



Chapter 18

Grouping and Advanced Editing of Sketched Objects

Learning Objectives

After completing this chapter, you will be able to:

- *Group sketched objects.*
- *Select and cycle through defined groups.*
- *Change properties and location of sketched objects.*
- *Perform editing operations on polylines.*
- *Explode compound objects and undo previous commands.*
- *Rename named objects and remove unused named objects.*

WORKING WITH THE GROUP MANAGER

Toolbar: Group > Group Manager
Menu: Tools > Group Manager
Command: GROUP



Group Manager

Figure 18-1 Invoking the **Group Manager** dialog box from the **Group** toolbar

Groups are simply a mechanism that enables you to form groups and edit objects by groups. Creating groups makes the object selection process easier and faster. Objects can be members of several groups. When you choose the **Group Manager** button from the **Group** toolbar (Figure 18-1), the **Group Manager** dialog box is displayed as shown in Figure 18-2. You can use this dialog box to group AutoCAD LT objects and assign a name to the group. Once you have created groups, you can select the objects by group name. The individual characteristics of an object are not affected by forming groups.

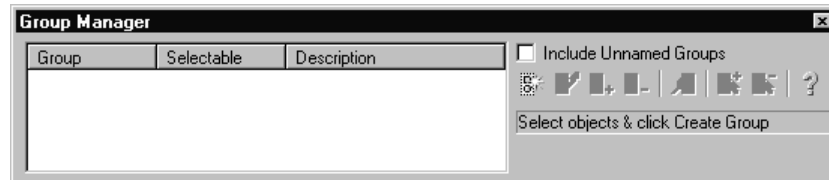


Figure 18-2 The **Group Manager** dialog box

The **Group Manager** dialog box provides the following options.

Group List Box

The **Group** list box in the **Group Manager** dialog box displays the names of the existing groups and whether a group is selectable or not. A selectable group is one in which all members are selected on selecting a single member. If a group is selectable, the bulb icon for that group will be yellow in color. Else, the bulb icon will be gray in color. This list also displays description regarding the group. To add a description about a group, click below the **Description** column. A text box is displayed where you can add the description.



Note

If the value of the **PICKSTYLE** System Variable is set to 1 or 3, the entire group is selected upon selecting any one object from the group. If the value is set to 0, the entire group is not selected upon selecting any one object from the group.

Create Group



Choosing this button allows you to create a new group consisting of the objects you have selected. Note that most of the options in this dialog box will be available only after you have created at least one group. Names assigned to the groups are displayed under the **Group** column. Choose this button and then enter the name of the group. Now, select the objects to be included in this group and choose the **Add to Group** button. You can also select

the objects first and then choose this button. This way you do not have to choose the **Add to Group** button to add the objects in the group.

Ungroup



This button of the **Group Manager** dialog box deletes the selected group from the **Group** list box and also removes the association of the objects in the group. This button is chosen when you want to delete a group.

Add to Group



Choosing this button allows you to add the selected objects to the group already created and selected under **Group** list box. This option is used when you want to add some more objects to the group already created.

Remove from Group



Choosing this button removes the selected objects from the group that is selected in the **Group** list box. You can remove a few selected objects from the group of number of objects by choosing this button.



Note

*Objects can be added or removed from a selection set by the **PICKADD** system variable. If the value of this system variable is set to 1, you can add or remove the objects from the group. If the value is set to 0, the current selection set replaces the previous selection set.*

Details



Choosing this button displays the **Group Manager - Details** dialog box, see Figure 18-3. This dialog box display all the details about the selected group.

Select Group



As you choose this button, all of the objects in the specified group are highlighted. You have to first select the group name from the **Group** list box and then choose the **Select Group** button to highlight the objects in the selected group.

Deselect Group



As you choose this button, the specified group is removed from the selection set. This option is used when more than one groups are selected at the same time. In this case, you can remove a particular group from the selection set by selecting the group from the **Group** list and then choosing this button.

Include Unnamed Groups

The **Include Unnamed Groups** check box is used to display the names of the unnamed groups in the **Group Manager** dialog box. The unnamed groups are created when you copy the objects that belong to a group. AutoCAD LT automatically groups and assigns a name to the copied objects. The format of the name is *An (for example, *A1, *A2, *A3). If you select the **Include**

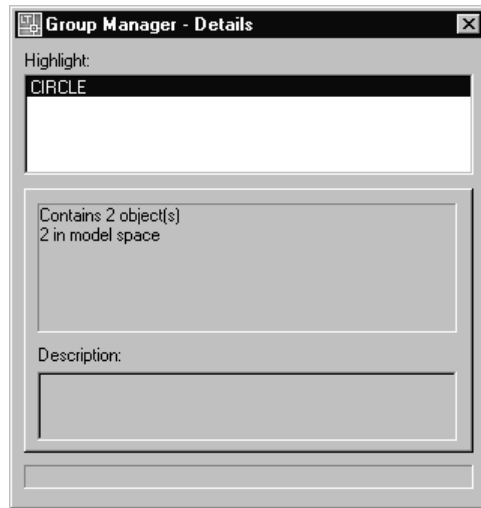


Figure 18-3 The **Group Manager - Details** dialog box

Unnamed Groups check box, the unnamed group names (*A1, *A2, *A3, ...) will also be displayed in the **Group** list box of the dialog box.



Tip

The unnamed groups can also be created by not assigning a name to the group. This is done by pressing the **ENTER** key when you have to specify the name of the group. AutoCAD LT automatically assigns a name to the new group. If the **Include Unnamed Groups** check box is selected, the unnamed group will be displayed under the **Group** list.




Note

You can also use the Command line for creating the groups. This is done using the **-GROUP** command. All the options of the **Group Manager** dialog box are also available when you use the command line.

SELECTING GROUPS

You can select a group by name by entering **G** at the **Select objects** prompt. For example, if you have to move a particular group, choose the **Move** button from the **Modify** toolbar and the prompt sequence will be as follows.

