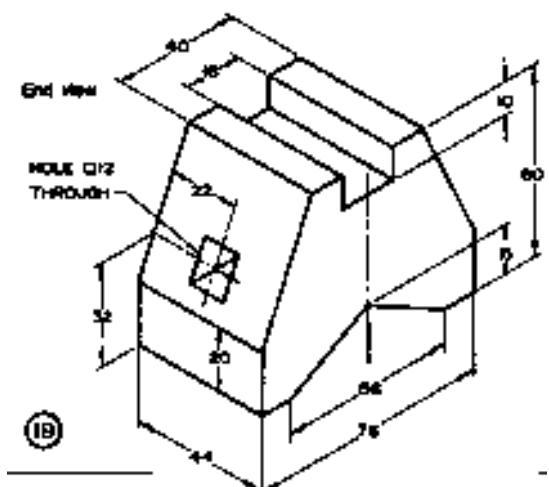
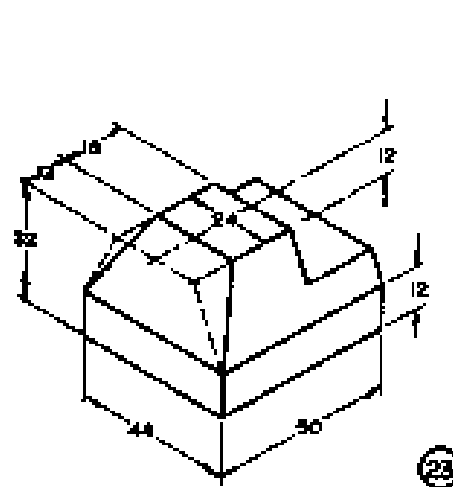
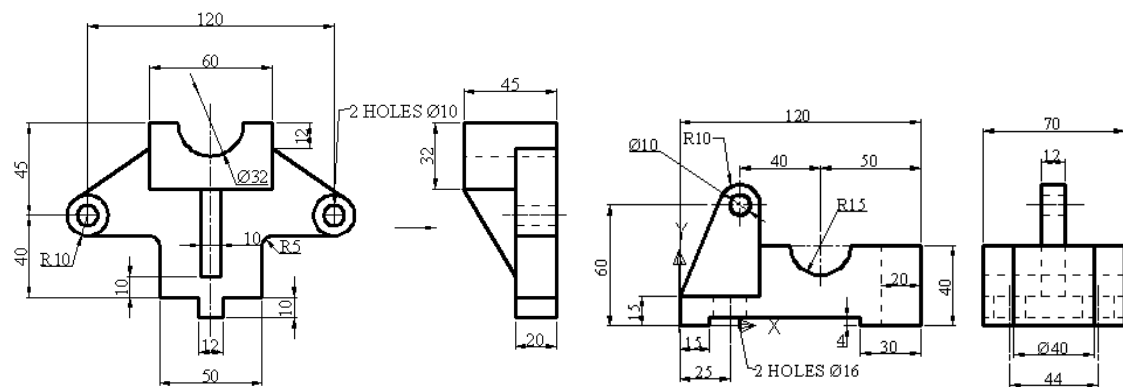
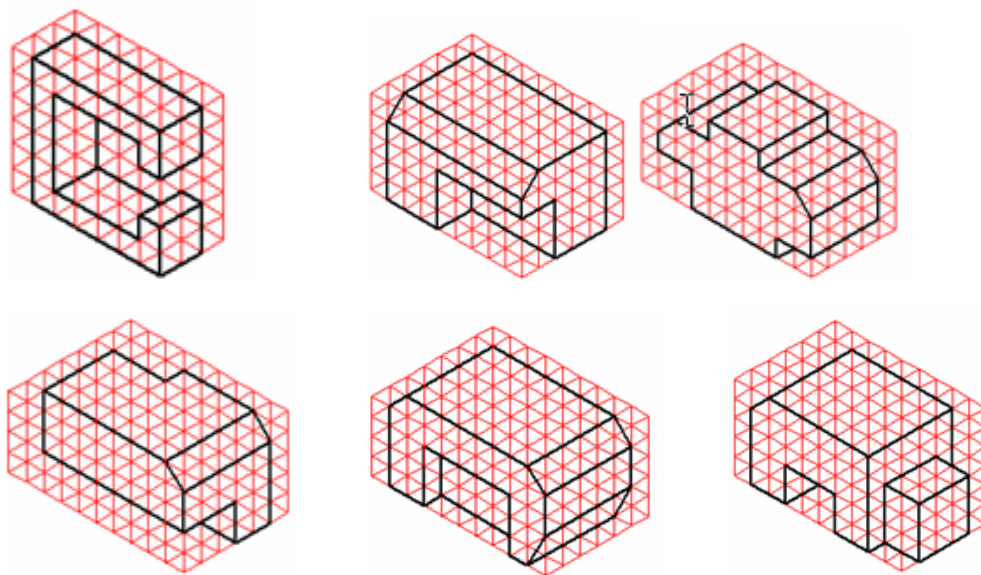


Drawing lines in isometric mode is fairly straightforward if the lines are parallel to one of the isometric plane angles. Circles and arcs in isometric mode must be drawn as ellipses and elliptical arcs. When you are in isometric mode the ELLIPSE command has an Isocircle option.

**Assignments:**

Set up a 5 x 5 grid and produce the isometric drawing of the COMPOSITE BLOCK drawn below. No dimensions are required and use a scale of 1: 1 (full size).

**Assignments:**



### Keyboard Shortcut Menus

Command aliases are shortcuts or alternative names for commands that you enter at the keyboard. This appendix lists the command aliases provided in the standard acad.pgp file. You can change or delete these aliases, or you can add some of your own by editing the acad.pgp file.

The acad.pgp file also defines aliases for external commands for Windows® and DOS, which are not listed here.

### Command Aliases

Commands	Aliases
3DARRAY	3a
3DFACE	3f
3DORBIT	3do
3DPOLY	3p
ADCENTER	adc
ALIGN	al
APPLOAD	ap
ARC	a
AREA	aa
ARRAY	ar
ATTDEF	att, dattdef
-ATTDEF	-att
ATTEDIT	ate
-ATTEDIT	-ate, atte
ATTEXT	ddatttext
BHATCH	h, bh
BLOCK	b
-BLOCK	-b
BOUNDARY	bo
-BOUNDARY	-bo
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