

Chapter 6

Editing Sketched Objects-II

Learning Objectives

After completing this chapter you will be able to:

- Understand the concept of grips and adjust grip settings.
- Stretch, move, rotate, scale, and mirror objects with grips.
- Use the **MATCHPROP** command to match the properties of the selected object.
- Use the **PROPERTIES** palette for editing the objects.
- Use the **QSELECT** command to select the objects.
- Manage the contents using the **DESIGNCENTER**.
- Use the various Inquiry commands.

EDITING WITH GRIPS

Grips provide a convenient and quick means of editing objects. With grips you can stretch, move, rotate, scale, and mirror objects, change properties, and load the Web browser. Grips are small squares that are displayed on an object at its definition points when the object is selected. The number of grips depends on the selected object. For example, a line has three grip points, a polyline segment has two, and an arc has three. Similarly, a circle has five grip points and a dimension (vertical) has five. When you select the **Enable grips** and the **Noun/verb selection** check boxes in the **Selection** tab of the **Options** dialog box, a small square (aperture box) at the intersection of the crosshair is displayed (Figure 6-1). The grip location of some of the objects is shown in Figure 6-2.

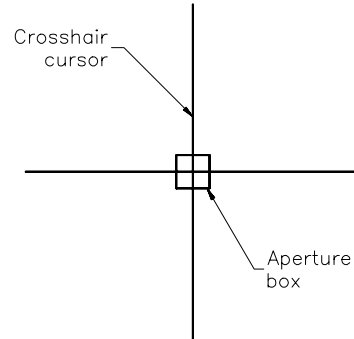


Figure 6-1 Aperture box at the intersection of crosshair

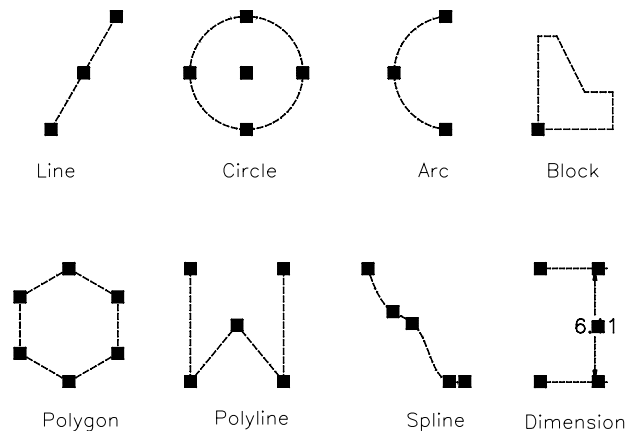


Figure 6-2 Grip location of various objects



Note

AutoCAD also displays a small square (aperture box) at the intersection of crosshairs when the **PICKFIRST** (Noun/Verb Selection) system variable is set to 1 (On).

TYPES OF GRIPS

Grips can be classified into three types: unselected grips, hover grips, and selected grips. Selected grips are also called hot grips. When you select an object, the grips are displayed at the definition points of the object, and the object is highlighted by displaying it as a dashed line. These grips are called unselected grips (blue). Now, if you move the cursor over the

unselected grip, and pause for a second, the grid is displayed in green color. These grips are called hover grips. Next, if you select a grip on this object, the grip becomes a hot grip (filled red square), Figure 6-3. Once the grip is hot, the object can be edited. To cancel the grip, press ESC. If you press ESC once, the hot grip changes to unselected grip. You can also snap to the unselected grip.

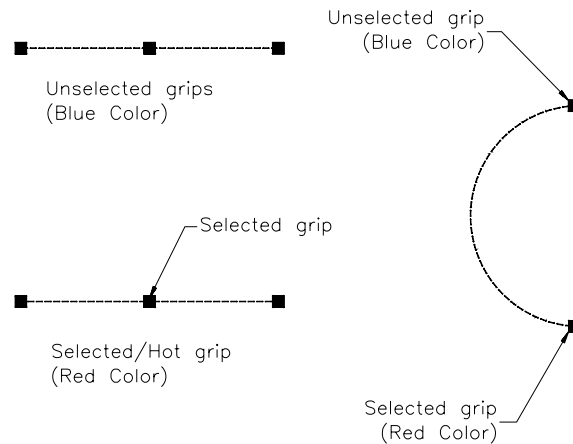


Figure 6-3 Figure showing the selected and unselected grips

ADJUSTING GRIP SETTINGS

Menu: Tools > Options
Command: OPTIONS

The grip settings can be adjusted using the options under the **Selection** tab of the **Options** dialog box. This dialog box can also be invoked by choosing **Options** from the shortcut menu, see Figure 6-4. The shortcut menu is displayed upon right-clicking in the drawing area.

The options related to grips that are provided under the **Selection** tab of the **Options** dialog box (Figure 6-5) are discussed next.

Grip Size Area

The **Grip Size** area of the **Selection** tab of the **Options** dialog box consists of a slider bar and a rectangular box that displays the size of the grip. To adjust the size of the grip, move the slider box left or right. The size of the grip can also be adjusted by using the **GRIPSIZE** system variable. The **GRIPSIZE** variable is defined in pixels, and its value can range from 1 to 255 pixels.

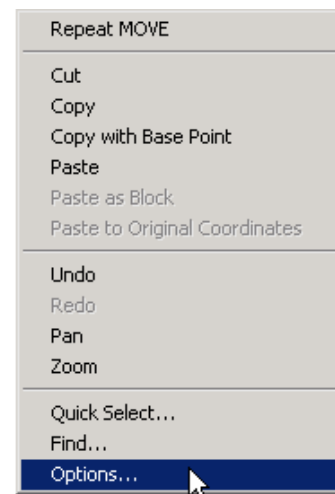


Figure 6-4 Invoking the **Options** dialog box from the shortcut menu

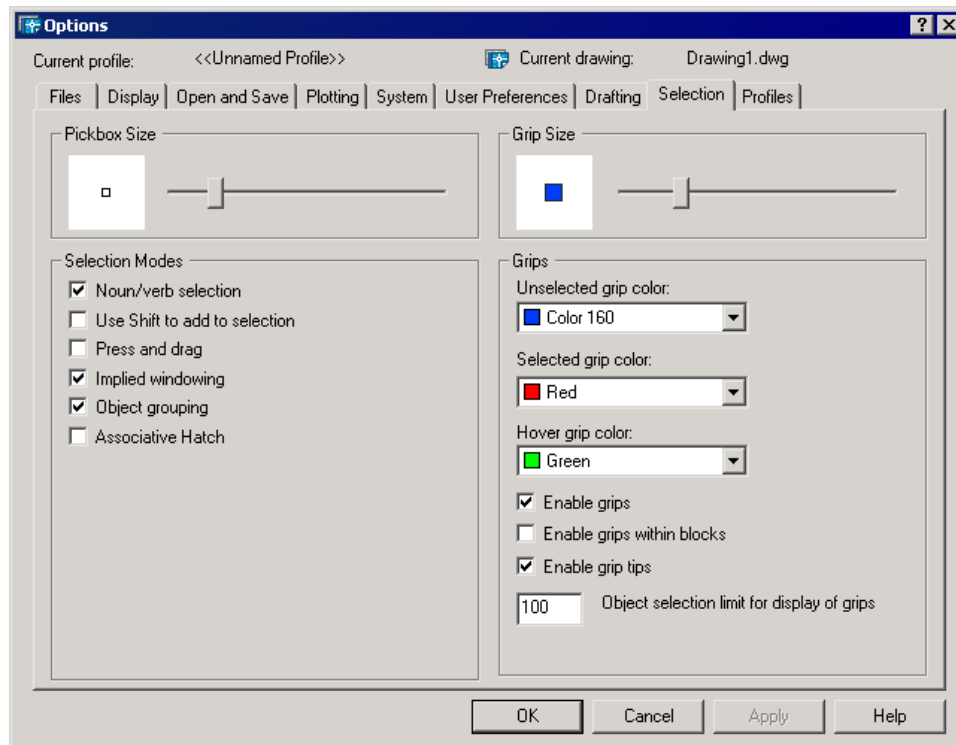


Figure 6-5 Selection tab of the **Options** dialog box

Grips Area

The Grips area is used to control the display and the color of the grips.

Unselected grip color

This drop-down list is used to set the color of the unselected grip. You can set the color by selecting it from this drop-down list or by selecting the **Select Color** option to display the **Select Color** dialog box. You can select the color for the unselected grip from this dialog box. This color can also be set using the **GRIPCOLOR** system variable.

Selected grip color

This drop-down list is used to set the color of the selected grip. You can set the color by selecting it from this drop-down list or by selecting the **Select Color** option to display the **Select Color** dialog box. You can select the color for the selected grip from this dialog box. This color can also be set using the **GRIPHOT** system variable.

Hover grip color*

This drop-down list is used to set the color of the hover grip. You can set the color by selecting it from this drop-down list or by selecting the **Select Color** option to display the **Select Color** dialog box. You can select the color for the hover grip from this dialog box. This color can

also be set using the **GRIPHOVER** system variable.

The Grips area has three check boxes; **Enable grips**, **Enable grips within blocks**, and **Enable grip tips**. The grips can be enabled by selecting the **Enable Grips** check box. They can also be enabled by setting the **GRIPS** system variable to 1. The second check box, **Enable grips within blocks**, enables the grips within a block. If you select this box, AutoCAD will display grips for every object in the block. If you disable the display of grips within a block, the block will have only one grip at its insertion point. You can also enable the grips within a block by setting the value of the **GRIPBLOCK** system variable to 1 (On). If **GRIPBLOCK** is set to 0 (Off), AutoCAD will display only one grip for a block at its insertion point (Figure 6-6). The third check box, **Enable grip tips**, enables you to display the grip tips when the cursor moves over the custom object that supports grip tips. If you disable this check box the grip tips are not displayed when the cursor moves over the custom object. You can also enable the grip tips by setting the value of the **GRIP TIPS** system variable to 0 (Off). If **GRIP TIPS** is set to 1 (On), AutoCAD will display the grip tips for the custom object.

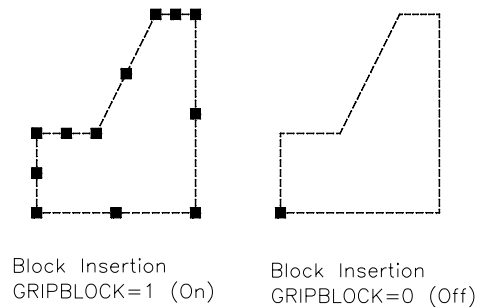


Figure 6-6 Block insertion with **GRIPBLOCK** set to 1 and to 0



Note

If the block has a large number of objects, and if **GRIPBLOCK** is set to 1 (On), AutoCAD will display grips for every object in the block. Therefore, it is recommended that you set the system variable **GRIPBLOCK** to 0 or clear the **Enable grips within blocks** check box in the **Selection** tab of the **Options** dialog box.

Object selection limit for display of grips*

This text box is used to specify the maximum number of objects that can be selected at a single attempt for the display of grips. If you select objects more than that specified in the text box using a single selection method, grips will not be displayed. Note that this limit is set only for those objects that are selected at a single attempt using any of the **Crossing window**, **Fence**, or the **All** options.

EDITING OBJECTS WITH GRIPS

As mentioned earlier, you can perform different kinds of editing operations using the selected grip. The editing operations are discussed next.

Stretching Objects With Grips (Stretch Mode)

If you select an object, AutoCAD displays unselected grips at the definition points of the object. When you select a grip for editing, you are automatically in the **Stretch** mode. The Stretch mode has a function similar to the **STRETCH** command. When you select a grip, it

acts as a base point and is called a base grip. You can also select several grips by holding the **SHIFT** key down and then selecting the grips. Now, release the **SHIFT** key and select one of the hot grips to stretch them simultaneously. The geometry between the selected base grips is not altered. You can also make copies of the selected objects or define a new base point. When selecting grips on text objects, blocks, midpoints of lines, centers of circles and ellipses, and point objects in the stretch mode, the selected objects are moved to a new location. The following example illustrates the use of the **Stretch** mode.

1. Use the **PLINE** command to draw a W-shaped figure as shown in Figure 6-7(a).
2. Select the object that you want to stretch [Figure 6-7(a)]. When you select the object, grips will be displayed at the endpoints of each object. A polyline has two grip points. If you use the **LINE** command to draw the object, AutoCAD will display three grips for each object.
3. Hold the **SHIFT** key down, and select the grips that you want to stretch [grips on the lower endpoints of the two vertical lines in Figure 6-7(b)]. The selected grips will become hot grips, and the color of the grip will change from blue to red.

