



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 objects.
- Use the **QSELECT** command to select objects.
- Manage contents using the **DESIGNCENTER**.
- Use the 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 it 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 triangular grips along with the small square grips. These triangular grips resemble arrowheads. These arrowheads point towards the direction, in which the arc can be edited dynamically. 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 crosshairs is displayed (Figure 6-1). The grip location of some of the objects is shown in Figure 6-2.

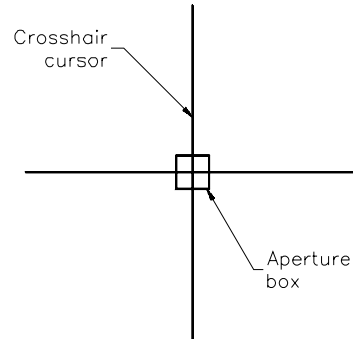


Figure 6-1 Aperture box at the intersection of crosshairs

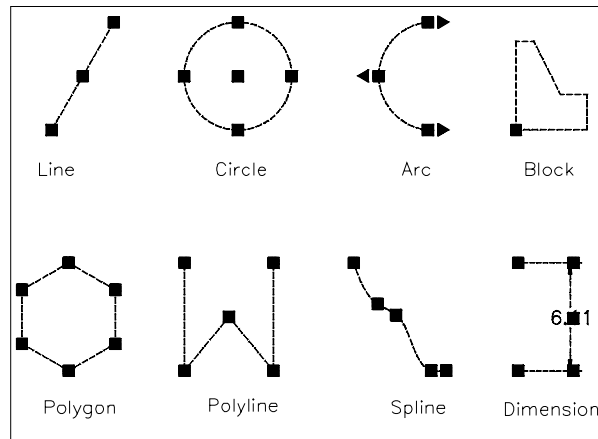


Figure 6-2 Grip location of various objects

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, which is highlighted 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 grip is displayed in green. These grips are called hover grips. Dimensions corresponding to a hover grip are displayed when you place the cursor on the grip, as shown in Figure 6-3. Next, if you select a grip on this object, it becomes a hot grip

(filled red square). 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 an unselected grip. You can also snap to the unselected grip.

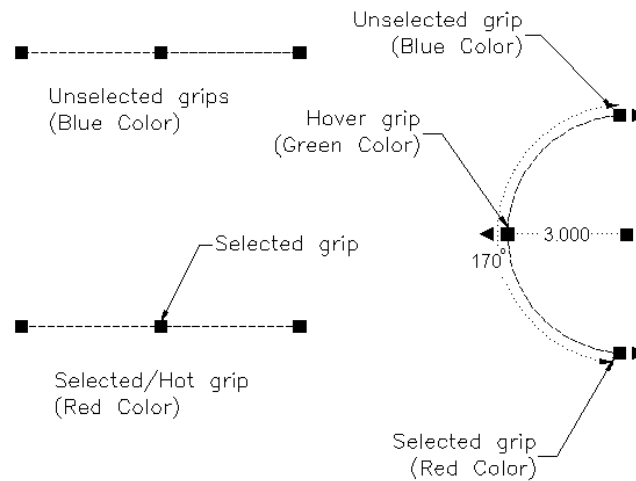


Figure 6-3 Hover grip dimensions being displayed

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 in 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, move the slider box left or right. The size 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.