Select objects: **G**
 Enter group name: *Enter the groups name.*
 n found
 Select objects: 

Instead of entering **G** at the **Select Objects:** prompt, if you select any member of a selectable group, all the group members get selected. Make sure that the **PICKSTYLE** system variable is

set to 1 or 3. Also, the group selection can be turned on or off by pressing **SHIFT+CTRL+A**. If the group selection is off and you want to erase a group from a drawing, invoke the **Erase** command from the **Modify** toolbar. The prompt sequence is as follows:

Select objects: **SHIFT+CTRL+A**

<Groups on> *Select an object that belongs to a group. (If the group has been defined as selectable, all objects belonging to that group will be selected.)*

If the group has not been defined as selectable, you cannot select all objects in the group, even if you turn the group selection on by pressing the **SHIFT+CTRL+A** keys. This setting can also be changed in the **Selection** tab of the **Options** dialog box. You can select or clear the **Object Grouping** check box in the **Selection Modes** area of the dialog box to turn group selectability on or off.



Tip

*The combination of **SHIFT**, **CTRL**, and **A** keys is used as a toggle to turn the group selection on or off.*

CYCLING THROUGH GROUPS

When you use the **GROUP** command to form object groups, AutoCAD LT lets you sequentially highlight the groups of which the selected object is a member. For example, assume that an object belongs to two different groups, and you want to highlight the objects in those groups. To accomplish this, press **CTRL** at the **Select objects** prompt of any command and select the object that belongs to different groups. AutoCAD LT will highlight the objects in one of the groups. Press the pick button of your pointing device to cycle through the groups (Figure 18-4). Invoke the **ERASE** command; the prompt sequence is given next.

Select objects: *Hold down CTRL.*

<Cycle on> *Select the object that belongs to different groups. (Press the pick button on your pointing device to cycle through the groups.)*

Select objects:

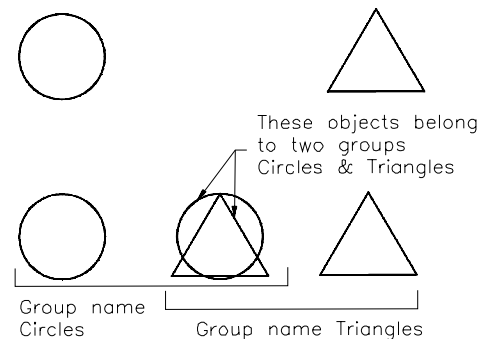


Figure 18-4 Cycling through objects of different groups

Also, if objects are very close or are directly on top of one another, such that they lie within the same selection pickbox, you can press **CTRL** at the **Select objects** prompt and repeatedly click your pointing device until the object you want is highlighted. Press **ENTER** to select it.

**Note**

You can also use the command line form of the **GROUP** command by entering **-GROUP** at the Command prompt.

CHANGING PROPERTIES OF AN OBJECT

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



AutoCAD LT provides you different options for changing the properties of an object. These options are discussed next.

Using the PROPERTIES Palette

The categories displayed in the **PROPERTIES** palette depend on the type of object selected. The **General** category displays the general properties of objects, such as color, layer, linetype, linetype scale, plot style, lineweight, hyperlink, and thickness. How to change the general properties of objects using the **PROPERTIES** palette has already been discussed earlier in Chapter 4, Working with Drawing Aids. Here, the **Geometry** category that contains properties that control the geometry of an object will be discussed. Depending on the type of object selected, the **Geometry** category will contain a set of different properties. If you have selected many types of objects in a drawing, the edit box at the top of the **PROPERTIES** palette displays **All** and only the **General** category is available in the palette. You can select a type of object from the drop-down list and the corresponding categories of the object properties are displayed in the palette.

Selecting Line

If you have selected a line in the drawing, the **PROPERTIES** palette appears similar to what is shown in Figure 18-5, where you can change the properties of the selected object.

Start X/Start Y/Start Z. The **Start X/Start Y/Start Z** fields in the **Geometry** category display the start point coordinates of the selected line. You can enter new values in these fields or use the **Pick Point** button that appears in the field you click in. When you choose the **Pick Point** button, a rubber-band line gets attached to the cursor on its start point and the endpoint of the line is fixed. You can now move the cursor and specify a new start point for the line.

End X/ End Y/ End Z. Similar to the start point, you can also modify the location of the endpoint of a line by entering new coordinate values in the **End X, End Y, End Z** fields. You can also

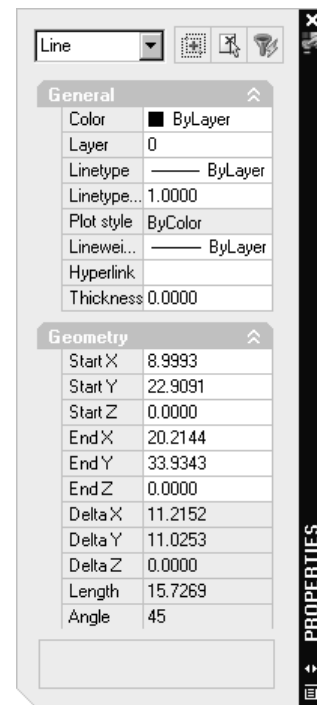


Figure 18-5 The **PROPERTIES** palette

choose the **Pick Point** button in the respective fields and select a new location for the endpoint of the selected line. This **Pick Point** button appears in the fields when you click in them.



Note

The **Delta X**, **Delta Y**, **Delta Z**, **Length**, and **Angle** fields are not available for change but the values in these fields get updated as the start and endpoints of the line are changed.

Selecting Circle

When you select a circle, the **Geometry** category displays the categories as displayed in Figure 18-6. You can modify the values in these fields. The effects of modifying the values are discussed next.

Center X/Center Y/Center Z. Here, under the **Center X/Center Y/Center Z** fields of the **Geometry** category, you can specify a new location of the center point of the circle by entering new coordinate values in the **Center X**, **Center Y**, and **Center Z** fields. When you click in any of these fields, the **Pick Point** button is displayed. You can choose this button to locate a new location of the center point of the circle. The circle will be drawn at the new location with the same radius as before. It is as if the circle has moved from the old location to a new one.