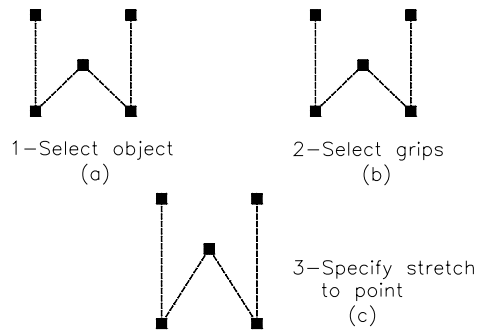


Figure 6-7 Using the **Stretch** mode to stretch the lines



Note

*You have to make sure that you hold the **SHIFT** key before selecting even the first grip. You can not hold the **SHIFT** key and select more grips if the first grip is selected without holding the **SHIFT** key.*

4. Select one of the selected (hot grip) grips, and specify a point to which you want to stretch the line [Figure 6-7(c)]. When you select a grip, the following prompt is displayed in the Command prompt area.

****STRETCH****

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

The Stretch mode has several options: **Base point**, **Copy**, **Undo**, and **eXit**. You can use the **Base point** option to define the base point and the **Copy** option to make copies.

5. Select the grip where the two lines intersect. Right-click to display the shortcut menu (Figure 6-8) and choose the **Copy** option. Select the points as shown in Figure 6-9(b). Each time you select a point, AutoCAD will make a copy.

If you press the SHIFT key when specifying the point to which the object is to be stretched, without selecting the copy option, then also AutoCAD allows you to make multiple copies of the selected object. Also, if you press the SHIFT key again when specifying the next point, the cursor snaps to a point whose location is based on the distance between the first two points, that is, the distance between the selected object and the location of the copy of the selected object.

6. Make a copy of the drawing as shown in Figure 6-9(c). Select the object, and then select the grip where the two lines intersect. When AutoCAD displays the ****STRETCH**** prompt, choose the **Base Point** option from the shortcut menu or enter B at the Command prompt. Select the bottom left grip as the base point, and then give the displacement point as shown in Figure 6-9(d).

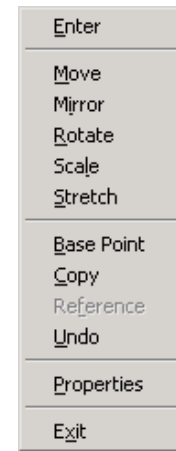


Figure 6-8 Selecting different Grip options from the shortcut menu

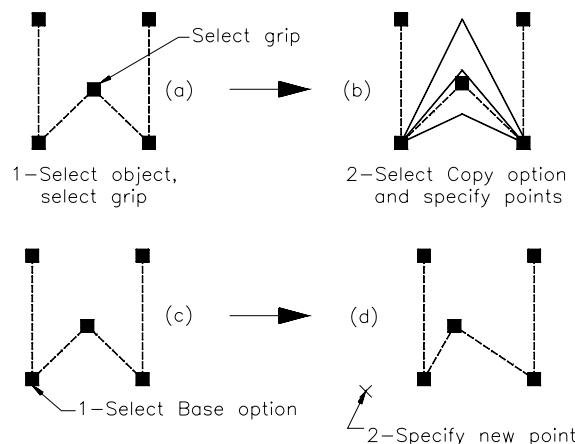


Figure 6-9 Using the Stretch mode's **Copy** and **Base point** options

7. To terminate the grip editing mode, right-click when the grip is hot to display the shortcut menu and then select **eXit**. You can also enter X at the Command prompt or press ESC to exit.



Note

You can select an option (**Copy** or **Base Point**) from the shortcut menu that can be invoked by right-clicking your pointing device after selecting a grip. The different modes can also be selected from the shortcut menu. You can also cycle through all the different modes by selecting a grip and pressing the ENTER key or the SPACEBAR.

Moving Objects with Grips (Move Mode)

The Move mode lets you move the selected objects to a new location. When you move objects, the size of the objects and their angle do not change. You can also use this mode to make copies of the selected objects or to redefine the base point. The following example illustrates the use of the **Move** mode.

1. Use the **LINE** command to draw the shape as shown in Figure 6-10(a). When you select the objects, grips will be displayed at the definition points and the object will be highlighted.

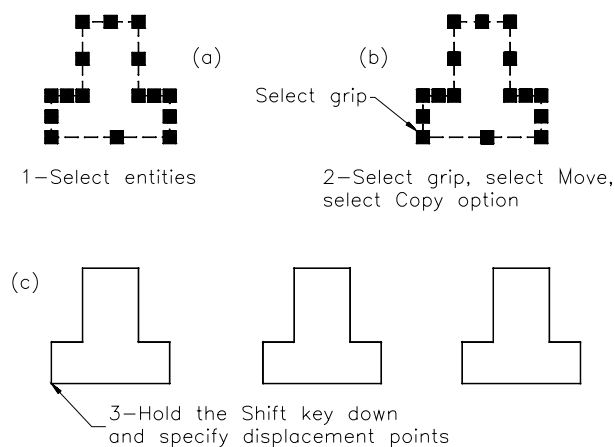


Figure 6-10 Using the **Move** mode to move and make copies of the selected objects

2. Select the grip located at the lower left corner, and then choose **Move** from the shortcut menu. You can also invoke the Move mode by entering **MOVE** or **MO** at the keyboard or giving a null response by pressing the SPACEBAR or ENTER key. AutoCAD will display the following prompt in the Command: prompt area.

****MOVE****

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

3. Hold down the SHIFT key, and then enter the first displacement point. The distance between the first and the second object defines the snap offset for subsequent copies. While holding down the SHIFT key, move the screen crosshair to the next snap point and select the point. AutoCAD will make a copy of the object at this location. If you release the SHIFT key, you can specify any point where you want to place a copy of the object. You can also enter coordinates to specify the displacement.

Rotating Objects with Grips (Rotate Mode)

The **Rotate** mode allows you to rotate objects around the base point without changing their

size. The options of Rotate mode can be used to redefine the base point, specify a reference angle, or make multiple copies that are rotated about the specified base point. You can access the Rotate mode by selecting the grip and then selecting **Rotate** from the shortcut menu, or by entering **ROTATE** or **RO** at the keyboard or giving a null response twice by pressing the SPACEBAR or the ENTER key. The following example illustrates the use of **Rotate** mode.

1. Use the **LINE** command to draw the shape as shown in Figure 6-11(a). When you select the objects, grips will be displayed at the definition points and the shape will be highlighted.
2. Select the grip located at the lower left corner and then invoke the Rotate mode. AutoCAD will display the following prompt:

****ROTATE****

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

3. At this prompt, enter the rotation angle. AutoCAD will rotate the selected objects by the specified angle [Figure 6-11(b)].
4. Make a copy of the original drawing as shown in Figure 6-11(c). Select the objects, and then select the grip located at the lower left corner of the object. Invoke the Rotate mode and then select the **Copy** option from the shortcut menu or enter **C** (Copy) at the Command prompt. Enter the rotation angle. AutoCAD will rotate the copy of the object through the specified angle [Figure 6-11(d)].

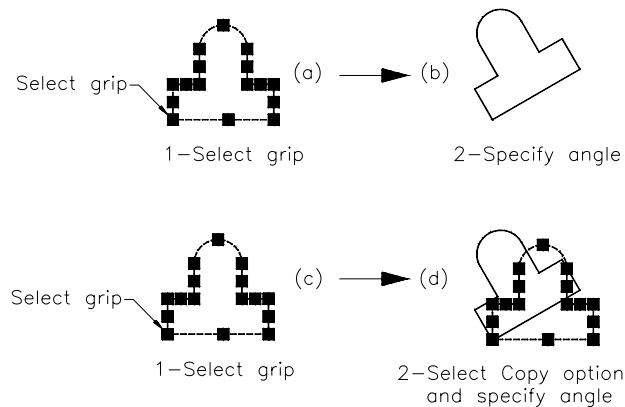


Figure 6-11 Using the **ROTATE** mode to rotate and make copies of the selected objects

5. Make another copy of the object as shown in Figure 6-12(a). Select the object, and then select the grip at point (P0). Access the Rotate mode and copy option as described earlier. Select the **Reference** option from the shortcut menu or enter **R** at the following prompt.

****ROTATE (multiple) ****

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

Specify reference angle <0>: *Select the grip at (P1).*

Specify second point: *Select the grip at (P2).*

Specify new angle or [Base point/Copy/Undo/Reference/eXit]: **45**

In response to the **Specify reference angle <0>**: prompt, select the grips at points (P1) and (P2) to define the reference angle. When you enter the new angle, AutoCAD will rotate and insert a copy at the specified angle [Figure 6-12(c)]. For example, if the new angle is 45-degree, the selected objects will be rotated about the base point (P0) so that the line P1P2 makes a 45-degree angle with respect to the positive X axis.

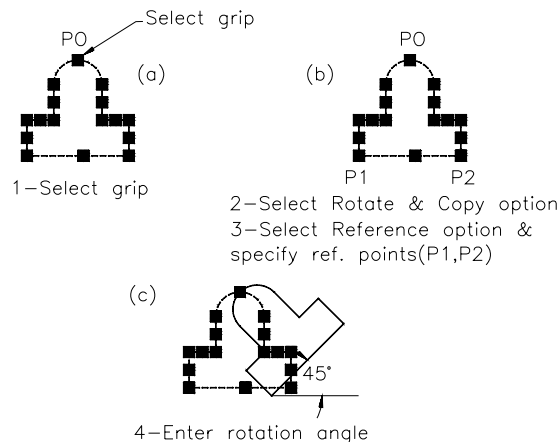


Figure 6-12 Using the **ROTATE** mode to rotate by giving a reference angle

Scaling Objects with Grips (Scale Mode)

The **Scale** mode allows you to scale objects with respect to the base point without changing their orientation. The options of **Scale** mode can be used to redefine the base point, specify a reference length, or make multiple copies that are scaled with respect to the specified base point. You can access the **Scale** mode by selecting the grip and then selecting Scale from the shortcut menu, or entering **SCALE** or **SC** on the keyboard, or giving a null response three times by pressing the SPACEBAR or the ENTER key. The following example illustrates the use of the **Scale** mode.

1. Use the **PLINE** command to draw the shape as shown in Figure 6-13(a). When you select the objects, grips will be displayed at the definition points, and the object will be highlighted.
2. Select the grip located at the lower left corner as the base grip, and then invoke the Scale mode. AutoCAD will display the following prompt in the Command prompt area.

****SCALE****

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

- At this prompt enter the scale factor or move the cursor and select a point to specify a new size. AutoCAD will scale the selected objects by the specified scale factor [Figure 6-13(b)]. If the scale factor is less than 1 (<1), the objects will be scaled down by the specified factor. If the scale factor is greater than 1 (>1), the objects will be scaled up.
- Make a copy of the original drawing as shown in Figure 6-13(c). Select the objects, and then select the grip located at the lower left corner of the object. Invoke the Scale mode. At the following prompt, enter C (Copy), and then enter B for base point.

****SCALE (multiple) ****

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

- At the **Specify base point** prompt, select the point (P0) as the new base point, and then enter **R** at the following prompt.

****SCALE (multiple) ****

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

Specify reference length $<1.000>$: *Select grips at (P1) and (P2).*

After specifying the reference length at the **Specify new length or [Base point/Copy/Reference/eXit]** prompt, enter the actual length of the line. AutoCAD will scale the objects so that the length of the bottom edge is equal to the specified value [Figure 6-13(c)].

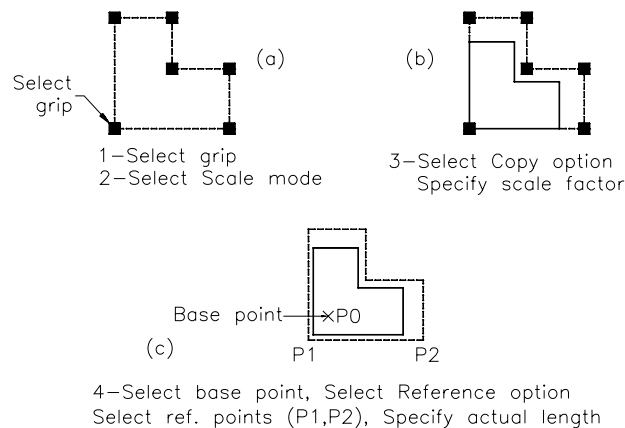


Figure 6-13 Using the **SCALE** mode to scale and make copies of selected objects

Mirroring Objects with Grips (Mirror Mode)

The **Mirror** mode allows you to mirror the objects across the mirror axis without changing the size of the objects. The mirror axis is defined by specifying two points. The first point is the base point, and the second point is the point that you select when AutoCAD prompts for the second point. The options of the Mirror mode can be used to redefine the base point and make a mirror copy of the objects. You can access the **Mirror** mode by selecting a grip and then choosing Mirror from the shortcut menu, or by entering **MIRROR** or **MI** at the keyboard, or giving a null response four times by pressing the SPACEBAR or the ENTER key. The following is the example for the **Mirror** mode.

1. Use the **PLINE** command to draw the shape as shown in Figure 6-14(a). When you select the object, grips will be displayed at the definition points and the object will be highlighted.
2. Select the grip located at the lower right corner (P1), and then invoke the Mirror mode. The following prompt is displayed.