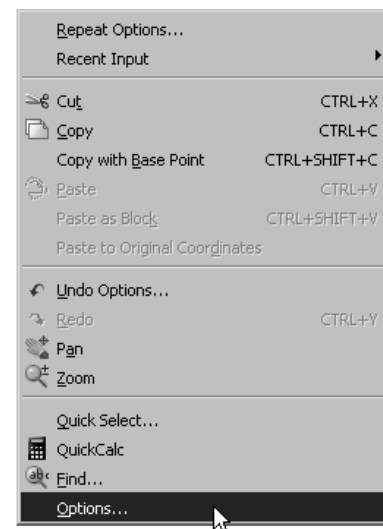


Figure 6-4 Invoking the **Options** dialog box from the shortcut menu

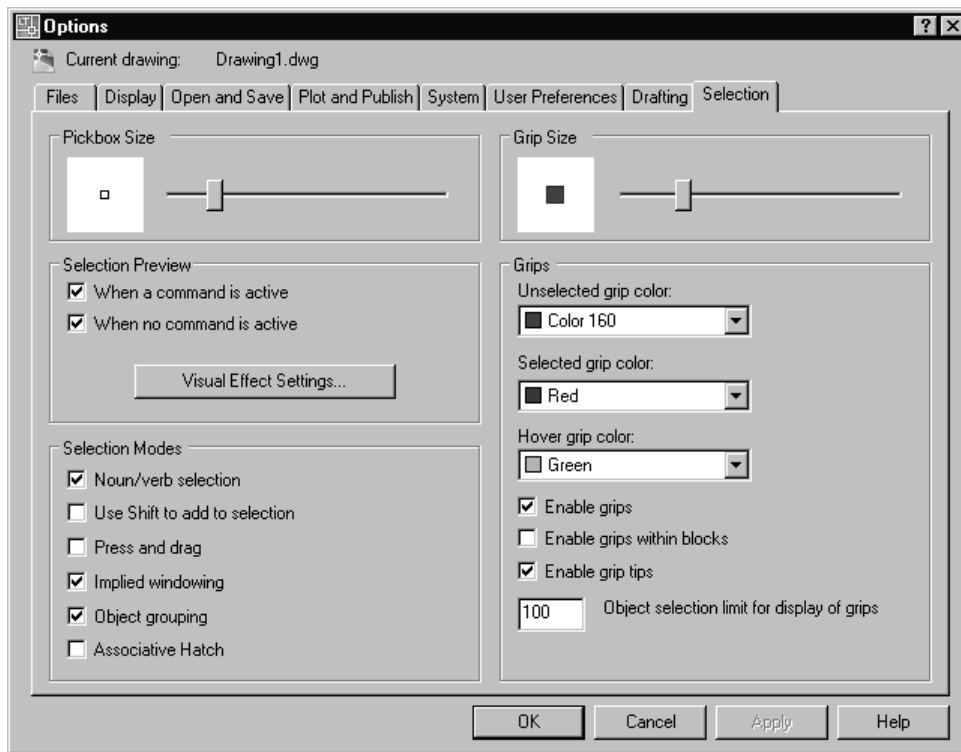


Figure 6-5 The **Selection** tab of the **Options** dialog box

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 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 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 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 LT 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 LT 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 **GRIPTIPS** system variable to 1 (On). If **GRIPTIPS** is set to 0 (Off), AutoCAD LT will display the grip tips for the custom object.

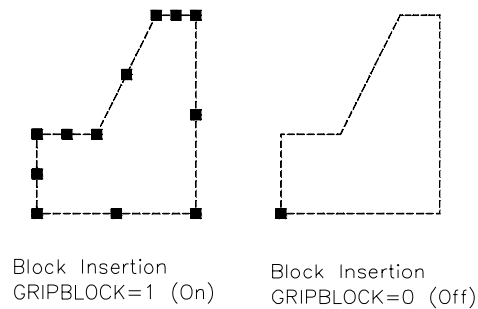


Figure 6-6 Block insertion with **GRIPBLOCK** set to 1 and to 0



Note

If the block has a large number of objects, and if **GRIPBLOCK** is set to 1 (On), AutoCAD LT 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, the 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 **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 the 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 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 LT 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 will change from blue to red.

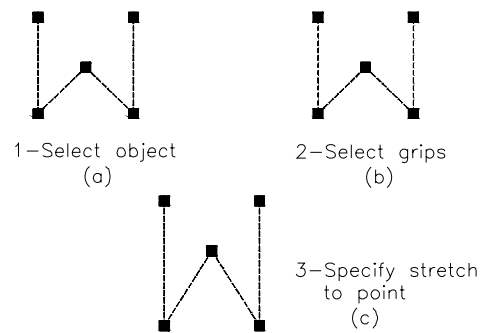


Figure 6-7 Using the **Stretch** mode to stretch the lines



Note

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

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 LT will make a copy. You can also specify stretch points by entering the numerical value in the dimensional input boxes in the drawing area. If you press the **CTRL** key when specifying the point to which the object is to be

stretched, without selecting the copy option, then also AutoCAD LT allows you to make multiple copies of the selected object. Also, if you press the CTRL 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 LT 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).
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.

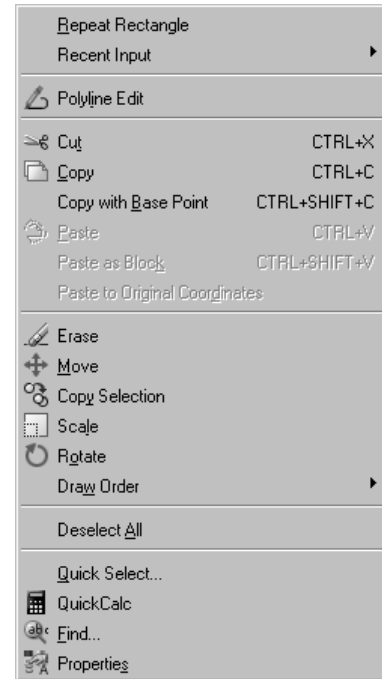


Figure 6-8 Selecting different Grip options from the shortcut menu



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.

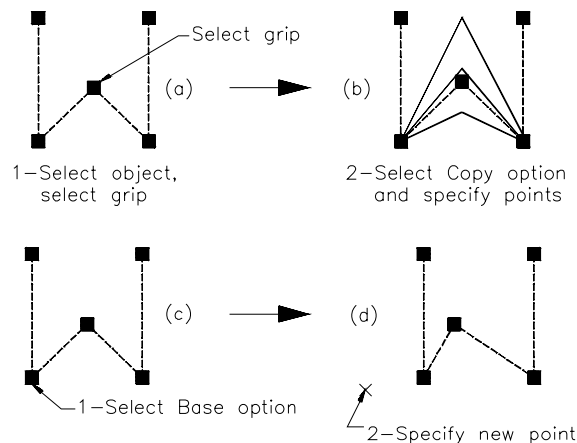


Figure 6-9 Using the Stretch mode's **Copy** and **Base point** options

Moving Objects with Grips (Move Mode)

The **Move** mode lets you move the selected objects to a new location. When you move objects, their size and angles 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.