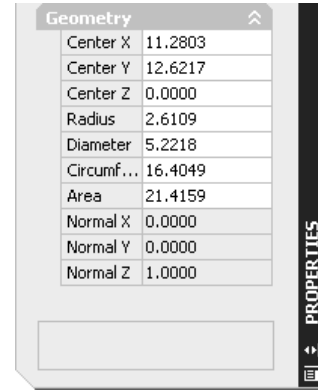


Figure 18-6 The **Geometry** category of the **PROPERTIES** palette for a circle

Radius/Diameter. You can enter a new value for the radius or diameter of the circle in the **Radius** or **Diameter** fields respectively. The circle's radius or diameter is modified as per the new values you have entered. As you modify the values of the radius or the diameter, the values in the **Circumference** and **Area** fields get modified accordingly.



Note

You can also enter new values of the circumference or area in the **Circumference** and **Area** fields, respectively, if you want to create a circle with a given value of circumference or area. You will notice that the radius and diameter values are modified accordingly. The **Normal X**, **Normal Y**, and **Normal Z** fields are not available for modifications.

Selecting Multiline Text

If you select multiline text written in a drawing and invoke the **PROPERTIES** palette, apart from the **General** category, two more categories of properties are displayed: **Text** and **Geometry**. The properties under these two categories (Figure 18-7) and how they can be modified are discussed next.

Contents. The current contents of the text are displayed in the **Contents** field under the **Text** category of the **PROPERTIES** palette. To modify the contents, click in the field and the [...] button is displayed at the right corner of the field. You can then choose the [...] button to display the **In-Place Text Editor** where you can make the necessary modifications using the options in the dialog box. Once you have made the changes, choose **OK** to exit the text editing

and return to the **PROPERTIES** palette. The changes you made are reflected in the value of the **Contents** field of the **PROPERTIES** palette and in the drawing.

Style. When you click in the **Style** field in the **PROPERTIES** palette, a drop-down list of the defined text styles is displayed. By default, only the **Standard** style is displayed. If you had defined more text styles in the drawing, they can be selected from the drop-down list and applied to the current text in the drawing.

Justify. All the text justification options are available in the **Justify** drop-down list. This drop-down list is available when you click in the **Justify** field. By default, the Top left justification is applied to the text. You can select a text justification from the drop-down list to be applied to the selected text in the drawing.

Direction. This drop-down list displays the possible directions for the text. This option is generally available for the multiline text objects.

Width. In the **Width** field you can enter a new value for the paragraph text width. The changes are evident when you have multiline text in the drawing. You can increase or decrease the value of the paragraph text.

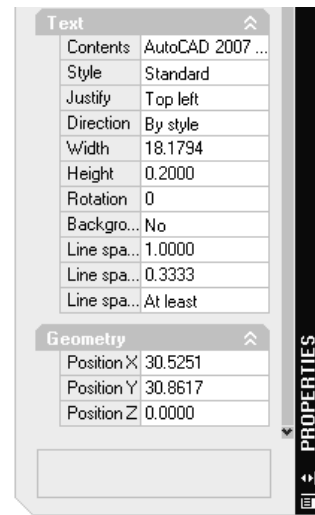
Height. You can enter a new value for the selected text in the drawing. The default text height is 0.2. But, you can make the text smaller or larger in the drawing by simply entering a new value in the **Height** field of the **PROPERTIES** palette.

Rotation. This field displays the current angle of the selected text and is **0-degree**, by default. You can change the angle of rotation of the text by entering an angle by which you want to rotate the selected text in the **Rotation** field and then pressing ENTER. The effect is immediately visible in the drawing.

Background mask. This field specifies whether or not any background mask is assigned to the multiline text.

Line space factor. The effect of changing the value of this field is more evident when you have selected a paragraph or multiline text. This value controls the spacing between the lines in a paragraph or multiline text. A scale factor of 1 is the default value. You can increase or decrease the spacing between the lines of the selected paragraph text by entering a new scale factor of your choice and pressing ENTER. The result is immediately visible in the drawing.

Line space distance. This field is used to modify the distance between the spacing of each line of the multiline text. Click in this field and enter the new value in it.



*Figure 18-7 The Geometry and Text categories of the **PROPERTIES** palette for text*

Line space style. This field, when selected, displays the two line spacing options in the corresponding drop-down list. They are **Exactly** and **At least**. **At least** is the default option.

Position X/Position Y/Position Z. You can change the location of the insertion point of the selected text by entering new X/Y/Z coordinates for the text in these fields. You can also use the **Pick Points** button that is displayed in these fields when you click in them. When you choose the **Pick Points** button, a rubber-band line is attached between the cursor and the selected text and you can move the cursor and specify a new location for the text on the screen. The values of the three fields get updated automatically.

Selecting Block Reference

When you select a block in the drawing and invoke the **PROPERTIES** palette, apart from the **General** category, two more categories containing the properties of the block are available. They are **Geometry** and **Misc**. The properties under these categories are discussed next.

Position X/Position Y/Position Z. Here, under the **Position X/Position Y/Position Z** fields of the **Geometry** category you can enter new X, Y, and Z coordinates for the selected block. The block reference shall move to the new location as you specify the coordinates. You can also click in any one of the fields to display the **Pick Points** button and choose it to specify a new location for the block on the screen.

Scale X/Scale Y/Scale Z. You can specify new X, Y, and Z scale factors for the selected block in the **Scale X**, **Scale Y**, and **Scale Z** fields, respectively. The current scale factors for the block are displayed in the **Scale X**, **Scale Y**, and **Scale Z** fields and as you change them, the modifications are reflected in the drawing.

Name. This field under the **Misc** category displays the name of the selected block reference and is not available for modifications.

Rotation. This field specifies the current rotation angle of the selected block. You can specify a new angle of rotation for the selected block in this field. After specifying an angle, when you press ENTER, the selected block in the drawing is rotated through the specified angle.

Block Unit. This field specifies the UNIT of the block.

Unit Factor. This field specifies the conversion factor between the block unit and the drawing unit.