****MIRROR****

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

3. At this prompt, enter the second point (P2). AutoCAD will mirror the selected objects with line P1P2 as the mirror axis as shown in Figure 6-14(b).
4. Make a copy of the original figure as shown in Figure 6-14(c). Select the object, and then select the grip located at the lower right corner (P1) of the object. Invoke the Mirror mode and then choose the **Copy** option to make a mirror image while retaining the original object. Alternatively, you can also hold down the SHIFT key and make several mirror copies by specifying the second point.

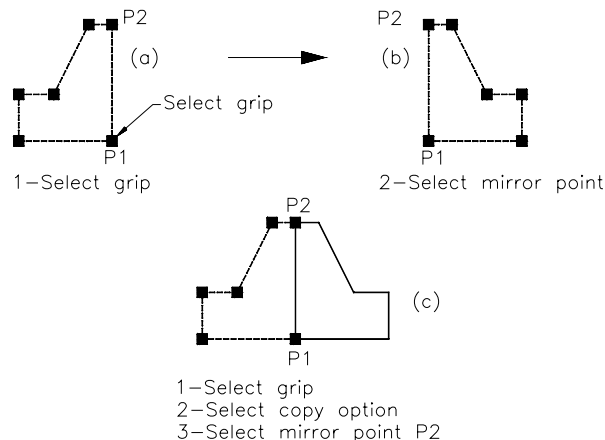


Figure 6-14 Using the **MIRROR** mode to create a mirror image of selected objects

5. Select point (P2) in response to the prompt **Specify second point or [Base point/Copy/Undo/eXit]**. AutoCAD will create a mirror image, and the original object will be retained.



Note

You can use some editing commands such as **ERASE**, **MOVE**, **ROTATE**, **SCALE**, **MIRROR**, and **COPY** on an object with unselected grips. However, this is possible only if the **PICKFIRST** system variable is set to 1 (On).

You cannot select an object when the grip is hot.

If you want to remove an object from the selection set displaying grips, press the **SHIFT** key and then select the particular object. This object, which is removed from the selection set, will now not be highlighted.

LOADING HYPERLINKS

If you have already added hyperlink to the object, you can also use the grips to open a file associated with the hyperlink. For example, the hyperlink could start a word processor, or activate the Web browser and load a Web page that is embedded in the selected object. If you want to launch the Web browser that provides hyperlinks to other Web pages, select the URL-embedded object and then right-click to display the shortcut menu. In the shortcut menu, select the **Hyperlink** option and AutoCAD will automatically load the Web browser. When you move the cursor over or near the object that contains a hyperlink, AutoCAD displays the hyperlink information with the cursor.

EDITING GRIPPED OBJECTS

You can also edit the properties of the gripped objects by using the **Properties** toolbar (Figure 6-15). The gripped objects are created when you select objects without invoking a command. The gripped objects are highlighted and will display grips (rectangular boxes) at their grip points. For example, if you want to change the color of the gripped objects, select the **Color** drop-down list in the **Properties** toolbar and then select a color. The color of the gripped objects will change to the selected color. Similarly, if you want to change the layer, lineweight, or linetype of the gripped objects, select the linetype, lineweight, or layer from the corresponding drop-down lists. If the gripped objects have different colors, linetypes, or lineweights, the Color Control, Linetype Control, and Lineweight Control boxes will appear blank. You can also change the Plot Style of the selected objects.

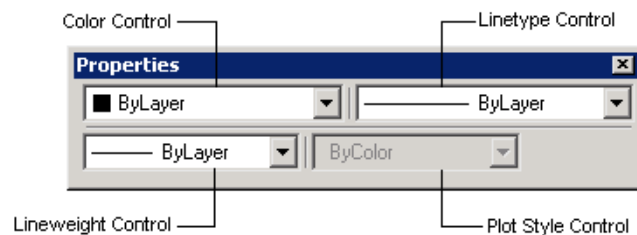


Figure 6-15 Using the **Properties** toolbar to change properties of the gripped objects

GRIP SYSTEM VARIABLES

System variable	Default	Setting	Function
GRIPS	1	1=On, 0=Off	Enables or disables Grip mode
GRIPBLOCK	0	1=On, 0=Off	Controls the display of grips in a block
GRIPCOLOR	160	1-255	Specifies the color of unselected grips
GRIPHOT	1	1-255	Specifies the color of selected grips
GRIPSIZE	5	1-255	Specifies the size of the grip box in pixels

CHANGING THE PROPERTIES USING THE PROPERTIES PALETTE

Toolbar:	Standard > Properties
Menu:	Modify > Properties
Command:	PROPERTIES



As mentioned earlier, each object has a number of properties associated to it such as the color, layer, linetype, line weight, and so on. You can modify the properties of an object by using the **PROPERTIES** command. When you invoke this command, AutoCAD will display the **PROPERTIES** palette, see Figure 6-16. The **PROPERTIES** palette can also be displayed when you double-click on the object to be edited. The contents of the **PROPERTIES** palette change according to the objects selected. For example, if you select text entity, the properties related to the text such as its height, justification, style, rotation angle, obliquing factor, and so on will be displayed.

The **PROPERTIES** palette can also be invoked from the shortcut menu displayed when you right-click in the drawing area. Choose the **Properties** option to display the **PROPERTIES** palette. If you select more than one object, the common properties of the selected objects will be displayed in the **PROPERTIES** palette. To change properties of the selected objects, you can click in the cell next to the name of the property and change the values manually or you can choose from the available options in the drop-down list, if one is available. You can cycle through the options by double-clicking in the property cell.



Note

*Some of the options of the **PROPERTIES** palette have been explained in Chapter 4. Other options of the **PROPERTIES** palette will be explained in detail in Chapter 18.*

CHANGING PROPERTIES USING GRIPS

You can also use the grips to change the properties of a single or multiple object. To change the properties of an object, select the object to display the grips and then right-click to display the shortcut menu. In the shortcut menu, choose the **Properties** option to display

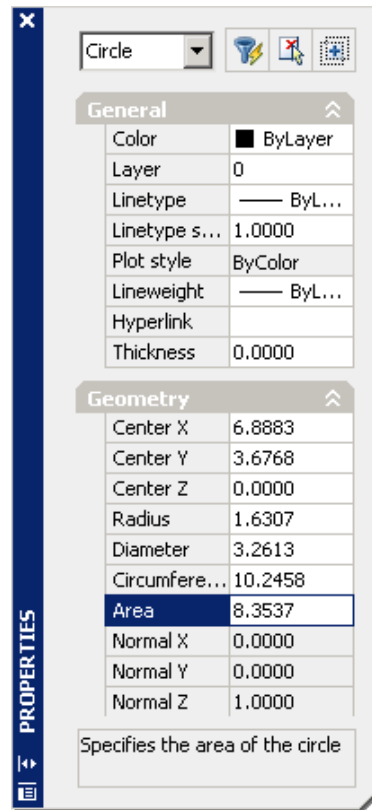


Figure 6-16 PROPERTIES palette for editing the properties of the circle

the **PROPERTIES** palette. If you select a circle, AutoCAD will display **Circle** in the **No selection** drop-down list available on the upper left corner of the **PROPERTIES** palette. Similarly, if you select text, **Text** is displayed in the drop-down list. If you select several objects, AutoCAD will display all the objects in the selection drop-down list of the **PROPERTIES** palette. You can use this palette to change the properties (color, layer, linetype, linetypes scale, lineweight, thickness, and so on) of the gripped objects.

MATCHING PROPERTIES OF THE SKETCHED OBJECTS

Toolbar:	Standard > Match Properties
Menu:	Modify > Match Properties
Command:	MATCHPROP



The **MATCHPROP** command can be used to change some properties like color, layer, linetype, and linetype scale of the selected objects. However, in this case you need a source object whose properties will be forced on the destination objects. When you invoke this command, AutoCAD will prompt you to select the source object and then the

destination objects. The properties of the destination objects will be changed to that of the source object. This command is a transparent command and can be used inside another command. The prompt sequence that will follow when you choose the **Match Properties** button from the **Standard** toolbar is given next.

Select Source Object: *Select the source object.*

Current active settings: Color Layer Ltype Ltscale Lineweight Thickness PlotStyle Text
Dim Hatch Polyline Viewport

Select destination object(s) or [Settings]:

If you select the destination object in the **Select destination object(s) or [Settings]** prompt, the properties of the source object will be forced on it. If you select the **Settings** option, AutoCAD displays the **Property Settings** dialog box (Figure 6-17). The properties displayed in this dialog box are those of the source object. You can use this dialog box to edit the properties that are copied from source to destination objects.

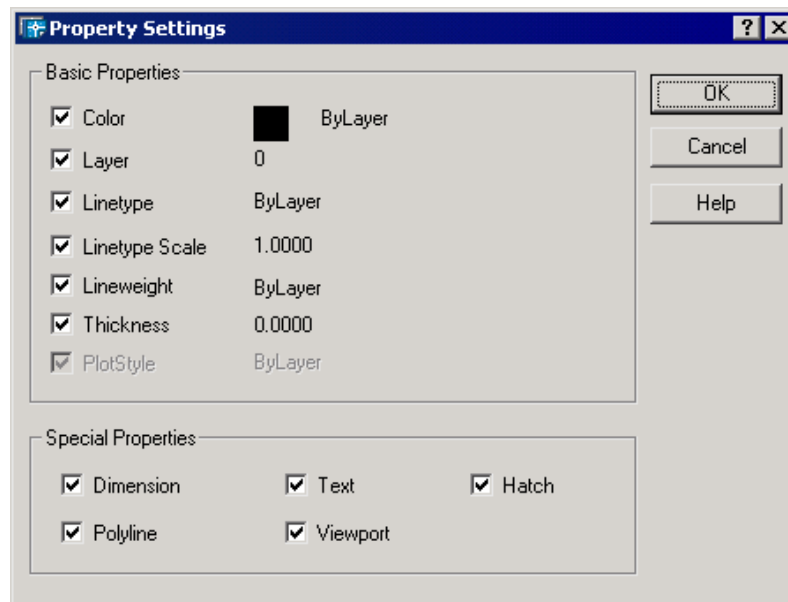


Figure 6-17 Property Settings dialog box

The table shown next lists the properties associated with different types of objects.

Object	Color & layer	Linetype & Linetype Scale	Line-weight	Thick-ness	Text	Dime-nsion	Hatch	Plot style
3dface	x	x	x					x
Arc	x	x	x	x				x
AttDef	x			x	x			x
Body	x	x						x
Circle	x	x	x	x				x
Dimension	x	x				x		x
Ellipse	x	x	x					x
Hatch	x		x				x	x
Image	x	x						x
Insert	x	x	x					x
Leader	x	x	x			x		x
Line	x	x	x	x				x
Mtext	x				x			x
OLE object								
Point	x		x	x				x
2D polyline	x	x	x		x		x	
3D polyline	x	x	x				x	
3D mesh	x	x	x				x	
Pface mesh	x	x	x				x	
Ray	x	x	x				x	
Region	x	x	x	x			x	
2D solid	x	x	x				x	
ACIS solid	x	x	x				x	
Spline	x	x	x				x	
Text	x	x		x	x			x
Tolerance	x	x	x			x		x
Trace	x	x	x	x			x	
Viewport	x		x					x
Xline	x	x	x				x	
Xref	x	x	x				x	
Zombie	x	x	x				x	

QUICK SELECTION OF THE SKETCHED OBJECTS

Menu: Tools > Quick Select
Command: QSELECT

The **QSELECT** command creates a new selection set that will either include or exclude all objects that match the specified object type and property criteria. The **QSELECT** command can be applied to the entire drawing or existing selection set. If a drawing is partially opened, **QSELECT** does not consider the objects that are not loaded. The **QSELECT** command can be invoked by choosing the **Quick Select** button in the **Properties** palette. In the shortcut

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menu, the **QSELECT** command can be invoked by choosing **Quick Select**. When you invoke this command, the **Quick Select** dialog box will be displayed, see Figure 6-18. The **Quick Select** dialog box specifies the object filtering criteria and creates a selection set from that criteria.