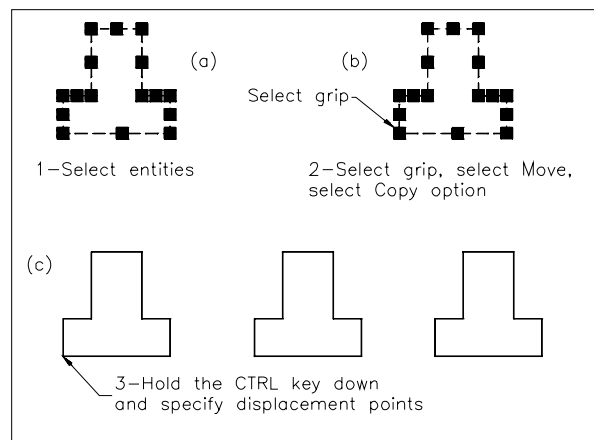


Figure 6-10 Using the **Move** mode to move and make copies of the selected objects

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 giving a null response by pressing the SPACEBAR or ENTER key. AutoCAD LT will display the following prompt in the Command prompt area:

****MOVE****

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

3. Hold down the CTRL 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 CTRL key, move the screen crosshairs to the next snap point and select the point. AutoCAD LT will make a copy of the object at this location. If you release the CTRL 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 the **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 giving a null response twice by pressing the SPACEBAR or the ENTER key. The following example illustrates the use of the **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 LT will display the following prompt:

****ROTATE****

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

3. At this prompt, enter the rotation angle. By default the rotation angle will be entered through the dimensional input below the cursor in the drawing area. AutoCAD LT 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 its lower left corner. Invoke the **Rotate** mode and then select the **Copy** option from the shortcut menu or enter **C** (Copy) at the Command prompt or the **Copy** option from the dynamic preview. Enter the rotation angle. AutoCAD LT will rotate the copy of the object through the specified angle [Figure 6-11(d)].

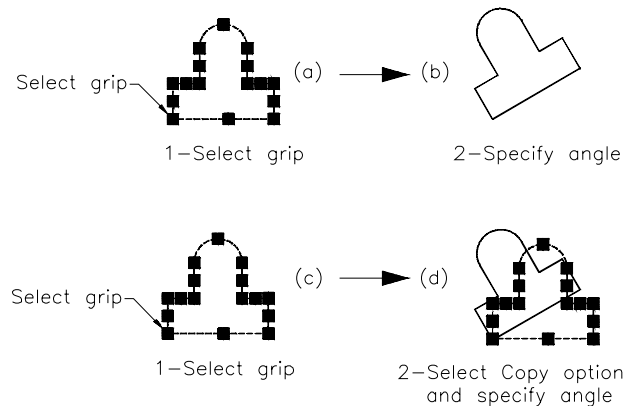


Figure 6-11 Using the **ROTATE** mode to rotate and make copies of the selected objects

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

****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 LT 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.

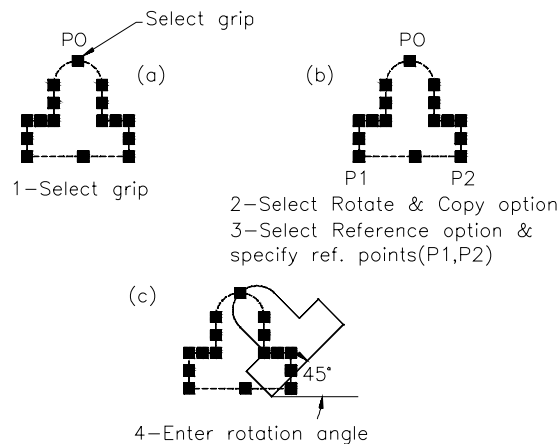


Figure 6-12 Using the **ROTATE** mode to rotate by giving a reference angle

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 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, they will be highlighted and the grips will be displayed at the definition points.
2. Select the grip located at the lower left corner as the base grip, and then invoke the **Scale** mode. AutoCAD LT 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 LT 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 their lower left corner. Invoke the **Scale** mode. At the following prompt, enter C (Copy), and then enter B for the 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 LT will scale the objects so that the length of the bottom edge is equal to the specified value [Figure 6-13(c)].

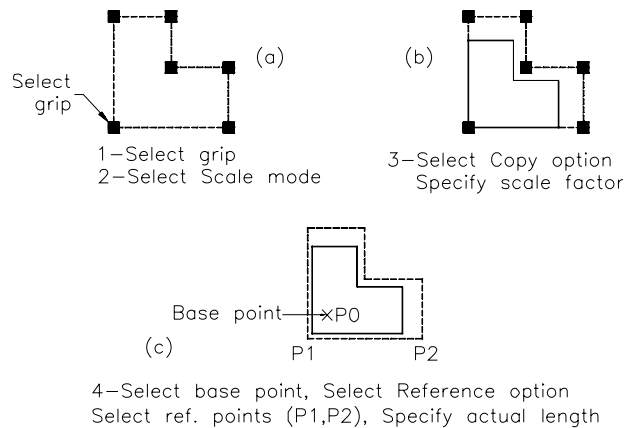


Figure 6-13 Using the **SCALE** mode to scale and make copies of the selected objects

Mirroring Objects with Grips (Mirror Mode)

The **Mirror** mode allows you to mirror the objects across the mirror axis without changing their size. 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 LT 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 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 LT 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 its lower right corner (P1). 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.

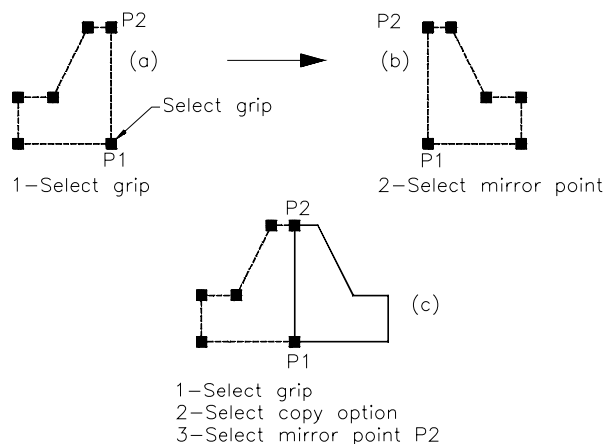


Figure 6-14 Using the **MIRROR** mode to create a mirror image of the selected objects

5. Select point (P2) in response to the prompt **Specify second point or [Base point/Copy/Undo/eXit]**. AutoCAD LT 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.