Selecting Attribute

When you select an attribute before converting it into a block, apart from the **General** category, the other three categories under which the properties of an attribute are displayed are **Text**, **Geometry**, and **Misc**. How to use the **PROPERTIES** palette to edit an attribute has already been discussed in Chapter 15, Defining Block Attributes.

Tag. You can modify the text of the tag by entering a new value in the **Tag** field under the **Text**

category. After you have modified the tag text and pressed ENTER, the old attribute text in the drawing is replaced by the new text.

Prompt. You can also change the text of the prompt and enter a new prompt value in the **Prompt** field.

Value. This field displays the default value of the selected attribute. You can change this default value by entering a new value in the **Value** field.

Style. When you click in this field, the **Style** drop-down list is available. You can select a style for the attribute text from this drop-down list. Only the text styles that already have been defined are available in this drop-down list and can be selected.

Justify. You can select an attribute text justification from the **Justification** drop-down list.

Height. You can change the height of the selected attribute text by entering a new value in this field.

Rotation. You can also rotate the selected attribute text through an angle that you specify in the **Rotation** field.

Width Factor. You can modify the width factor of the attribute text by entering a new value in the **Width Factor** field of the **PROPERTIES** palette.

Obliquing. The attribute text can also be made slanting by entering an obliquing angle in the **Obliquing** field of the **PROPERTIES** palette.

Text alignment X/Text alignment Y/Text alignment Z. The **Text alignment X/Text alignment Y/Text alignment Z** fields of the **Geometry** category display the X, Y, and Z coordinates of the alignment point of the selected attribute. Note that these fields will be available only if you modify the properties such as the text alignment of the attributes, while defining them. If these fields are available and you click on them, the **Pick Point** button appears in the field. You can use this button to specify the new location of the attribute.



Note

*If you modify the text alignment of the first attribute and define the remaining attributes below the previous, the remaining attributes will also have the **Text Alignment** fields active.*

Position X/Position Y/Position Z. You can enter new X, Y, and Z coordinates in the **Position X**, **Position Y**, and **Position Z** fields of the **Geometry** category to relocate the attribute text. You can also choose the **Pick Points** button that is displayed in these fields when you click in them to relocate the selected attribute text.

Misc. As discussed earlier, under the **Misc** category, you can redefine the attribute modes that have been defined at the time of the attribute definition. You can also make the attribute text

appear upside-down or backwards by selecting **Yes** from the **Upside-down** drop-down list and the **Backward** drop-down list, respectively.



Note

Choose **QuickCalc** button from the respective fields to open the Quick Calculator. Using the Quick Calculator you can enter the numerical values as mathematical equations. These equations may include scientific calculations. You can also perform units conversion using the Quick Calculator.

Using the CHANGE Command

Command: CHANGE

With the help of the **CHANGE** command, you can change some of the characteristics associated with an object, such as location, color, layer, lineweight, and linetype. You can also change the geometry of the object using this command. The **CHANGE** command has two options, **Change point** and **Properties**, which are discussed next.

Change Point Option

You can change various features and the location of an object with the **Change point** option of the **CHANGE** command. For example, to change the endpoint of a line or a group of lines, the prompt sequence is given next.

Command: **CHANGE**

Select objects: *Select the line(s).*

Select objects:

Select Change point or [Properties]: *Specify a point to be used as a new endpoint.*



Tip

When the **ORTHO** mode is on, you can use the **CHANGE** command to change the endpoint of a line or group of lines to make them parallel to either the X axis or the Y axis.

To change various features associated with the text, the prompt sequence is given next.

Command: **CHANGE**

Select objects: *Select the text.*

Select objects:

Specify change point or [Properties]:

Specify new text insertion point <no change>: *Specify the new text insertion point (location of text) or press ENTER for no change.*

Enter new text style <current>: *Enter the name of the new text style or press ENTER to accept the current style.*

Specify new height <current>: *Specify the new text height or press ENTER to accept the current value.*

Specify new rotation angle <current>: *Specify the new rotation angle or press ENTER to accept the current value.*

Enter new text <current>: *Enter the new text or press ENTER to accept the current text.*

The properties of an attribute definition text can also be changed just as you change the properties of the text. The prompt sequence for changing the properties of an attribute definition text is as follows.

Command: **CHANGE**

Select objects: *Select the attribute definition text.*

Select objects:

Specify change point or [Properties]:

Specify new text insertion point <no change>: *Specify the new attribute definition text insertion point (location) or press ENTER if you do not want to modify the location of the attribute text.*

Enter new text style <current>: *Enter the name of the new text style or press ENTER to accept the current style.*

Specify new height <current>: *Specify the new attribute definition text height or press ENTER to accept the current value.*

Specify new rotation angle <current>: *Specify the new rotation angle or press ENTER to accept the current value.*

Enter new tag <current>: *Enter a new tag or press ENTER to accept the current value of tag.*

Enter new prompt <current>: *Enter a new prompt or press ENTER to accept the current value.*

Enter new default value <current>: *Enter the new default value or press ENTER to accept the current value.*

You can also change the position of an existing block and specify a new rotation angle to it using the **CHANGE** command. The prompt sequence is as follows.

Command: **CHANGE**

Select objects: *Select the block.*

Select objects:

Specify change point or [Properties]: *Specify the new block insertion point or press ENTER.*

Specify new block insertion point <no change>: *Specify the new block insertion point or press ENTER to accept the current location of the block.*

Specify new block rotation angle <current>: *Specify the new rotation angle for the block or press ENTER to accept the current rotation angle.*

The radius of a circle can also be changed with the **Change point** option of the **CHANGE** command by specifying the new radius in the case of the circle. To change the radius of a circle, the prompt sequence is given next.

Command: **CHANGE**

Select objects: *Select the circle.*

Select objects:

Specify change point or [Properties]: *Select a point to specify the radius of the circle.*

If more than one circle is selected, the same prompts are repeated for the next circle after the first circle is modified.

The Properties Option

The **Properties** option can be used to change the characteristics associated with an object.

Changing the layer of an object. If you want to change the layer on which an object exists and the other characteristics associated with layers, you can use the **LAyer** option of the **Properties** option. The prompt sequence is given next.