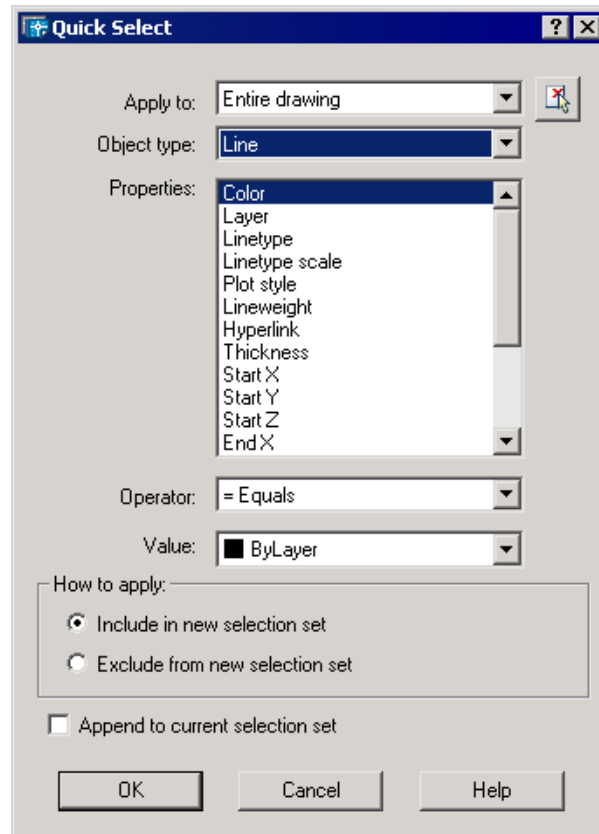


Figure 6-18 Quick Select dialog box

Apply to

The **Apply to** drop-down list specifies whether to apply the filtering criteria to the entire drawing or to the current selection set. If there is an existing selection set, the **Current selection** is the default value. Otherwise the entire drawing is the default value. You can select the objects to create a selection set by choosing the **Select objects** button provided on the right side of this drop-down list. The **Quick Select** dialog box is temporarily closed when you choose this button and you will be prompted to select the objects. The dialog box will be redisplayed once a selection set is made.

Object type

This drop-down list specifies the type of object to be filtered. It lists all the available object types and if some objects are selected, it lists all the selected object types. **Multiple** is the default setting.

Properties

This list box displays the properties to be filtered. All the properties related to the object type will be displayed in this list box. The property selected from this list box will define the options that will be available in the **Operator** and **Value** drop-down list.

Operator

This drop-down list specifies the range of the filter for the chosen property. The filters that are available are given next.

- Equals =
- Not Equal <>
- Greater than >
- Less than <
- Select All
- Wildcard Match (For Hyperlink property)



Note

The **Value** drop-down list will not be available when you select **Select All** from the **Operator** drop-down list.

Value

This drop-down list specifies the property value of the filter. If the values are known, it becomes a list of the available values from which you can select a value. Otherwise you can enter a value.

How to apply Area

The options under this area are used to specify whether the filtered entities will be included or excluded from the new selection set. This area provides the following two radio buttons.

Include in new selection set

If this radio button is selected, the filtered entities will be included in the new selection set. If selected, this radio button creates a new selection set composed only of those objects that conform to the filtering criteria.

Exclude from new selection set

If this radio button is selected, the filtered entities will be excluded from the new selection set. This radio button creates a new selection set of objects that do not conform to the filtering criteria.

Append to current selection set

This creates a cumulative selection set by using multiple uses of Quick Select. It specifies whether the objects selected using the **QSELECT** command replace the current selection set or append the current selection set.

**Tip**

*Quick Select supports custom objects (objects that are created by some other applications) and their properties. If custom objects have other properties than AutoCAD, then the source application of the object should be running for the properties to be available by the **QSELECT**.*

MANAGING CONTENTS USING THE DESIGNCENTER*

Toolbar: Standard > DesignCenter
Menu: Tools > DesignCenter
Command: ADCENTER



The **DESIGNCENTER** window is used to locate and organize drawing data, and to insert blocks, layers, external references, and other customized drawing content. These contents can be selected from either your own files, local drives, a network, or the Internet. You can even access and use the contents between files or from the Internet. You can use the **DESIGNCENTER** to conveniently drag and drop any information that has been previously created into a current drawing. This is a powerful tool that reduces repetitive tasks of again creating information that already exists. To invoke the **DESIGNCENTER** window, choose the **DesignCenter** button from the **Standard** toolbar. The **DESIGNCENTER** window is displayed, see Figure 6-19.

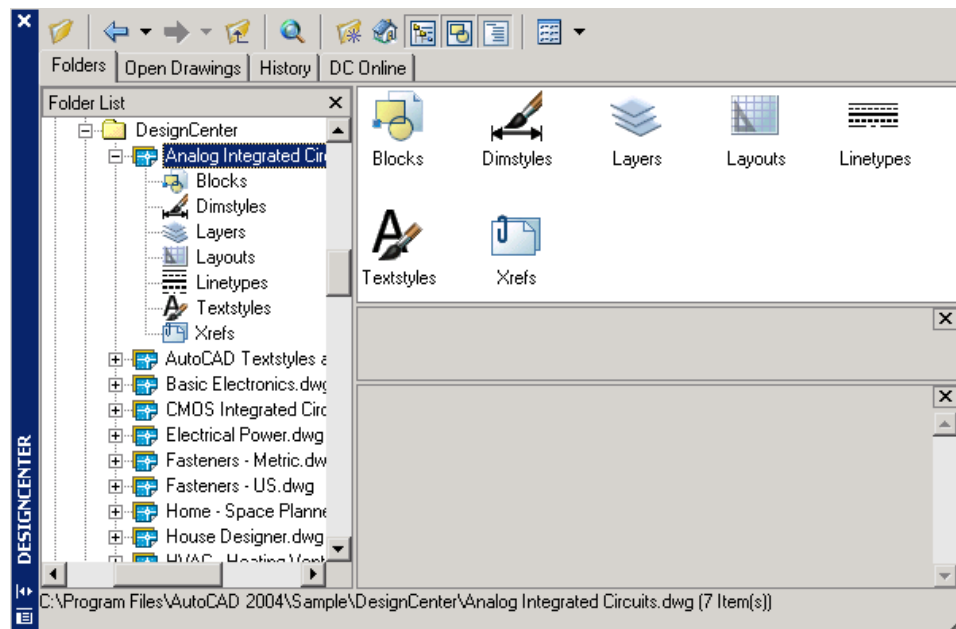


Figure 6-19 DESIGNCENTER window

This window can be moved to any location on the screen by picking and dragging it with the grab bar located on the left of the window. You can also resize it by clicking the borders and dragging them to the right or left. Right-clicking on the title bar of the window displays a

shortcut menu that gives options to move, resize, close, dock, and hide the **DESIGNCENTER** window. The **Auto-Hide** button available on the grab bar acts as a toggle for hiding and displaying the **DESIGNCENTER**. Also, double-clicking on the title bar of the window docks the **DESIGNCENTER** window if it is undocked and vice versa. To use this option make sure that the **Allow Docking** option is selected from the shortcut menu that is displayed by right-clicking on the grab bar.



Note

The **DESIGNCENTER** can be turned on and off by pressing the **CTRL+2** keys.

Figure 6-20 shows the **DESIGNCENTER** toolbar buttons. When you choose the **Tree View Toggle** button on the **DESIGNCENTER** toolbar, it displays the **Tree View** (Left Pane) with a tree view of the contents of the drives. If the tree view is not displayed, you can also right-click in the window and choose **Tree** from the shortcut menu that is displayed. Now, the window is divided into two parts, the **Tree View** (left pane) and the **Palette** (right pane). The **Palette** displays folders, files, objects in a drawing, images, Web-based content, and custom content. You can also resize both the **Tree View** and the **Palette** by clicking and dragging the bar between them to the right or the left.

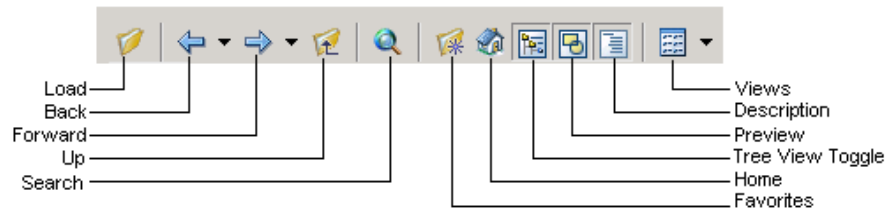


Figure 6-20 **DESIGNCENTER** toolbar buttons

The **DESIGNCENTER** has four tabs provided below the **DESIGNCENTER** toolbar buttons. They are **Folders**, **Open drawings**, **History**, and **DC Online**. The description of these tabs is given next.

Folders Tab*

The **Folders** tab lists all the folders and files available in the local and network drives. When this tab is selected, the **Tree View** displays the tree view of the contents of the drives and the **Palette** displays the various folders, and files in a drawing, images, and the Web-based content available in the selected drive.

In the **Tree View** you can browse the contents of any folder by clicking on the plus sign (+) adjacent to it to expand the view. Further expanding the contents of a file displays the categories such as **Blocks**, **Dimstyles**, **Layers**, **Layouts**, **Linetypes**, **Textstyles**, and **Xrefs**. Clicking on any one of these categories in the **Tree View** displays the listing under the selected category in the **Palette** (Figure 6-21). Alternately, right-clicking a particular folder, file, or category of the file contents displays a shortcut menu. The **Explore** option in this shortcut menu also further expands the selected folder, file, or category of contents to display the

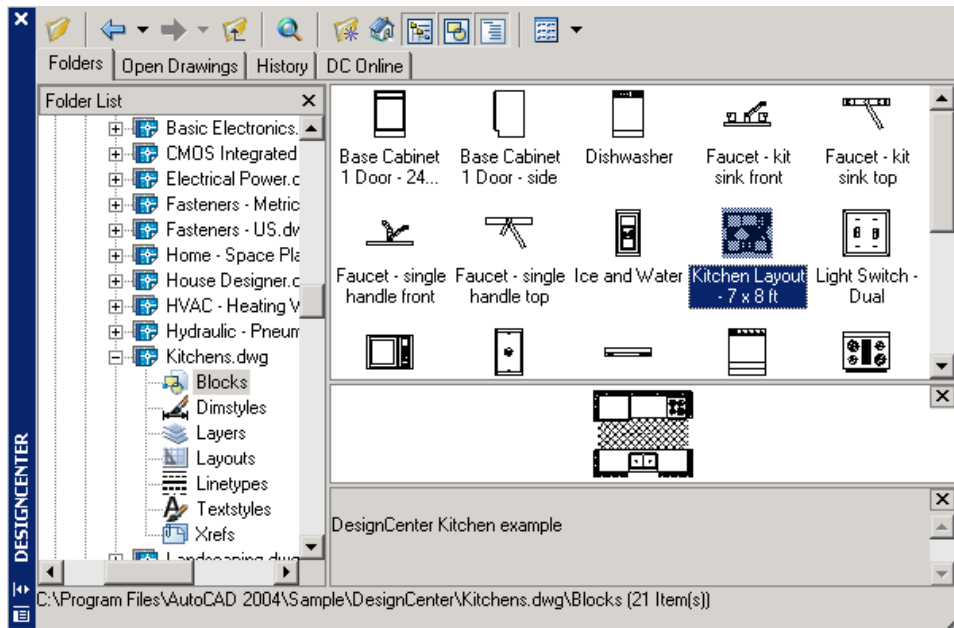


Figure 6-21 *DESIGNCENTER displaying Tree pane, Palette, Preview pane, and the Description box*

listing of contents respectively. Choosing the **Preview** button from the toolbar displays an image of the selected object or file in a **Preview pane** below the Palette. Choosing the **Description** button displays a brief text description of the selected item if it has one in the **Description** box. When you click on a specific block name in the palette, its preview image and description that was defined earlier when creating the block are displayed in the Preview pane and the Description box respectively.

You can drag and drop any of the contents into the current drawing, or add them by double-clicking on the contents. These are then reused as part of the current drawing. When you double-click on specific Xrefs and blocks, AutoCAD displays the **External Reference** dialog box and the **Insert** dialog box respectively to help in attaching the external reference and inserting the block, respectively. Right-clicking a block displays the options of **Insert Block**, **Copy**, or **Create Tool Palette** and right-clicking an Xref displays the options of **Attach Xref** or **Copy** in the shortcut menus. Similarly, when you double-click a layer, text style, dimstyle, layout, or linetype style, they also get added to the current drawing. If any of these named objects already exist in the current drawing, duplicate definition is ignored and it is not added again. When you right-click on a specific linetype, layer, textstyle, layout, or dimstyle in the palette, a shortcut menu is displayed that gives you an option to **Add** or **Copy**. The **Add** option directly adds the selected named object to the current drawing. The **Copy** option copies the specific named object to the clipboard from where you can paste it into a particular drawing.

**Note**

You will learn more about inserting blocks in Chapter 14, Working with Blocks.

Right-clicking a particular folder or file in the **Tree View** displays a shortcut menu. The various options available in the shortcut menu besides those discussed earlier are **Add to Favorites**, **Organize Favorites**, **Create Tool Palette**, and **Set as Home**. **Add to Favorites** adds the selected file or folder to the **Favorites** folder, which contains the most often accessed files and folders. **Organize Favorites** allows you to reorganize the contents of the **Favorites** folder. When you select **Organize Favorites** from the shortcut menu, the **Autodesk** folder is opened in a window. **Create Tool Palette** adds the blocks of the selected file or folder to the **TOOL PALETTES** window, which contains the predefined blocks. **Set as Home** sets the selected file or folder as the **Home** folder. You will notice that when the **Design Center** command is invoked the next time, the file that was last set as the **Home** folder is displayed selected in the **DESIGNCENTER**.