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 no longer be highlighted.

LOADING HYPERLINKS

If you have already added a hyperlink to the object, you can also use the grips to open a file associated with it. 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. 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, choose the **Hyperlink > Open** option and AutoCAD LT will automatically load the Web browser. When you move the cursor over or near the object that contains a hyperlink, AutoCAD LT 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, see 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, 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, 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 using this toolbar.



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

CHANGING PROPERTIES USING 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 LT 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 a text entity, the related properties 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 the properties of the selected objects, click in the cell next to the name of the property and change the values manually. Alternatively 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 will be explained in detail in Chapter 18.

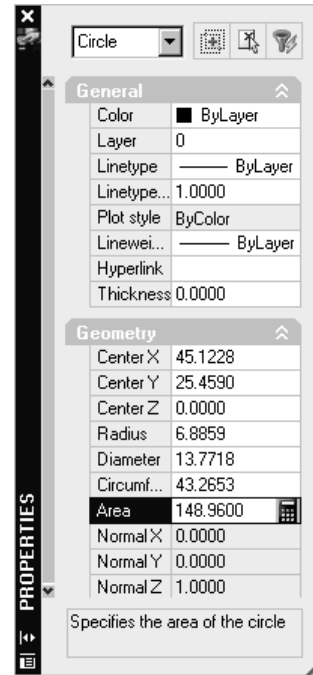


Figure 6-16 The **PROPERTIES** palette for editing the properties of the circle

CHANGING PROPERTIES USING GRIPS

You can also use the grips to change the properties of a single or multiple object. To change the properties, select the object to display the grips and then right-click to display the shortcut menu. Choose the **Properties** option to display the **PROPERTIES** palette. If you select a circle, AutoCAD LT will display **Circle** in the **No selection** drop-down list 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 LT 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 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 LT 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 Table Material Shadow display

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 LT displays the **Property Settings** dialog box (Figure 6-17). The properties displayed are those of the source object. You can use this dialog box to edit the properties that are copied from the source to destination objects.

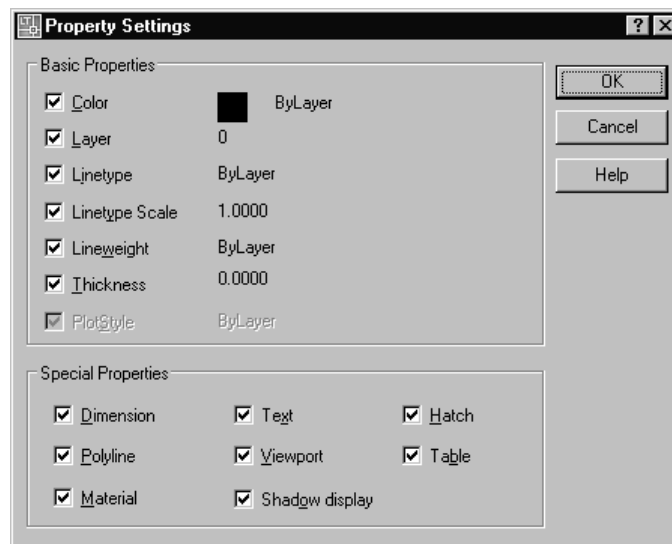


Figure 6-17 The Property Settings dialog box

QUICK SELECTION OF 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 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 it.

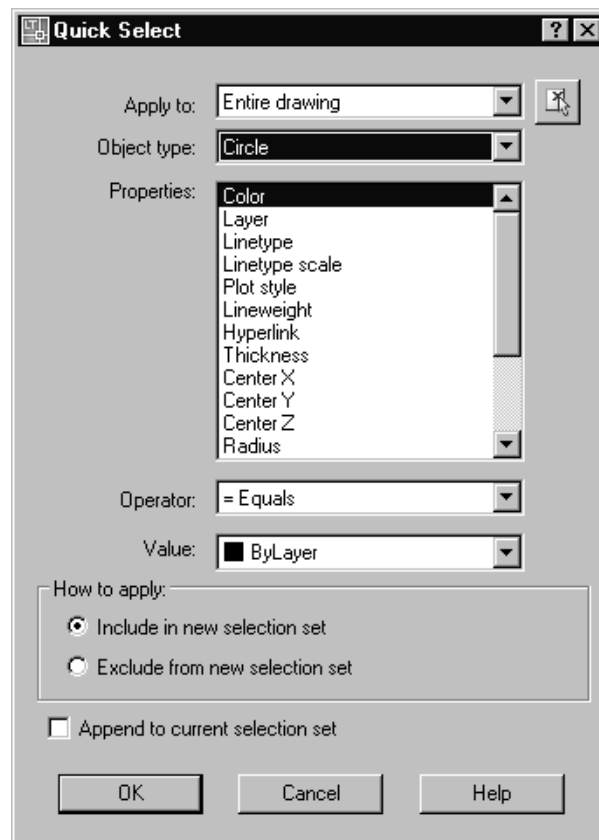


Figure 6-18 The *Quick Select* dialog box

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 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 properties other than AutoCAD LT, 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 the files or from the Internet. You can use the **DESIGNCENTER** to conveniently drag and drop any information that has been previously created into the current drawing. This powerful tool reduces the repetitive tasks of 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.