Command: **CHANGE**

Select objects: *Select the object whose layer you want to change.*

Select objects: *If you have finished selection, press ENTER.*

Specify change point or [Properties]: **P**

Enter property to change [Color/Elev/LAyer/LType/LtScale/LWeight/Thickness/PLotstyle]:

LA

Enter new layer name <0>: *Enter a new layer name.*

Enter property to change [Color/Elev/LAyer/LType/LtScale/LWeight/Thickness]: 

Changing the color of an object. Similarly, you can change the color of the selected object with the **Color** option of the **CHANGE** command. To change the color and linetype to match the layer, you can use the **Properties** option of the **CHANGE** command and then change the color or linetype by entering **BYLAYER** at the **New color <Current>** prompt. The command prompt sequence for this option is similar to the one displayed previously.



Note

A block, originally created on layer 0, assumes the color of the new layer. Otherwise (if it was not created on layer 0), it will retain the color of the layer on which it was created.

Changing the thickness of an object. You can also change the thickness of the selected object in a similar manner. After you invoke the **CHANGE** command and select the object to change, use the **Properties** option and enter **T** at the **Enter property to change** prompt. You are then prompted to enter the new thickness you want to assign to the selected object.

Changing the lineweight of an object. You can similarly also change the lineweight of the selected object by using the **LWeight** option of the **CHANGE** command. The lineweight values are predefined and if you enter a value that is not predefined, a predefined value of lineweight, which comes closest to the specified value, is assigned to the selected object.



Tip

*You can see the effect of changing the thickness of an object in a 3D view. Choose **SE Isometric** from the **View > 3D Views** menu to view the changed thickness value of an object. To get back to the original view, simply choose the **Undo** button in the **Standard** toolbar.*

Changing the linetype/Linetype scale of an object. Similarly, you can change the linetype and linetype scale of the selected objects by entering the appropriate option at the **Enter property to change** prompt.

Changing the elevation of an object. You can also change the Z axis elevation of a selected object by entering E at the **Enter property to change** prompt, provided all the points in the particular object have the same Z value. After changing the elevation value of an object, you can view the changes in the 3D view.

Changing the plotstyle of an object (for named plot style drawings). You can also change the plotstyle of an object using this command. Enter **PL** at the [Color/Elev/Layer/LType/LtScale/LWeight/Thickness/Plotstyle] prompt and enter the new plotstyle.

Using the CHPROP Command

Command: CHPROP

You can also use the **CHPROP** command to change the properties of an object using the command line. The prompt sequence is as follows:

Command: **CHPROP**

Select objects: *Select the object whose properties you want to change.*

Select objects:

Enter property to change [Color/Layer/LType/LtScale/LWeight/Thickness]: *Specify the property to be modified.*



Note

*Apart from the commands discussed, you can also use the **Properties** toolbar to change some of the properties of the selected objects such as layer, color, linetype, linewidth, and plot style. These options have been discussed in Chapter 4, Working with Drawing Aids.*

Exercise 1

General

Draw a hexagon on layer OBJ in red color. Let the linetype be hidden. Now, use the **PROPERTIES** command to change the layer to some other existing layer, the color to yellow, and the linetype to continuous. Also, in the **PROPERTIES** palette, under the **Geometry** category, specify a vertex in the **Vertex** field or select one using the arrow (Next or Previous) buttons and then relocate it. The values of the **Vertex X** and **Vertex Y** fields change as the coordinate values of the vertices change. Also, notice the change in the value in the **Area** field. You can also assign a start and end linewidth to each of the hexagon segments between the specified vertices. Use the **LIST** command to verify that the changes have taken place.

EXPLODING COMPOUND OBJECTS

Toolbar: Modify > Explode
Menu: Modify > Explode
Command: EXPLODE



The **EXPLODE** command is used to split compound objects such as blocks, polylines, regions, polyface meshes, polygon meshes, multilines, 3D solids, 3D meshes, bodies, or dimensions into the basic objects that make them up (Figure 18-8). For example, if you explode a polyline or a 3D polyline, the result will be ordinary lines or arcs (tangent specification and width are not considered). When a 3D polygon mesh is exploded, the result is 3D faces, points, and lines. Upon exploding 3D solids, the planar surfaces of the 3D solid turn into regions, and nonplanar surfaces turn into bodies. Multilines are changed to lines. Regions turn into lines, ellipses, splines, or arcs. 2D polylines lose their width and tangent specifications and 3D polylines explode into lines. When a body is exploded, it changes into single-surface bodies, curves, or regions. When a leader is exploded, the components are lines, splines, solids, block inserts, text or multiline text, tolerance objects, and so on. Multiline text explodes into single line text. This command is especially useful when you have inserted an entire drawing and you need to alter a small detail. After you invoke the **EXPLODE** command, you are prompted to select the objects you want to explode. After selecting the objects, press ENTER or right-click to explode the selected objects and then end the command.

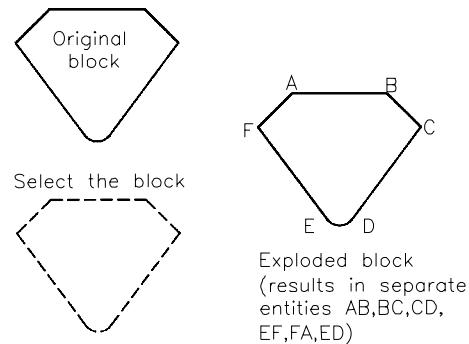


Figure 18-8 Using the **EXPLODE** command

When a block or dimension is exploded, there is no visible change in the drawing. The drawing remains the same except that the color and linetype may have changed because of floating layers, colors, or linetypes. The exploded block is turned into a group of objects that can be modified separately. To check whether the explosion of the block has taken place, select any object that was a part of the block. If the block has been exploded, only that particular object will be highlighted. With the **EXPLODE** command, only one nesting level is exploded at a time. Hence, if there is a nested block or a polyline in a block and you explode it, the inner block or the polyline will not be exploded. Attribute values are deleted when a block is exploded, and the attribute definitions are redisplayed.



Note

Remember that the drawings inserted using the **XREF** command cannot be exploded.