Open Drawings Tab*

The **Open Drawings** tab lists all the drawings that are open, including the current drawing which is being worked on. When you select this tab, the **Tree View** (left pane) displays the tree view of all the drawings that are currently open and the **Palette** (right pane) displays the various contents in the selected drawing.

History Tab*

The **History** tab lists the most recent locations accessed through the **DESIGNCENTER**. When you select this tab, the **Tree View** (left pane) and the **Palette** (right pane) are replaced by a list box. Right-clicking a particular file displays a shortcut menu. The various options available in the shortcut menu are **Explore**, **Folders**, **Open Drawings**, **Delete**, and **Search**. The **Explore** option invokes the **Folders** tab of the **DESIGNCENTER** with the file selected in the **Tree View** and the contents available in the selected file displayed in the **Palette View**. The **Folders** option invokes the **Folders** tab of the **DESIGNCENTER**. The **Open Drawings** option invokes the **Open Drawings** tab of the **DESIGNCENTER**. The **Delete** option deletes the selected drawing from the History list. The **Search** option allows you to search for drawings or named objects such as blocks, textstyles, dimstyles, layers, layouts, external references, or linetypes. When you choose the **Search** option from the shortcut menu, the **Search** dialog box is displayed as shown in Figure 6-22.

You can select the type of object you want to search for from the **Look for** drop-down list. You can look for a drawing or a named object in your hard drives. If you are looking for a named object, you can first select the drive where you think it might be located from the **In** drop-down list or use the **Browse** button to locate a folder. The **Search subfolders** check box is selected by default. Then, you can enter the name of the named object you are looking for in the **Search for the word(s)** text box and then choose the **Search Now** button to start the search operation.

If you are looking for drawing files, three tabs are available in the **Search** dialog box that

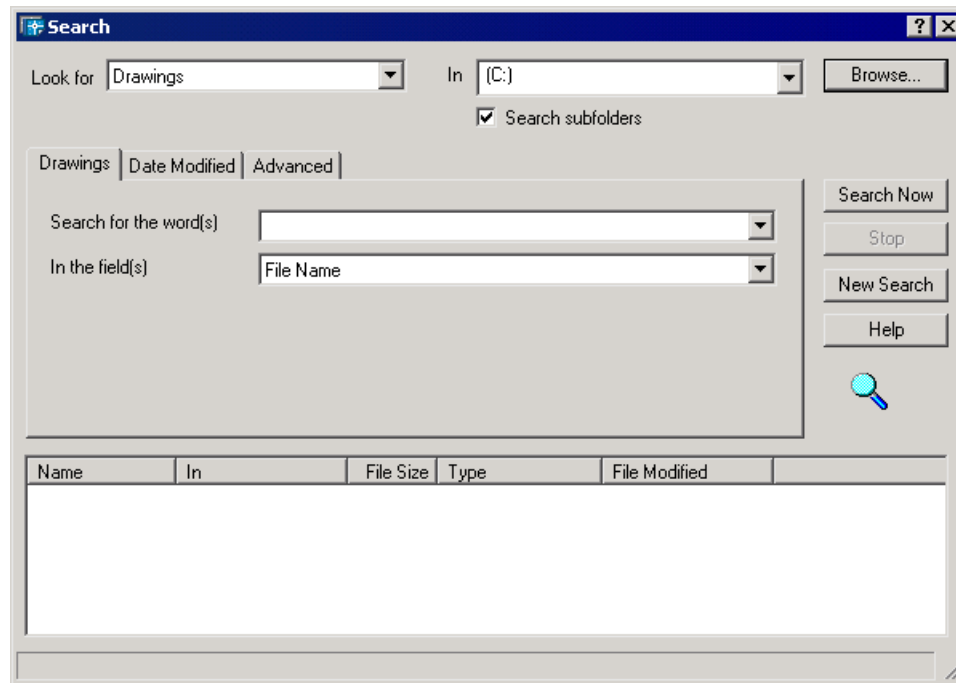


Figure 6-22 Search dialog box

provide additional search criteria. You can conduct a search based on the following additional criteria.

1. Words that are part of the **File name**, **Title**, **Subject**, **Author's name**, and **keyword text** stored in the **summary tab** of the **Drawing Properties**. In the **Drawings** tab, enter the words in the **Search for the word(s)** edit box. The field in which the particular words are to be located can be selected from the **In the field(s)** drop-down list of the **Search** dialog box. You can enter text that might be part of the File name, title, subject, author, or description text of a particular drawing.
2. **Date last modified on**. You can choose the **Date Modified** tab in the **Search** dialog box and select the appropriate radio buttons there.
3. In the **Advanced** tab, you can select the category to look for from the **Containing** drop-down list of the **Search** dialog box. You can choose either of the following categories on which to base your search: **Block name**, **Block and drawing description**, **Attribute tag**, or **Attribute value**. If the named object or drawing you are searching for does not contain any blocks or attributes, and does not have any drawing description, you need not select any of these from the **Containing** drop-down list. In such a case, the Containing text edit box is not available. The text blocks, attributes, or drawing description can be entered in the **Containing text** edit box. You can also specify the **At least** or **At most** sizes of the file you are searching for in the **Size is** edit boxes to further narrow down the search.

After you have entered the appropriate criteria for searching the specific file, choose the **Search Now** button. The name, location, file size, type, and date last modified on are displayed at the bottom of the dialog box under the **Name**, **In**, **File Size**, **Type**, and **File Modified** columns of the list box respectively.

Now, when you double-click on the drawing name, it gets highlighted in the tree view of the **DESIGNCENTER**, and its contents are displayed in the palette, or it gets loaded in the palette. You can also right-click on the drawing name in the list box of the **Search** dialog box to display a shortcut menu. This shortcut menu provides you with the options of loading a drawing in the palette, inserting it as a block, attaching it as an external reference, opening it as a current drawing, or copying it to the pasteboard.

DC Online Tab*

The **DC Online** tab allows you to download the symbols, information regarding various manufacturer's products, and the online catalogs of various products from the **DesignCenter Online** window. To access the **DesignCenter Online** after establishing the Web connection choose the **Reconnect to DesignCenter** button. In the **DesignCenter Online** window the **Tree View** displays various folders under the **Standard Parts**, **Manufactures**, and the **Aggregators** heading. You can select the desired folder from the **Tree View** and the various contents available in the selected folder are displayed on the **Palette**. The preview and the description of the selected content are displayed in the Preview window. You can double-click or drag and drop the selected content from the Web page in the current drawing.

Choosing the **Back** button in the **DESIGNCENTER** toolbar displays the last item selected in the **DESIGNCENTER**. If you pick the down arrow available on the **Back** button, a list of the five recently visited items is displayed. You can view the desired item in the **DESIGNCENTER** by selecting it from the list. The **Forward** button is available only if you have chosen the **Back** button once. This button displays the same page as the current page before you choose the **Back** button. The **Up** button moves one level up in the tree structure from the current location. Choosing the **Favorites** button displays shortcuts to files and folders that are accessed frequently by you and are stored in the **Favorites** folder. This reduces the time you take to access these files or folders from their normal location. Choosing the **Tree View Toggle** button in the **DESIGNCENTER** toolbar displays or hides the tree pane with the tree view of the contents in a hierarchical form. Choosing the **Load** button displays the **Load** dialog box, as shown in Figure 6-23, whose options are similar to those of the standard **Select file** dialog box. When you select a file here and choose the **Open** button, AutoCAD displays the selected file and its contents in the **DESIGNCENTER**.

The **Views** button gives four display format options for the contents of the palette: **Large icons**, **Small icons**, **List**, and **Details**. The **List** option lists the contents in the palette while the **Details** option gives a detailed list of the contents in the palette with the name, file size, and type.

Right-clicking in the palette displays a shortcut menu with all the options provided in the

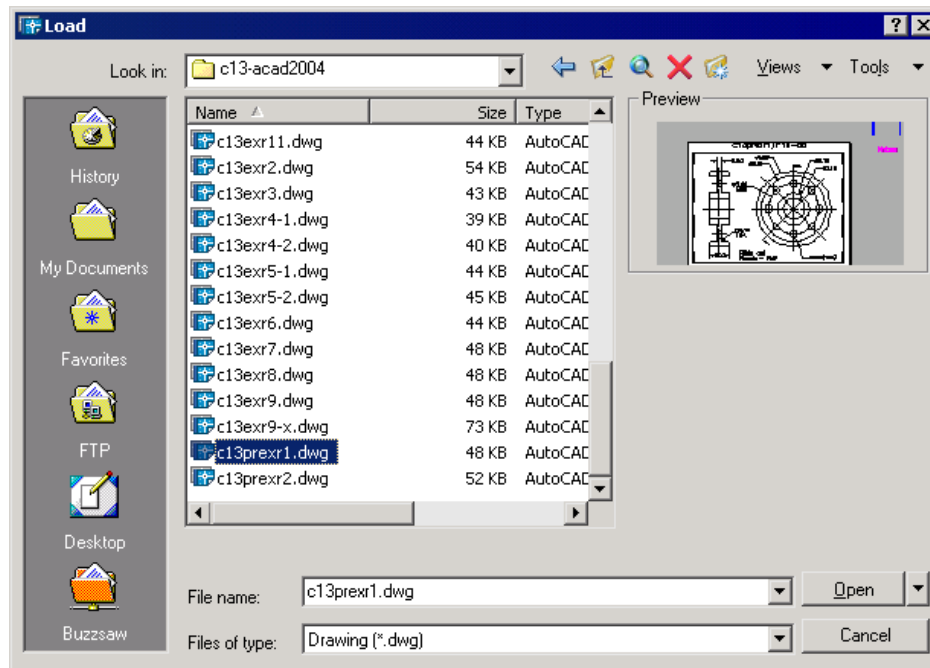


Figure 6-23 Load dialog box

DESIGNCENTER in addition to the **Add to Favorites**, **Organize favorites**, **Refresh**, and **Create Tool Palette of Blocks** options. The **Refresh** option refreshes the palette display if you have made any changes to it. The **Create Tool Palette of Blocks** option adds the drawings of the selected file or folder to the **TOOL PALETTES**, which contains the predefined blocks. The following example will illustrate how to use the **DESIGNCENTER** to locate a drawing and then use its contents into a current drawing.

Example 1

Architectural

Use the **DESIGNCENTER** to locate and view contents of the drawing *Kitchens*. Also, use the **DESIGNCENTER** to insert a block from this drawing and also import a layer and textstyle from the *8th floor lighting.dwg* located in the **Sample** folder. Use these to make a drawing of a Kitchen plan (*MyKitchen.dwg*) and then add text to it as shown in Figure 6-24.

1. Open a new drawing using the **Start from Scratch** option. Make sure to select the **Imperial (feet and inches)** option in the **Create New Drawing** dialog box.
2. Change the units to **Architectural** using the **Drawing Units** dialog box. Increase the limits to 10', 10'. Invoke the **ALL** option of the **ZOOM** command to increase the drawing display area.

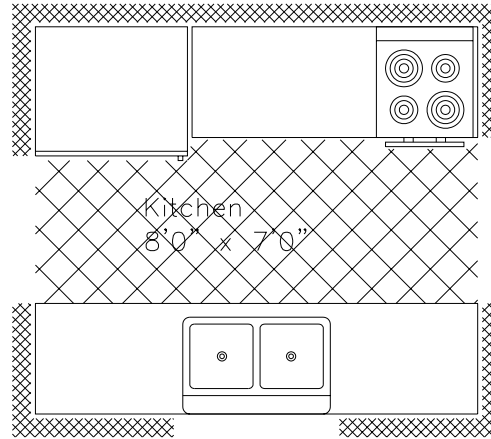


Figure 6-24 Drawing for Example 1

3. Choose the **DesignCenter** button from the **Standard** toolbar. The **DESIGNCENTER** window is displayed at its default location.
4. In the **DESIGNCENTER** toolbar, choose the **Tree View Toggle** button to display the **Tree View** and the **Palette** (if not already displayed). Also, choose the **Preview** button. You can resize the window, if need be, to view both the **Tree View** and the **Palette**, conveniently.
5. Choose the **Search** button in the **DESIGNCENTER** to display the **Search** dialog box. Here, select **Drawings** from the **Look for** drop-down list and **C:** from the **In** drop-down list. Select the **Search subfolders** check box. In the **Drawings** tab, type **Kitchens** in the **Search for the word(s)** edit box and select **File Name** from the **In the field(s)** drop-down list. Now, choose the **Search Now** button to commence the search. After the drawing has been located, its details and path are displayed in a list box at the bottom of the dialog box.
6. Now, right-click on *Kitchens.dwg* in the list box of the **Search** dialog box and choose **Load into Content Area** from the shortcut menu. You will notice that the drawing and its contents are displayed in the **Tree view**.
7. Close the **Search** dialog box.
8. Double-click on *Kitchens.dwg* in the **Tree View** to display its contents, in case they are not displayed. You can also expand the contents by clicking on the + sign located on the left of the file name in the **Tree view**.
9. Select **Blocks** in the **Tree View** to display the list of blocks in the drawing in the **Palette**. Using the left mouse button, drag and drop the block **Kitchen Layout-7x8 ft** in the current drawing.