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 the options to move, resize, close, dock, and hide the **DESIGNCENTER** window. The **Auto-Hide** button 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

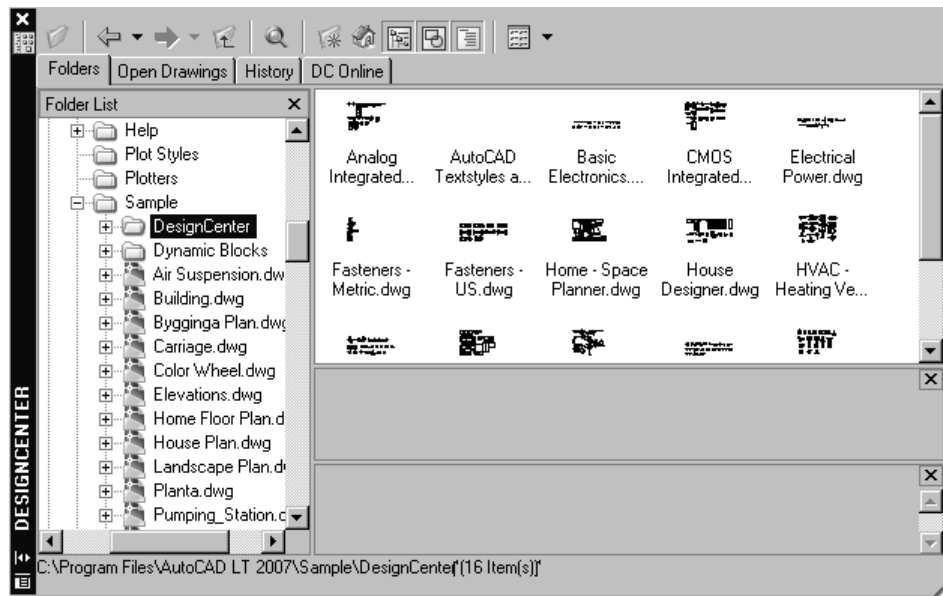


Figure 6-19 The **DESIGNCENTER** window

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.

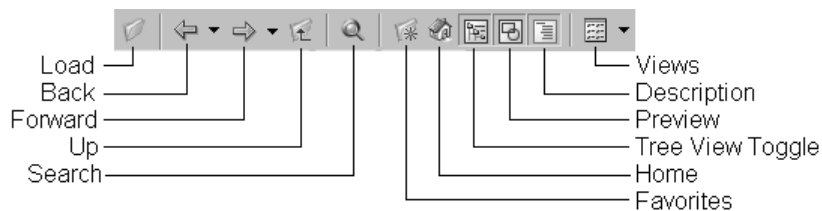


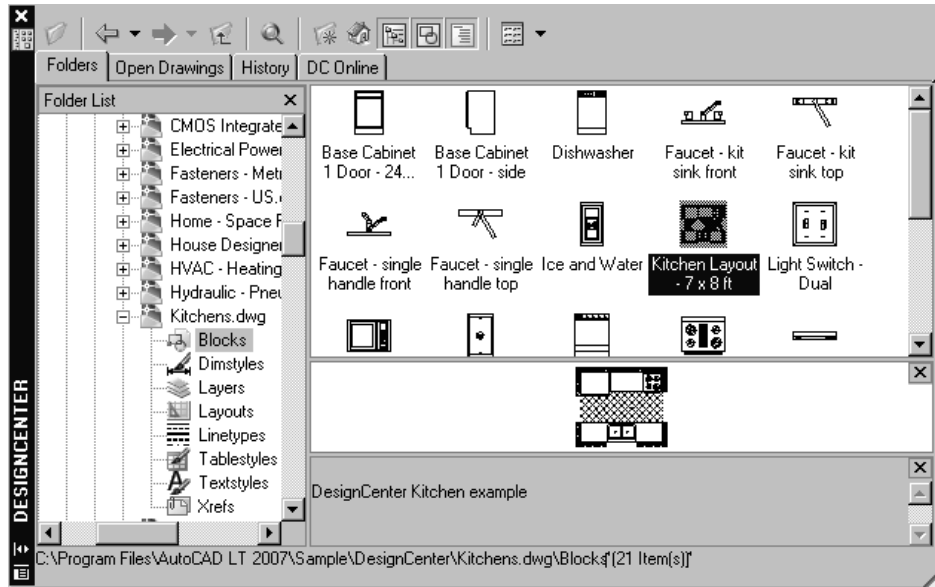
Figure 6-20 The **DESIGNCENTER** toolbar buttons

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 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 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.



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

The **Explore** option in this shortcut menu also further expands the selected folder, file, or category of contents to display the listing of the 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 them. These are then reused as part of the current drawing. When you double-click on specific Xrefs and blocks, AutoCAD LT 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 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 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 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.

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 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 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, 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 LT 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 **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 the contents of the drawing *Kitchens.dwg*. Also, use the **DESIGNCENTER** to insert a block from this drawing and import a layer and a textstyle from the *Hotel Model.dwg* file 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 6-22.

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.