**Tip**

If you want to insert a block in the form of its separate components, while using the **INSERT** command, select the **Explode** check box in the **Insert** dialog box. Also, if you are using the **-INSERT** command, type * in front of the block name at the **Enter block name or [?]:** prompt, the block will be inserted in the drawing as separate component objects and not as an entire block.

EDITING POLYLINES

A polyline can assume various characteristics such as width, linetype, joined polyline, and closed polyline. You can edit polylines, polygons, or rectangles to attain the desired characteristics using the **PEDIT** command. In this section, we will be discussing how to edit simple 2D polylines. The following are the editing operations that can be performed on an existing polyline using the **PEDIT** command. They are discussed in detail later in this chapter.

1. A polyline of varying widths can be converted to a polyline of uniform width.
2. An open polyline can be closed and a closed one can be opened.
3. You can remove bends and curved segments between two vertices to make a straight polyline.
4. A polyline can be split up into two and individual polylines or polyarcs connected to one another can be joined into a single polyline.
5. If you select an entity that is not a polyline to edit using the **PEDIT** command, you are prompted to specify whether you want the entity to be converted into a polyline. You can avoid this prompt by setting the value of the **PEDITACCEPT** variable to 1.
6. Multiple polylines can be edited.
7. The appearance of a polyline can be changed by moving and adding vertices.
8. Curves of arcs and B-spline curves can be fitted to all vertices in a polyline, with the specification of the tangent of each vertex being optional.
9. The linetype generation at the vertices of a polyline can be controlled.
10. Multiple polylines can be joined together to form a single polyline.

Editing a Single Polyline

| | |
|-----------------|----------------------------|
| Toolbar: | Modify II > Edit Polyline |
| Menu: | Modify > Object > Polyline |
| Command: | PEDIT |



Apart from the methods displayed in the **PEDIT** command box, as shown, you can also invoke the **PEDIT** command by choosing **Polyline Edit** from the shortcut menu that is displayed when you select a polyline and right-click. You can use the **PEDIT** command

to edit any type of polyline. When you invoke this command, the following prompt sequence is displayed.

Select polyline or [Multiple]:

If the selected entity is not a polyline, the message that is displayed is given next.

Object selected is not a polyline.
Do you want to turn it into one? <Y>:

If you want to turn the object into a polyline, respond by entering Y and pressing ENTER or by simply pressing ENTER. To let the object be as it is, enter N. AutoCAD LT will then prompt you to select another polyline or object to edit. As mentioned earlier, you can avoid this prompt by setting the value of the **PEDITACCEPT** variable to **1**. The subsequent prompts and editing options depend on the type of polyline that has been selected. AutoCAD LT provides you the option of either selecting a single polyline or multiple polylines. In this case, a single 2D polyline is selected, the next prompt displayed is given next.

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: *Enter an option or press ENTER to end command.*



Note

*Depending on the type of polyline selected, the **PEDIT** command options change. In this chapter, the 2D polyline options have been discussed. The 3D polylines and 3D Polygon Mesh editing options will be discussed in Chapter 23, Getting Started with 3D.*

The options available in **Single** polyline selection method are discussed next:

Close (C) Option

This option is used to close an open polyline. **Close** creates the segment that connects the last segment of the polyline to the first. You will get this option only if the polyline is not closed. Figure 18-9 illustrates this option.

Open (O) Option

If the selected polyline is closed, the **Close** option is replaced by the **Open** option. Entering **O**, for open, removes the closing segment, see Figure 18-9.

Join (J) Option

This option appends lines, polylines, or arcs whose endpoints meet a selected polyline at any of its endpoints and adds (joins) them to it (Figure 18-10). This option can be used only if a polyline is open. After this option has

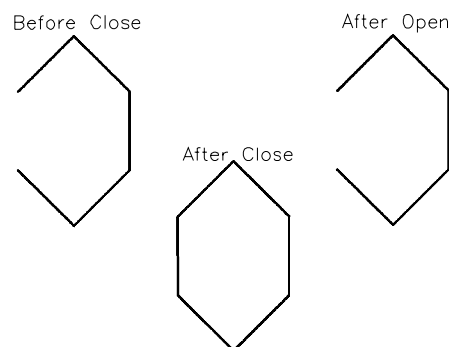


Figure 18-9 The **Close** and the **Open** options

been selected, AutoCAD LT asks you to select objects. Once you have chosen the objects to be joined to the original polyline, AutoCAD LT examines them to determine whether any of them has an endpoint in common with the current polyline, and joins such an object with the original polyline. The search is then repeated using new endpoints. They will not join if the endpoint of the object does not exactly meet the polyline. The line touching a polyline at its endpoint to form a T will not be joined. If two lines meet a polyline in a Y shape, only one of them will be selected, and this selection is unpredictable. To verify which lines have been added to the polyline, use the **LIST** command or select a part of the object. All the segments that are joined to the polyline will be highlighted.

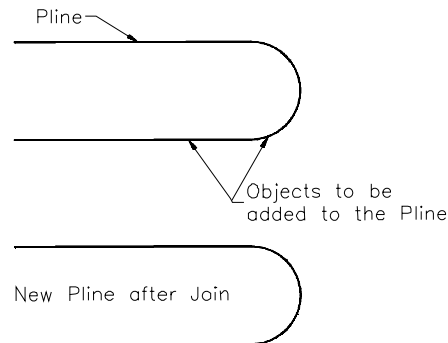


Figure 18-10 Using the **Join** option

Width (W) Option

The **W** option allows you to define a new, unvarying width for all segments of a polyline (Figure 18-11). It changes the width of a polyline with a constant or a varying width. The desired new width can be specified either by entering the width at the keyboard or by specifying the width as the distance between the two specified points. Once the width has been specified, the polyline assumes it. Here, you will change the width of the given polyline in the figure from 0.02 to 0.05 (Figure 18-12). After you invoke the **PEDIT** command, the next prompts are as follows.