10. Now, double-click on the *8th floor lighting.dwg* file located in the *Sample* folder in the same directory to display its contents in the **Palette**.
11. Select **Layers** in the **Tree View** to display the layers in the drawing. Drag and drop or double-click the layer **LOGO** from the **Palette** to the current drawing. Now, you can use this layer for placing the text in the current drawing after making it the current layer.
12. Select **Textstyles** to display the list of Textstyles in the **Palette**. Select **ROMANS** in the **Palette** and drag and drop it in the current drawing. You can use this textstyle for adding text to the current drawing.
13. Use the imported data to add text to the current drawing and complete it as shown in the Figure 6-24.
14. Save the current drawing as *MyKitchen.dwg*.

MAKING INQUIRIES ABOUT OBJECTS AND DRAWINGS

When you create a drawing or examine an existing one, you often need some information about the drawing. In manual drafting, you inquire about the drawing by performing measurements and calculations manually. Similarly, when drawing in an AutoCAD environment, you will need to make inquiries about data pertaining to your drawing. The inquiries can be about the distance from one location on the drawing to another, the area of an object like a polygon or circle, coordinates of a location on the drawing, and so on. AutoCAD keeps track of all the details pertaining to a drawing. Since inquiry commands are used to obtain information about the selected objects, these commands do not affect the drawings in any way. The following is the list of Inquiry commands:

AREA	DIST	ID	LIST	DBLIST
STATUS	TIME	DWGPROPS	MASSPROP	



Note

The **MASSPROP** command will be discussed in Chapter 25.

For most of the Inquiry commands, you are prompted to select objects; once the selection is complete, AutoCAD switches from graphics mode to text mode, and all the relevant information about the selected objects is displayed. For some commands information is displayed in the AutoCAD Text Window. The display of the text screen can be tailored to your requirements using a pointing device. Therefore, by moving the text screen to one side, you can view the drawing screen and the text screen simultaneously. If you select the **minimize** button or select the close button, you will return to the graphics screen. You can also return to the graphics screen by entering the **GRAPHSCR** command at the Command prompt. Similarly, you can return to the AutoCAD Text Window by entering **TEXTSCR** at the Command prompt.

Measuring the Area of the Objects

Toolbar: Inquiry > Area
Menu: Tools > Inquiry > Area
Command: AREA



Finding the area of a shape or an object manually is time-consuming. In AutoCAD, the **AREA** command is used to automatically calculate the area of an object in square units. This command saves time when calculating the area of shapes, especially when the shapes are complicated or irregular.

You can use the default option of the **AREA** command to calculate the area and perimeter or circumference of the space enclosed by the sequence of specified points. For example, to find the area of an object (one which is not formed of a single object) you have created with the help of the **LINE** command (Figure 6-25), you need to select all the vertices of that object. By selecting the points, you define the shape of the object whose area is to be found. This is the default method for determining the area of an object. The only restriction is that all the points you specify should be in a plane parallel to the XY plane of the current UCS. You can make the best possible use of object snaps such as **ENDpoint**, **INTERsect**, and **TANGent**, or even use running Osnaps, to help you select the vertices quickly and accurately. For AutoCAD to find the area of a shape, the shapes need not have been drawn with polylines; nor do the lines need to be closed. However, curves must be approximated with short straight segments. In such cases, AutoCAD computes the area by assuming that the first point and the last point are joined. The prompt sequence that will follow when you choose the **Area** button from the **Inquiry** toolbar is given next.

Select all the vertices
using OSNAP

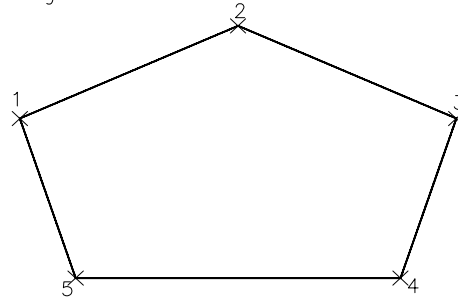



Figure 6-25 Using the **AREA** command

Specify first corner point or [Object/Add/Subtract]: *Specify first point.*
 Specify next corner point or press ENTER for total: *Specify the second point.*
 Specify next corner point or press ENTER for total: *Continue selecting until all the points enclosing the area have been selected.*
 Specify next corner point or press ENTER for total: 
 Area = X, Perimeter = Y



Here, X represents the numerical value of the area and Y represents the circumference/perimeter. It is not possible to accurately determine the area of a curved object, such as an arc, with the default (Point) option. However, the approximate area under an arc can be calculated by specifying several points on the given arc. If the object whose area you want to find is not closed (formed of independent segments) and has curved lines, you should use the following steps to determine the accurate area of such an object.

1. Convert all the segments in that object into polylines using the **PEDIT** command.
2. Join all the individual polylines into a single polyline. Once you have performed these operations, the object becomes closed and you can then use the **Object** option of the **AREA** command to determine the area.

If you specify two points on the screen, the **AREA** command will display the value of the area as 0.00; the perimeter value is the distance between the two points.

Object Option

You can use the **Object** option to find the area of objects such as polygons, circles, polylines, regions, solids, and splines. If the selected object is a polyline or polygon, AutoCAD displays the area and perimeter of the polyline. In case of open polylines, the area is calculated assuming that the last point is joined to the first point but the length of this segment is not added to the polyline length unlike the default option. If the selected object is a circle, ellipse, or planar closed spline curve, AutoCAD will provide information about its area and circumference. For a solid, the surface area is displayed. For a 3D polyline, all vertices must lie in a plane parallel to the *XY* plane of the current UCS. The extrusion direction of a 2D polyline whose area you want to determine should be parallel to the *Z* axis of the current UCS. In case of polylines which have a width, the area and length of the polyline are calculated using the centerline. If any of these conditions is violated, an error message is displayed on the screen. The following prompt sequence appears when you choose the **Area** button.

Specify first corner point or [Object/Add/Subtract]: O 
Select objects : Select an object 
Area = (X), Circumference = (Y)

X represents the numerical value of the area, and Y represents the circumference/perimeter.



Tip

*In many cases, the easiest and most accurate way to find the area of an area enclosed by multiple objects is to use the **BOUNDARY** command to create a polyline, and then use the **AREA Object** option.*

Add Option

Sometimes you want to add areas of different objects to determine a total area. For example, in the plan of a house, you need to add the areas of all the rooms to get the total floor area. In such cases, you can use the **Add** option. Once you invoke this option, AutoCAD activates the **Add** mode. By using the **First corner point** option at the **Specify first corner point or [Object/Subtract]:** prompt, you can calculate the area and perimeter by selecting points on the screen. Pressing ENTER after you have selected points defining the area that is to be added, calculates the total area, since the **Add** mode is on. The command prompt is as follows.

Specify next corner point or press ENTER for total [ADD mode]:

If the polygon whose area is to be added is not closed, the area and perimeter are calculated assuming that a line that connects the first point to the last point is added to close the

polygon. The length of this area is added in the perimeter. The **Object** option adds the areas and perimeters of selected objects. While using this option, if you select an open polyline, the area is calculated considering the last point is joined to the first point but the perimeter does not consider the length of this assumed segment, unlike the **First corner point** option. When you select an object, the area of the selected object is displayed on the screen. At this time the total area is equal to the area of the selected object. When you select another object, AutoCAD displays the area of the selected object as well as the combined area (total area) of the previous object and the currently selected object. In this manner you can add areas of different objects. Until the **Add** mode is active, the string **ADD** mode is displayed along with all subsequent object selection prompts to remind you that the **Add** mode is active. When the **AREA** command is invoked, the total area is initialized to zero.

Subtract Option

The action of the **Subtract** option is the reverse of that of the **Add** option. Once you invoke this option, AutoCAD activates the **Subtract** mode. The **First corner point** and **Object** options work similar to the way they work in the **ADD** mode. When you select an object, the area of the selected object is displayed on the screen. At this time, the total area is equal to the area of the selected object. When you select another object, AutoCAD displays the area of the selected object as well as the area obtained by subtracting the area of the currently selected object from the area of the previous object. In this manner, you can subtract areas of objects from the total area. Until the **Subtract** mode is active, the string **SUBTRACT** mode is displayed along with all subsequent object selection prompts, to remind you that the **Subtract** mode is active. To exit the **AREA** command, press ENTER (null response) at the **Specify first corner point or [Object/Add/Subtract]** prompt. The prompt sequence for these two modes for Figure 6-26 is given next.

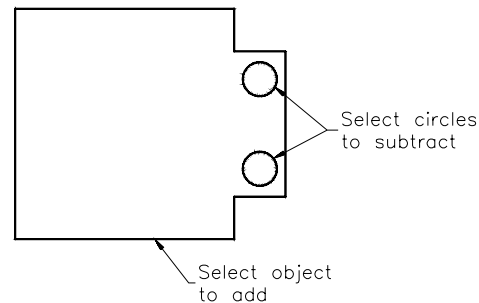


Figure 6-26 Using the **Add** and **Subtract** options

```
Specify first corner point or [Object/Add/Subtract]: A 
Specify first corner point or [Object/Subtract]: O 
(ADD mode) Select objects: Select the polyline.
Area = 2.4438, Perimeter = 6.4999
Total area = 2.4438
(ADD mode) Select objects: 
Specify first corner point or [Object/Subtract]: S 
Specify first corner point or [Object/Add]: O 
(SUBTRACT mode) Select object: Select the circle.
Area = 0.0495, Circumference = 0.7890
Total area = 2.3943
(SUBTRACT mode) Select objects: Select the second circle.
```


Area = 0.0495, Circumference = 0.7890

Total area = 2.3448

(SUBTRACT mode) Select object:

Specify first corner point or [Object/Add]:

The **AREA** and **PERIMETER** system variables hold the area and perimeter (or circumference in the case of circles) of the previously selected polyline (or circle). Whenever you use the **AREA** command, the **AREA** variable is reset to zero.



Tip

If an architect wants to calculate the area of flooring and skirting in a room, the **Area** command provides you with the area and the perimeter of the room. You can use these parameters to calculate the skirting.

Measuring the Distance Between Two Points

Toolbar: Inquiry > Distance
Menu: Tools > Inquiry > Distance
Command: DIST



The **DIST** command is used to measure the distance between two selected points (Figure 6-27). The angles that the selected points make with the X axis and the XY plane are also displayed. The measurements are displayed in current units. Delta X (horizontal displacement), delta Y (vertical displacement), and delta Z are also displayed. The distance computed by the **DIST** command is saved in the **DISTANCE** variable. The prompt sequence that will follow when you choose the **Distance** button is given next.

Specify first point: *Specify a point.*

Specify second point: *Specify a point.*

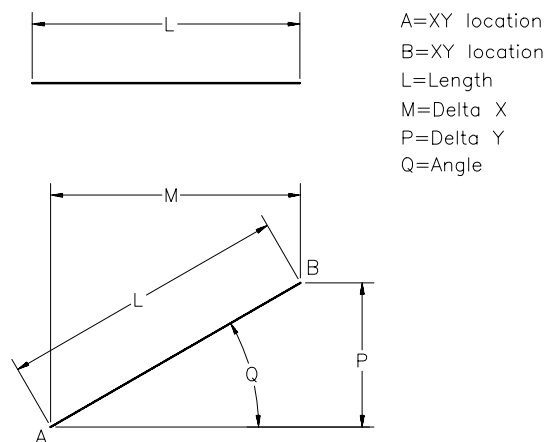


Figure 6-27 Using the **DIST** command

AutoCAD returns the following information.

Distance = *Calculated distance between the two points.*
 Angle in XY plane = *Angle between the two points in the XY plane.*
 Angle from XY plane = *Angle the specified points make with the XY plane.*
 Delta X = *Change in X, Delta Y = Change in Y, Delta Z = Change in Z.*

If you enter a single number or fraction at the **Specify first point** prompt, AutoCAD will convert it into the current unit of measurement and display it in the command line.

Command: **DIST**
 First point: 3-3/4 (Enter a number or a fraction.)
 Distance = 3.7500



Note

The Z coordinate is used in 3D distances. If you do not specify the Z coordinates of the two points between which you want to know the distance, AutoCAD takes the current elevation as the Z coordinate value.

Identifying the Location of a Point on the Screen


Toolbar: Inquiry > Locate Point
Menu: Tools > Inquiry > ID Point
Command: ID



The **ID** command is used to identify the position of a point you specify by displaying the X, Y, and Z coordinates of the point. The prompt sequence that will follow when you choose the **Locate Point** button from the **Inquiry** toolbar is given next.