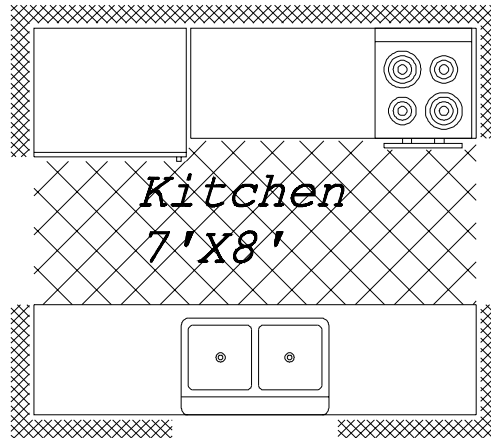


Figure 6-22 Drawing for Example 1

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:** (or the drive in which AutoCAD LT 2007 is installed) 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 expand the tree view and 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 *Hotel Model.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 **7BRIDGE** 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 text styles in the **Palette**. Select **ITALICA** 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 Figure 6-22.
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 it. In manual drafting, you can inquire about the drawing by performing measurements and calculations manually. Similarly, when drawing in an AutoCAD LT environment, you will need to make inquiries about the 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 LT keeps track of all these details. Since inquiry commands are used to obtain information about the selected objects, they do not affect the drawings in any way. The following is the list of Inquiry commands:

AREA	DIST	ID	LIST
TIME	DWGPROPS	MASSPROP	

For most of the Inquiry commands, you are prompted to select objects; once the selection is complete, AutoCAD LT switches from the graphics mode to the text mode, and all the relevant information about the selected objects is displayed. For some commands, information is displayed in the AutoCAD LT 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 LT Text Window by entering **TEXTSCR** at the Command prompt.

Measuring Area of 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 LT, 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 6-23), you need to select all the vertices of that object. By selecting the points, you can 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 the object snaps such as the **END**point, **INT**ersect, and **TAN**gent, or even use running Osnaps, to help you select the vertices quickly and accurately. For AutoCAD LT, 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 LT 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

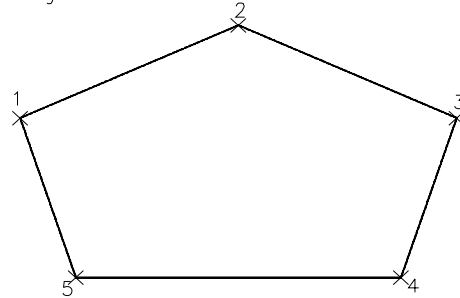



Figure 6-23 Using the **AREA** command

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 LT 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 LT 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), Perimeter = (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 region 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 LT 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 the 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 LT 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 LT 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 LT 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 6-24 is given next.

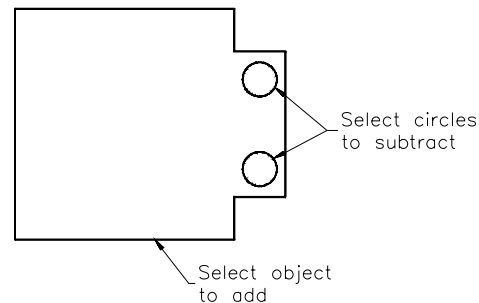


Figure 6-24 Measuring the area of a sketch using the **Add** and **Subtract** options

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 its area and the perimeter. 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, as shown in Figure 6-25. The angles that the selected points make with the X axis and the XY plane are also displayed. The measurements are displayed in the 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.

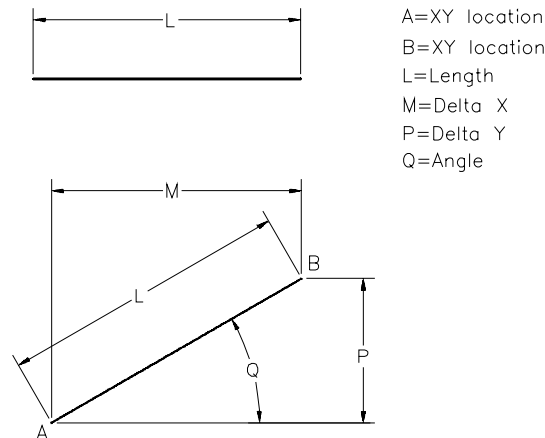


Figure 6-25 Using the **DIST** command

Specify first point: *Specify a point.*

Specify second point: *Specify a point.*

AutoCAD LT 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 LT 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 LT 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 LT 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 LT 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 LT 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 LT 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 LT shifts you from the graphics screen to the AutoCAD LT 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 LT 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 LT pauses to allow you to press ENTER to continue the listing.