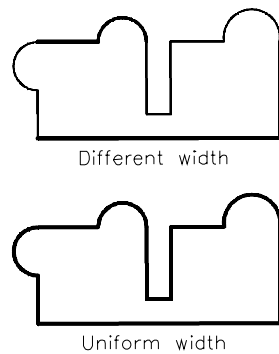


Figure 18-11 Making the width of a polyline uniform

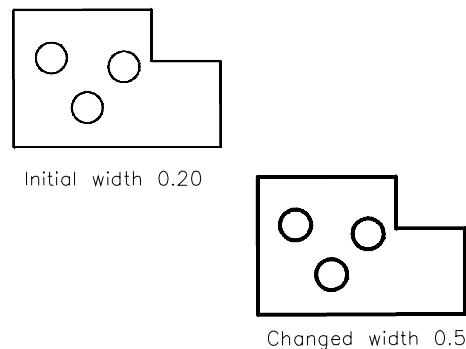


Figure 18-12 Entering a new width for all segments

Select polyline or [Multiple]: *Select the polyline.*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **W**

Specify new width for all segments: **0.05**

**Note**

Circles drawn using the **CIRCLE** command cannot be changed to polylines. However, polycircles can be drawn using the **Pline Arc** option (by drawing two semicircular polyarcs) or by using the **DONUT** command (you can modify the thickness using the **PEDIT Width** option).

Edit vertex (E) Option

The **Edit vertex** option lets you select a vertex of a polyline and perform different editing operations on the vertex and the segments following it. A polyline segment has two vertices. The first one is at the start point of the polyline segment; the other one is at the endpoint of the segment. When you invoke this option, an X marker appears on the screen at the first vertex of the selected polyline. If a tangent direction has been specified for this particular vertex, an arrow is generated in that direction. After this option has been selected, the next prompt appears with a list of options for this prompt. The prompt sequence, after you invoke the **PEDIT** command, is given next.

Select polyline or [Multiple]: *Select the polyline to be edited.*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Enter an editing option or press ENTER to accept default.*

All the options for the **Edit vertex** option are discussed next.

Next and Previous options. These options move the X marker to the next or the previous vertex of a polyline. The default value in the **Edit vertex** option is one of these two options. The option that is selected as default is the one you chose last. In this manner, the **Next** and **Previous** options help you to move the X marker to any vertex of the polyline by selecting one of these two options, and then pressing ENTER to reach the desired vertex. These options cycle back and forth between first and last vertices, but cannot move past the first or last vertices, even if the polyline is closed (Figure 18-13).

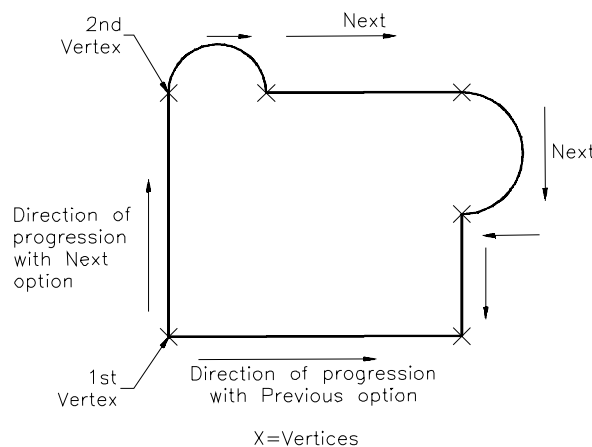


Figure 18-13 The **Next** and **Previous** options

The prompt sequence for using this option, after you invoke the **PEDIT** command, is given next.

Select polyline or [Multiple]: *Select the polyline to be edited.*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Enter N or P to move to the next or previous vertices respectively.*

Break option. With the **Break** option, you can remove a portion of the polyline, as shown in Figure 18-14 or break it at a single point. The breaking of the polyline can be specified between any two vertices. By specifying two different vertices, all the polyline segments and vertices between the specified vertices are erased. If one of the selected vertices is at the endpoint of the polyline, the **Break** option will erase all the segments between the first vertex and the endpoint of the polyline. The exception to this is that AutoCAD LT does not erase the entire polyline, if you specify the first vertex at the start point (first vertex) of the polyline and the second vertex at its endpoint (last vertex). If both vertices are at the endpoint of the polyline, or only one vertex is specified and its location is at the endpoint of the polyline, no change is made to the polyline. The last two selections of vertices are treated as invalid by AutoCAD LT, which acknowledges this by displaying the message ***Invalid***.

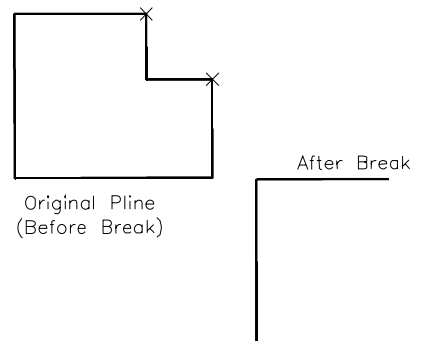


Figure 18-14 Using the **Break** option

To use the **Break** option, first you need to move the marker to the first vertex where you want the split to start. The placement of the marker can be achieved with the help of the **Next** and **Previous** options. Once you have selected the first vertex to be used in the **Break** operation, invoke the **Break** option by entering **B** at the Command prompt. AutoCAD LT takes the vertex where the marker (X) is placed as the first point of the breakup. The next prompt asks you to specify the position of the next vertex for the breakup. You can enter **GO** at this prompt if you want to split the polyline at one vertex only. Otherwise, use the **Next** or **Previous** option to specify the position of the next vertex and then enter **GO**. The polyline segments between the two selected vertices is erased. The prompt sequence after you have invoked the **PEDIT** command is given next.

Select polyline or [Multiple]: *Select the polyline to be edited.*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Enter N or P to locate the first vertex for the Break option.*

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: **B**

Once you invoke the **Break** option, AutoCAD LT treats the vertex where the marker (X) is displayed as the first point for splitting the polyline. The next prompt issued is given next.

Enter an option [Next/Previous/Go/eXit] <N>: *Enter G if you want to split the polyline at one vertex only or move the X marker using the **Next** or **Previous** option to specify the position of the next vertex for breakup.*

After you have specified the next position of the X marker using the **Next** and **Previous** options, entering **Go** deletes the polyline segment between the two markers specified. Now, exit the **Enter a vertex editing option** prompt using the **eXit** option.