Specify point: *Specify the point to be identified.*
 X = X coordinate Y = Y coordinate Z = Z coordinate

AutoCAD takes the current elevation as the Z coordinate value. If an **Osnap** mode is used to snap to a 3D object in response to the **Specify point** prompt, the Z coordinate displayed will be that of the selected feature of the 3D object. You can also use the **ID** command to identify the location on the screen. This can be realized by entering the coordinate values you want to locate on the screen. AutoCAD identifies the point by drawing a blip mark at that location if the **BLIPMODE** system variable is on. For example, the following is the prompt sequence to find where the position X = 2.345, Y = 3.674, and Z = 1.0000 is located on the screen.

Specify point: 2.345,3.674,1.00 
 X = 2.345 Y = 3.674 Z = 1.0000

The coordinates of the point specified in the **ID** command are saved in the **LASTPOINT** system variable. You can locate a point with respect to the **ID** point by using the relative or polar coordinate system. You can also snap to this point by typing @ when AutoCAD prompts for a point.


Listing Information About Objects

Toolbar:	Inquiry > List
Menu:	Tools > Inquiry > List
Command:	LIST



The **LIST** command displays all the information pertaining to the selected objects. The information is displayed in the AutoCAD Text Window. The prompt sequence that follows when you choose the **List** button from the **Inquiry** toolbar is given next.

Select objects: *Select objects whose data you want to list.*

Select objects: 

Once you select the objects to be listed, AutoCAD shifts you from the graphics screen to the AutoCAD Text Window. The information displayed (listed) varies from object to object. The information on an object's type, its coordinate position with respect to the current UCS (user coordinate system), the name of the layer on which it is drawn, and whether the object is in model space or paper space is listed for all types of objects. If the color, lineweight, and the linetype are not BYLAYER, they are also listed. Also, if the thickness of the object is greater than 0, that is also displayed. The elevation value is displayed in the form of a Z coordinate (in the case of 3D objects). If an object has an extrusion direction different from the Z axis of the current UCS, the object's extrusion direction is also provided.

More information based on the objects in the drawing is also provided. For example, for a line the following information is displayed.

1. The coordinates of the endpoints of the line.
2. Its length (in 3D).
3. The angle made by the line with respect to the X axis of the current UCS.
4. The angle made by the line with respect to the XY plane of the current UCS.
5. Delta X, Delta Y, Delta Z: this is the change in each of the three coordinates from the start point to the endpoint.
6. The name of the layer in which the line was created.
7. Whether the line is drawn in Paper space or Model space.

The center point, radius, true area, and circumference of circles is displayed. For polylines, this command displays the coordinates. In addition, for a closed polyline, its true area and perimeter are also given. If the polyline is open, AutoCAD lists its length and also calculates the area by assuming a segment connecting the start point and endpoint of the polyline. In the case of wide polylines, all computation is done based on the centerlines of the wide segments. For a selected viewport, the **LIST** command displays whether the viewport is on and active, on and inactive, or off. Information is also displayed about the status of Hideplot and the scale relative to paper space. If you use the **LIST** command on a polygon mesh, the size of the mesh (in terms of M, X, N), the coordinate values of all the vertices in the mesh, and whether the mesh is closed or open in M and N directions are all displayed. As mentioned before, if all the information does not fit on a single screen, AutoCAD pauses to allow you to press ENTER to continue the listing.

Listing Information About All Objects in a Drawing

Command: DBLIST

The **DBLIST** command displays information pertaining to all the objects in the drawing. Once you invoke this command, information is displayed in the Command prompt. If you want to display the drawing information in the AutoCAD Text Window, press the F2 key on the keyboard. If the information does not fit on a single screen, AutoCAD pauses to allow you to press ENTER to continue the listing. To terminate the command, press ESC. To return to the graphics screen, close the AutoCAD Text Window. This command can be invoked by entering **DBLIST** at the Command prompt.

Checking the Time-Related Information

Menu: Tools > Inquiry > Time
Command: TIME

The time and date maintained by your system are used by AutoCAD to provide information about several time factors related to the drawings. Hence, you should be careful about setting the current date and time in your computer. The **TIME** command can be used to display information pertaining to time related to a drawing and the drawing session. The display obtained by invoking the **TIME** command is similar to the following.

```
Command: TIME
Current time:           Monday, April 26 1999 at 11:22:35.069 AM
Times for this drawing:
Created:                Thursday, April 15 1999 at 09:34:42.157 AM
Last updated:           Friday, April 16 1999 at 02:54:25.700 PM
Total editing time:     0 days 04:05:00.440
Elapsed timer (on):     0 days 04:05:00.390
Next automatic save in: 0 days 01:57:38.181
```

Enter option [Display/ON/OFF/Reset]:

The foregoing display gives you information on the following.

Current Time

Provides today's date and the current time.

Created

Provides the date and time that the current drawing was created. The creation time for a drawing is set to the system time when the **NEW**, **WBLOCK**, or **SAVE** command is used to create that drawing file.

Last updated

Provides the most recent date and time you saved the current drawing. In the beginning, it is

set to the drawing creation time, and it is modified every time you use the **QUIT** or **SAVE** command to save the drawing.

Total editing time

This specifies the total time spent on editing the current drawing since it was created. If you terminate the editing session without saving the drawing, the time you have spent on that editing session is not added to the total time spent on editing the drawing. Also, the last update time is not revised.

Elapsed timer

This timer operates while you are in AutoCAD. You can stop this timer by entering OFF at the **Enter an option [Display/ON/OFF/Reset]:** prompt. To activate the timer, enter ON. If you want to know how much time you have spent on the current drawing or part of the drawing in the current editing session, use the **Reset** option as soon as you start working on the drawing or part of the drawing. This resets the user-elapsed timer to zero. By default, this timer is ON. If you turn this timer OFF, the time accumulated in this timer up to the time you turned it OFF will be displayed.

Next automatic save in

As mentioned in the earlier chapters, AutoCAD automatically saves the drawing after the specified interval of time. The **Next automatic save in** option specifies when the next automatic save will be performed. The automatic save time interval can be set in the **Options** dialog box (**Open and Save** tab) or with the **SAVETIME** system variable. If the time interval has been set to zero, the **TIME** command displays the following message.

Next automatic time save in: <disabled>

If the time interval is not set to zero, and no editing has taken place since the previous save, the **TIME** command displays the following message.

Next automatic time save in: <no modification yet>

If the time interval is not set to zero, and editing has taken place since the previous save, the **TIME** command displays the following message.

Next automatic time save in: 0 days hh:mm:ss.msec

hh stands for hours

mm stands for minutes

ss stands for seconds

msec stands for milliseconds

At the end of the display of the **TIME** command, AutoCAD prompts as follows.

Enter an option [Display/ON/OFF/Reset]:

The information displayed by the **TIME** command is static. This means that the information is not updated dynamically on the screen. If you respond to the last prompt with **DISPLAY** (or **D**), the display obtained by invoking the **TIME** command is repeated. This display contains updated time values.

With the **ON** response, the user-elapsed timer is started, if it was off. As mentioned earlier, when you enter the drawing editor, by default the timer is on. The **OFF** response is just the opposite of the **ON** response and stops the user-elapsed time, if it is on. With the **Reset** option, you can set the user-elapsed time to zero.

Obtaining Drawing Status Information

Menu: Tools > Inquiry > Status
Command: STATUS

The **STATUS** command displays information about the prevalent settings of various drawing parameters, such as snap spacing, grid spacing, limits, current space, current layer, current color, and various memory parameters.

Command: **STATUS**

Once you enter this command, AutoCAD displays information similar to the following:

```

106 objects in Drawing.dwg
Model space limits are   X: 0.0000       Y: 0.0000(On)
                        X: 6.0000       Y: 4.4000
Model space uses         X:0.6335       Y:-0.2459  **Over
                        X:8.0497       Y: 4.9710  **Over
Display shows           X: 0.0000       Y:-0.2459
                        X: 8.0088       Y: 5.5266
Insertion base is       X: 0.0000       Y: 0.0000       Z: 0.0000
Snap resolution is      X: 0.2500       Y: 0.2500
Grid spacing is         X: 0.2500       Y: 0.2500

Current space:          Model space
Current layout:         Model
Current layer:          OBJ
Current color:          BYLAYER .... 7 (white)
Current linetype:       BYLAYER .... CONTINUOUS
Current lineweight:     BYLAYER
Current elevation:      0.0000 thickness: 0.0000
Fill on Grid on Ortho off Qtext off Snap off Tablet on
Object snap modes: None
Free dwg disk space: 2047.7 MBytes
Free temp disk: 2047.7 MBytes
Free physical memory: 15.0 MBytes
Free swap file space: 1987.5 MBytes

```

All the values (coordinates and distances) on this screen are given in the format declared in the **UNITS** command. You will also notice ****Over** in the **Model space uses** or **Paper space uses** line. This signifies that the drawing is not confined within the drawing limits. The amount of memory available on the disk is given in the **Free dwg disk space:** line. Information on the name of the current layer, current color, current space, current linetype, current lineweight, current elevation, snap spacing (snap resolution), grid spacing, various tools that are on or off (such as Ortho, Snap, Fill, Tablet, Qtext), and which object snap modes are active is also provided by the display obtained by invoking the **STATUS** command.

Displaying Drawing Properties

Menu: File > Drawing Properties
Command: DWGPROPS

The **DWGPROPS** command displays information about the drawing properties. On choosing **Drawing Properties** from the **File** Menu, the **Drawing Properties** dialog box is displayed (Figure 6-28). This dialog box has four tabs under which information about the drawing is displayed. The information displayed in this dialog box helps you look for the drawing more easily. The tabs are as follows.

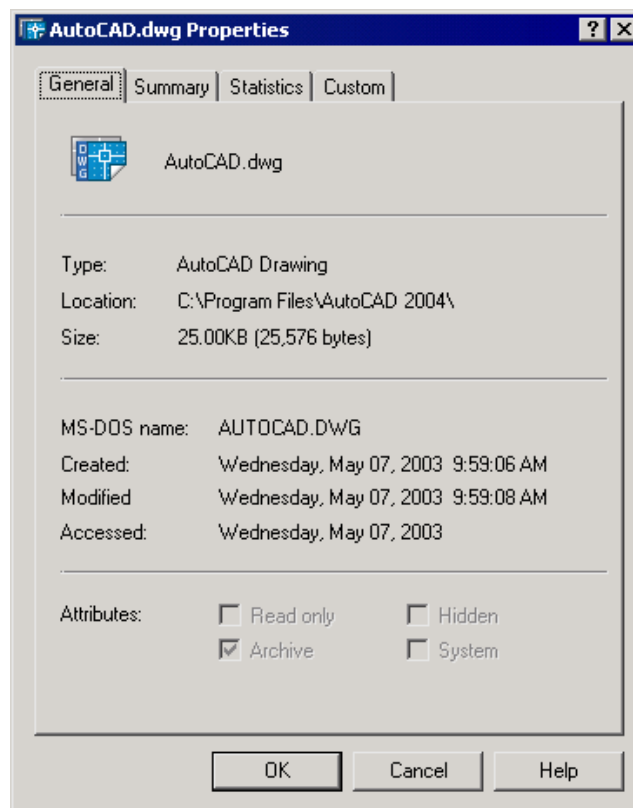


Figure 6-28 Drawing Properties dialog box

General

This tab displays general properties about the drawing like the **Type**, **Size**, and **Location**.

Summary

The **Summary** tab displays predefined properties like the Author, title, and subject.

Statistics

This tab stores and displays data such as the file size and data such as the dates when the drawing was last saved on or modified on.

Custom

This tab displays custom file properties including values assigned by you.

Self-Evaluation Test

Answer the following questions, and then compare your answers to the correct answers given at the end of this chapter.

1. The number of grips depends on the selected object. (T/F)
2. You can use the **Options** dialog box to modify the grips parameters. (T/F)
3. You need at least one source object while using the **MATCHPROP** command. (T/F)
4. You cannot drag and drop the entities from the **DESIGNCENTER** window. (T/F)
5. A grip is a small square that is displayed on an object at its _____ points.
6. A line has _____ grip points and a polyline has _____ .
7. You can enable grips within a block by setting the system variable _____ to 1 (On).
8. The color of the unselected grips can also be changed by using the _____ system variable.
9. You can access the Mirror mode by selecting a grip and then entering _____ or _____ from the keyboard or giving a null response by pressing the SPACEBAR four times.
10. The _____ drop-down list will not be available if you select **Select All** from the **Operator** drop-down list in the **Quick Select** dialog box.

Review Questions

Answer the following questions.