Checking Time-Related Information

Menu: Tools > Inquiry > Time
Command: TIME

The time and date maintained by your system are used by AutoCAD LT 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:      Tuesday, September 05, 2006 at 6:59:41:157 PM
Times for this drawing:
Created:           Tuesday, September 05, 2006 at 3:51:19:396 PM
Last updated:      Tuesday, September 05, 2006 at 3:51:19:396 PM
Total editing time: 0 days 03:08:22.522
Elapsed timer (on): 0 days 03:08:21.961
Next automatic save in: 0 days 00:37:11.432
```

Enter option [Display/ON/OFF/Reset]: *Enter the required option.*

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, as shown in Figure 6-26. This dialog box has four tabs under which information about the drawing is displayed. This information helps you look for the drawing more easily. The tabs are as follows.

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 or modified.

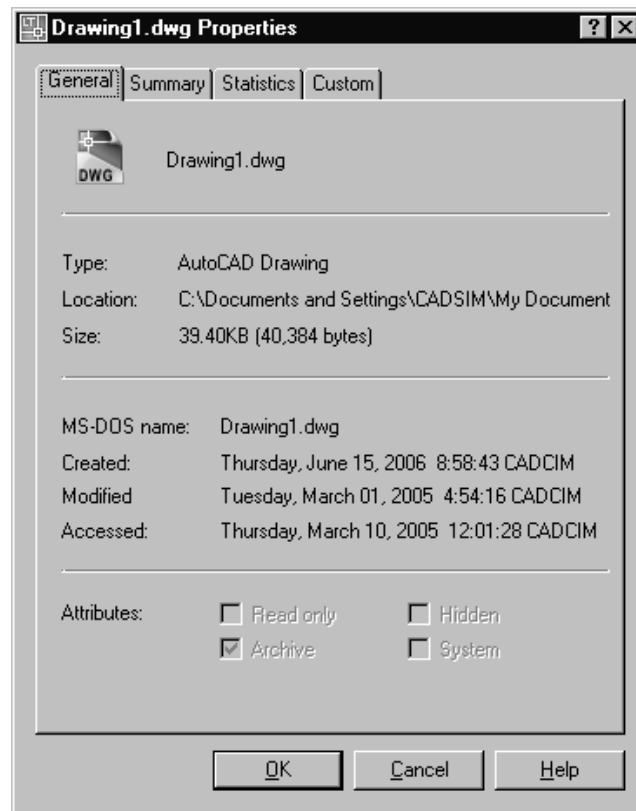


Figure 6-26 The *Drawing Properties* dialog box

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 those 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 a hyperlink to the object, you can also use the grips to open a file associated with it. (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 6-27(a).
2. Use grips (**Stretch** mode) to get the shape, as shown in Figure 6-27(b).
3. Use the **Rotate** and **Stretch** modes to get the copies, as shown in Figure 6-27(c).

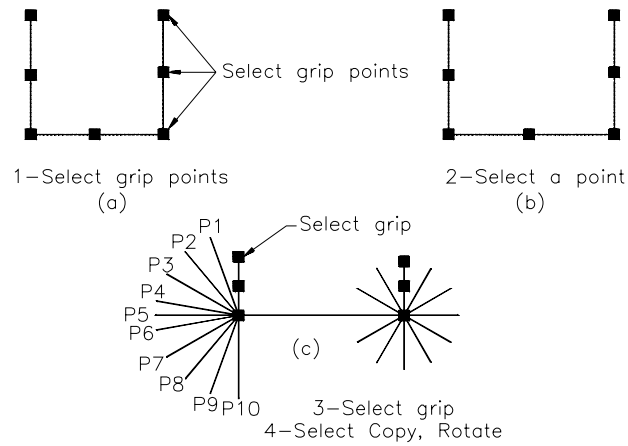


Figure 6-27 Drawing for Exercise 1

Exercise 2

Mechanical

Use the draw and editing commands to create the sketch shown in Figure 6-28.

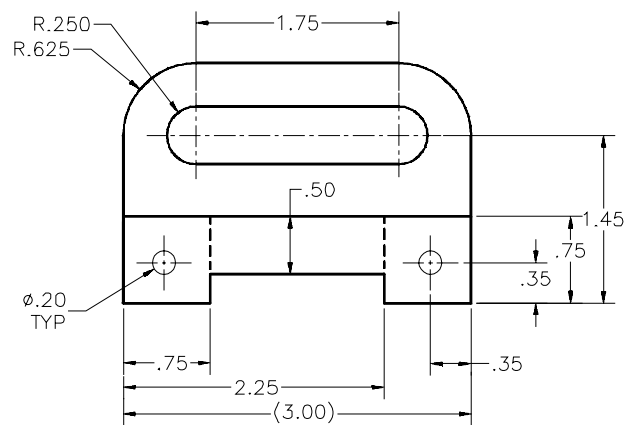


Figure 6-28 Drawing for Exercise 2

Exercise 3

Mechanical

Use the draw and editing commands to create the sketch shown in Figure 6-29.

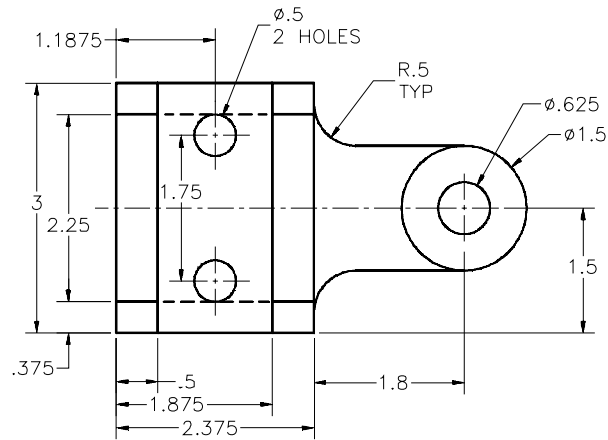


Figure 6-29 Drawing for Exercise 3

Problem-Solving Exercise 1

Mechanical

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

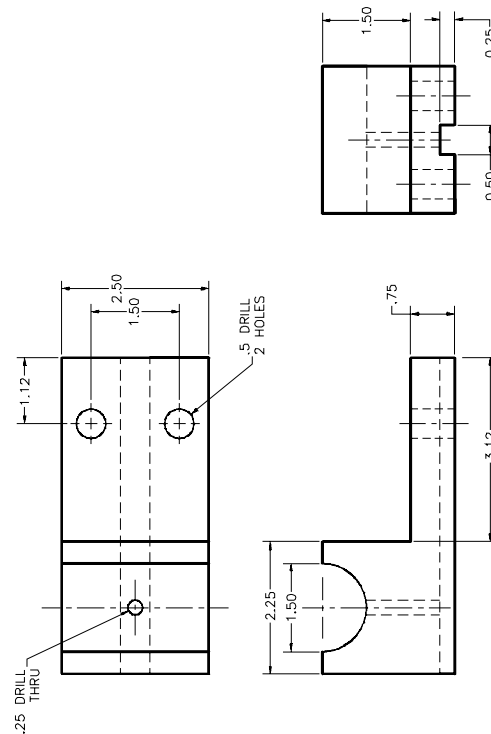


Figure 6-30 Drawing for Problem Solving Exercise 1

Problem-Solving Exercise 2*Mechanical*

Use the draw and editing commands to create the drawing shown in Figure 6-31. Assume the missing dimensions.

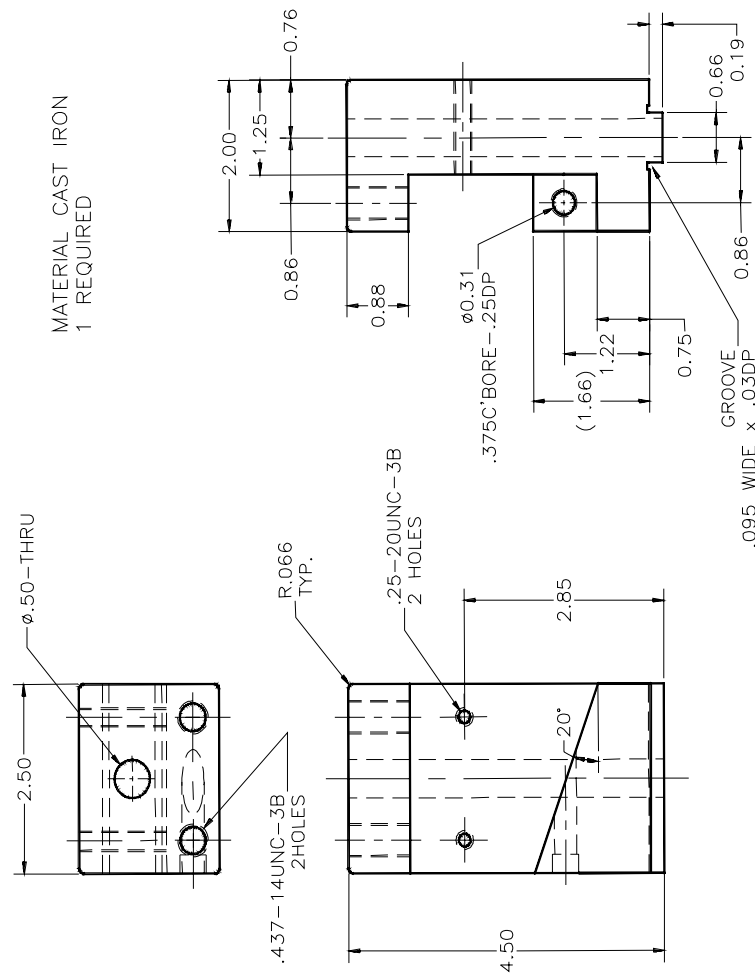


Figure 6-31 Drawing for Problem Solving Exercise 2

Problem-Solving Exercise 3*Mechanical*

Draw the sketch shown in Figure 6-32 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.

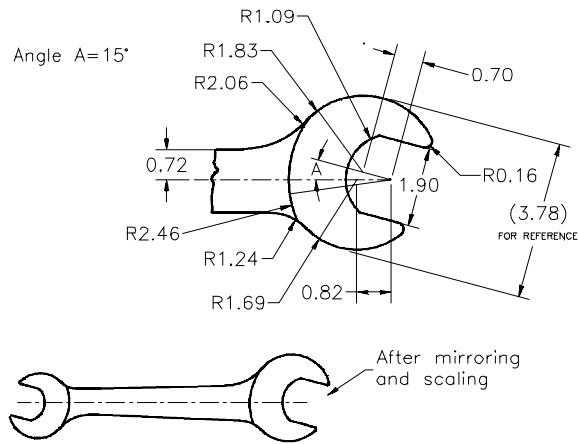


Figure 6-32 Drawing for Problem Solving Exercise 3

Problem-Solving Exercise 4

Architectural

Draw the reception desk shown in Figure 6-33. To get the dimensions of the chairs, refer to Problem Solving Exercise 3 of Chapter 5.

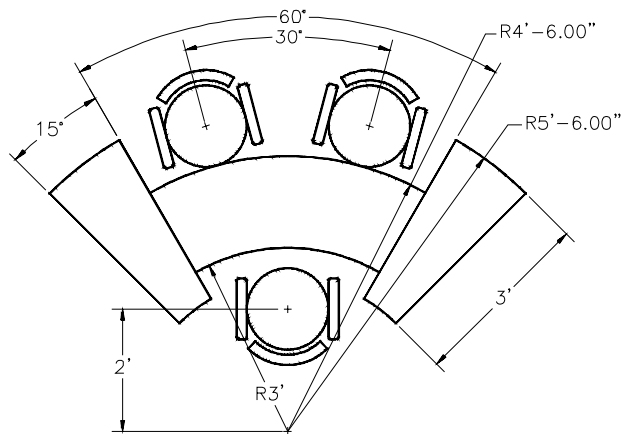


Figure 6-33 Drawing for Problem Solving Exercise 4

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