1. If you select a grip of an object, the grip becomes a hot grip. (T/F)
2. To cancel the grip, press the ESC key once. (T/F)
3. The Rotate mode allows you to rotate objects around the base point without changing their size. (T/F)
4. If you have already added hyperlink to the object, you can also use the grips to open a file associated with the hyperlink. (T/F)
5. Which system variable is used to modify the color of the selected grip?
 - (a) **GRIPCOLOR**
 - (b) **GRIPHOT**
 - (c) **GRIPCOLD**
 - (c) **GRIPBLOCK**
6. Which system variable is used to enable the display of the grips inside the blocks?
 - (a) **GRIPCOLOR**
 - (b) **GRIPHOT**
 - (c) **GRIPCOLD**
 - (c) **GRIPBLOCK**
7. Which system variable is used to modify the size of the grips?
 - (a) **GRIPCOLOR**
 - (b) **GRIPSIZE**
 - (c) **GRIPCOLD**
 - (c) **GRIPBLOCK**
8. By holding down which key you can select and make more than one grips hot?
 - (a) SHIFT
 - (b) CTRL
 - (c) ESC
 - (c) ALT
9. Which system variable is used to enable the grip mode?
 - (a) **GRIPCOLOR**
 - (b) **GRIPHOT**
 - (c) **GRIPS**
 - (c) **GRIPBLOCK**
10. When you double-click on a circle, the _____ palette is displayed.
11. The **GRIPSIZE** is defined in pixels, and its value can range from _____ to _____ pixels.
12. When you select a grip for editing, you are automatically in the _____ mode.
13. The _____ mode lets you move the selected objects to a new location.

14. The _____ mode allows you to scale the objects with respect to the base point without changing their orientation.
15. The Mirror mode allows you to mirror the objects across the _____ without changing the size of the objects.

Exercises

Exercise 1

General

1. Use the **LINE** command to draw the shape as shown in Figure 6-29(a).
2. Use grips (Stretch mode) to get the shape as shown in Figure 6-29(b).
3. Use the Rotate and Stretch modes to get the copies as shown in Figure 6-29(c)

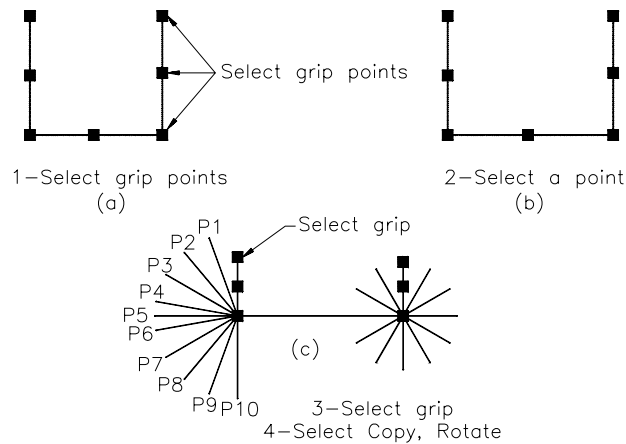


Figure 6-29 Drawing for Exercise 1

Exercise 2

Mechanical

Use the draw and editing commands to make the drawing shown in Figure 6-30.

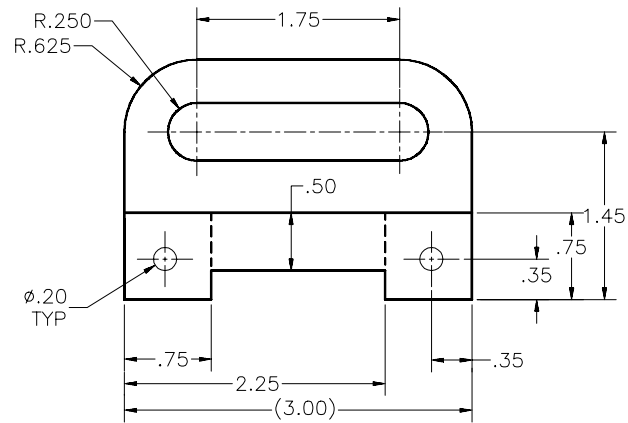


Figure 6-30 Drawing for Exercise 2

Exercise 3

Mechanical

Use the draw and editing commands to make the drawing shown in Figure 6-31.

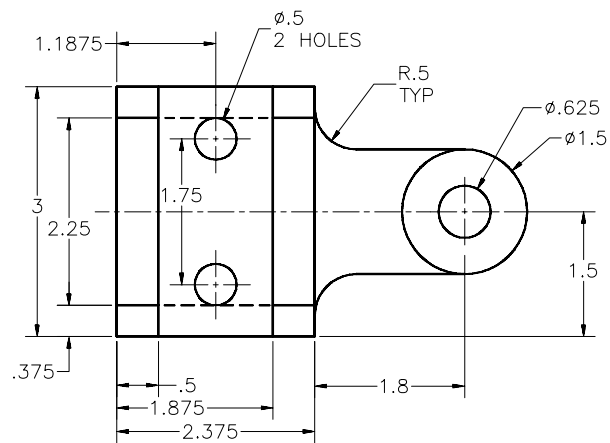


Figure 6-31 Drawing for Exercise 3

Problem Solving Exercise 1*Mechanical*

Use the draw and editing commands to make the drawing shown in Figure 6-32.

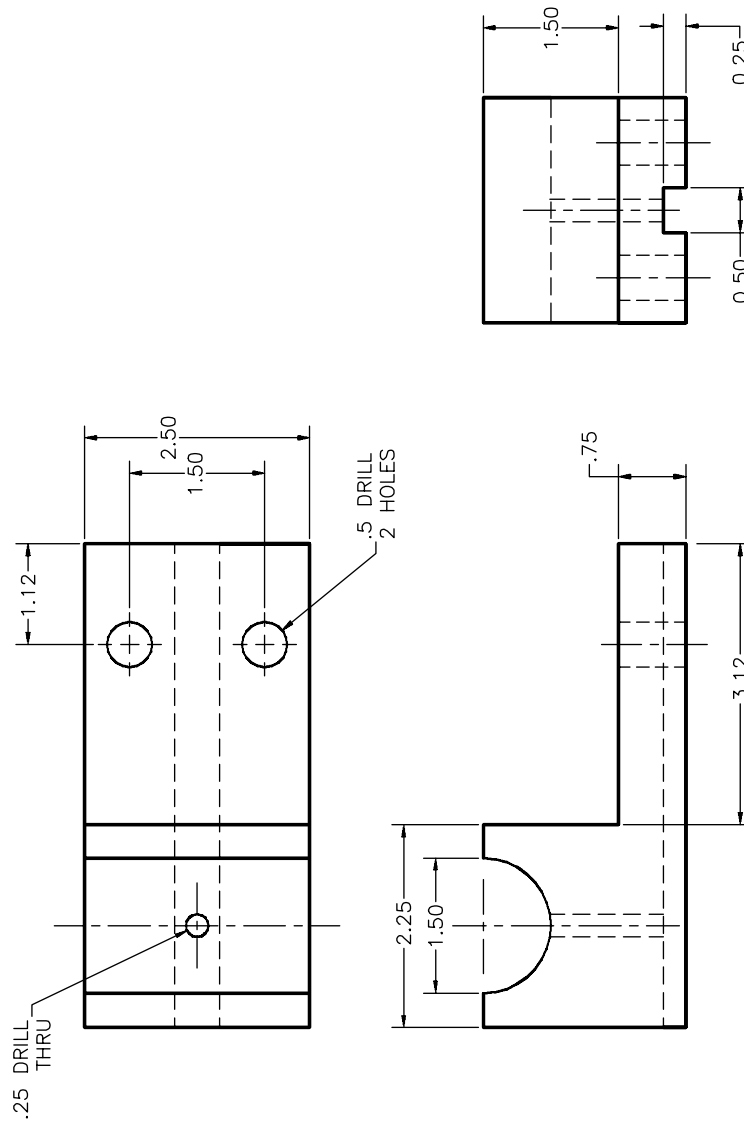


Figure 6-32 Drawing for Problem Solving Exercise 1

Problem Solving Exercise 2*Mechanical*

Use the draw and editing commands to make the drawing shown in Figure 6-33. Assume the missing dimensions.

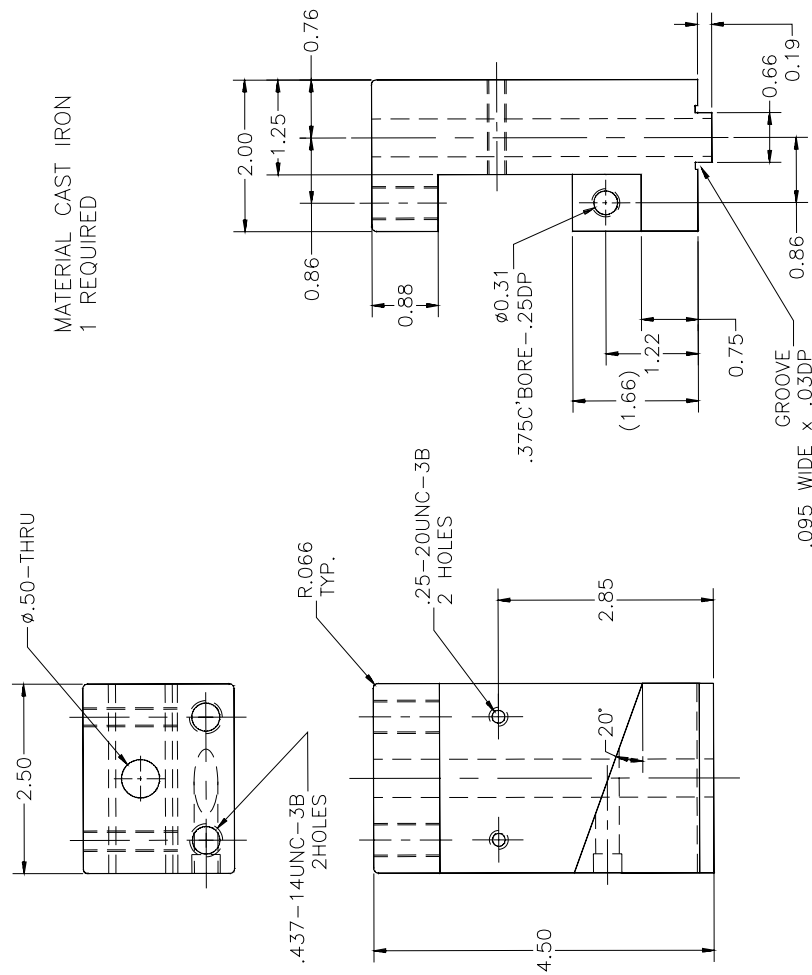


Figure 6-33 Drawing for Problem Solving Exercise 2

Problem Solving Exercise 3*Mechanical*

Draw Figure 6-34 using draw and edit commands. Use the **MIRROR** command to mirror the shape 9 units across the *Y* axis so that the distance between two center points is 9 units. Mirror the shape across the *X* axis and then reduce the mirrored shape by 75 percent. Join the two ends to complete the shape of the open end spanner. Save the file. Assume the missing dimensions. Note that this is not a standard size spanner.

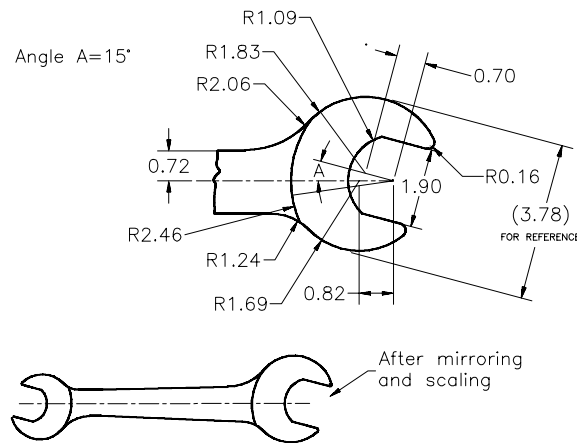


Figure 6-34 Drawing for Problem Solving Exercise 3

Problem Solving Exercise 4*Architectural*

Draw the reception desk shown in Figure 6-35. To get the dimensions of the chairs, refer to Problem Solving Exercise 3 of Chapter 5.

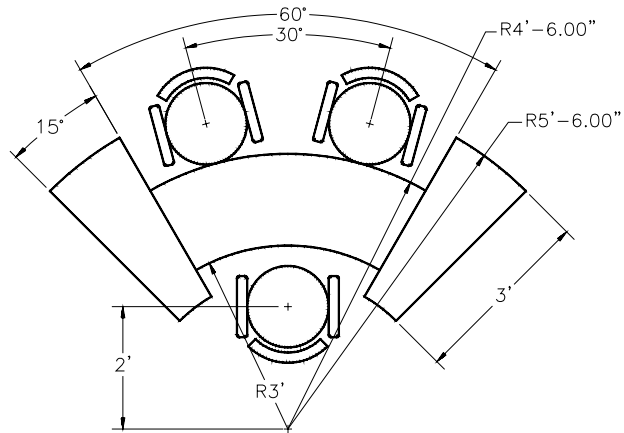


Figure 6-35 Drawing for Problem Solving Exercise 4

Answers to Self-Evaluation Test

1 - T, 2 - T, 3 - T, 4 - F, 5 - definition, 6 - three, two, 7 - GRIPBLOCK, 8 - GRIPCOLOR, 9 - MIRROR, MI, 10 - Value