Insert option. The **Insert** option allows you to define a new vertex and add it to the polyline (Figure 18-15). You can invoke this option by entering I for Insert. You should invoke this option only after you have moved the marker (X) to the vertex after which the new vertex is to be added. The new vertex is inserted immediately after the vertex with the X mark. After you have invoked the **PEDIT** command, the prompt sequence for using the **Insert** option is as follows.

Select polyline or [Multiple]: *Select the polyline to be edited.*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Move the marker to the vertex after which the new vertex is to be inserted.*

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: **I**

Specify location for new vertex: *Move the cursor and select to specify the location of the new vertex or enter the coordinates of the new location.*

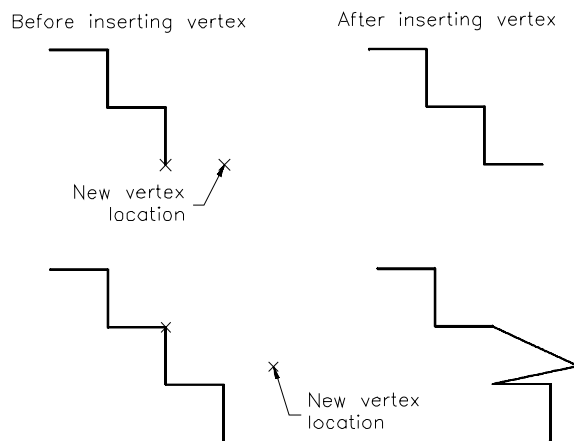


Figure 18-15 Using the **Insert** option to define new vertex points

Move option. This option lets you move the X-marked vertex to a new position (Figure 18-16). Before invoking the **Move** option, you must move the X marker to the vertex you want to relocate

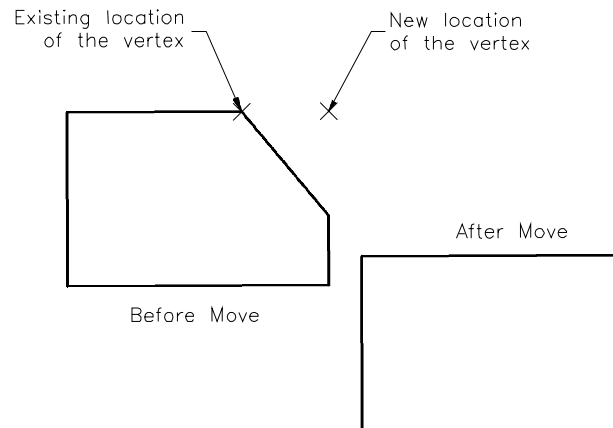


Figure 18-16 The **Move** option

by selecting the **Next** or **Previous** option. The prompt sequence for relocating a vertex after you invoke the **PEDIT** command is given next.

Select polyline or [Multiple]: *Select the polyline to be edited.*
 Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**
 Enter a vertex editing option
 [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Enter N or P to move the X marker to the vertex you want to relocate.*
 Enter a vertex editing option
 [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: **M**
 Specify new location for marked vertex: *Specify the new location for the selected vertex by selecting a new location using the pick button of your pointing device or by entering its coordinate values.*
 Enter a vertex editing option
 [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Enter an option or enter X to exit.*

Regen option. The **Regen** option regenerates the polyline to display the effects of edits you have made, without having to exit the vertex mode editing. It is used most often with the **Width** option.

Straighten option. The **Straighten** option can be used to straighten polyline segments or arcs between specified vertices (Figure 18-17). It deletes the arcs, line segments, or vertices between the two specified vertices and substitutes them with one polyline segment.

The prompt sequence to use this option after you have invoked the **PEDIT** command is given next.

Select polyline or [Multiple]: *Select the polyline to be edited.*
 Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**
 Enter a vertex editing option
 [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Move the*

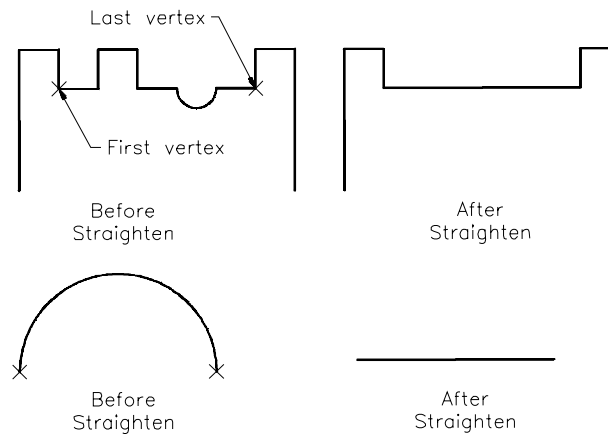


Figure 18-17 Using the **Straighten** option to straighten polylines

marker to the desired vertex from where you want to start applying the **Straighten** option with the **Next** or **Previous** option.

Enter a vertex editing option

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: **S**

Enter an option [Next/Previous/Go/eXit] <N>: Move the marker to the next desired vertex, until you reach the vertex you want to straighten.

Enter an option [Next/Previous/Go/eXit] <N>: **G**

The polyline segments between the two marker locations are replaced by a single straight line segment. If you specify a single vertex, the segment following the specified vertex is straightened, if it is curved.

Tangent option. The **Tangent** option is used to associate a tangent direction to the current vertex (marked by X). The tangent direction is used in the curve fitting or the **Fit** option of the **PEDIT** command. This option is discussed in detail in the subsequent section on curve fitting. The prompt issued on using the **Tangent** option is given next.

Specify direction of vertex tangent: *Specify a point or enter an angle.*

You can specify the direction of the vertex tangent by entering an angle at the previous prompt or by selecting a point to express the direction with respect to the current vertex. You can then move the marker to another vertex using the **Next** or **Previous** option and change its direction of tangent or enter X to exit the **Enter a vertex editing option** prompt.

Width option. The **Width** option lets you change the starting and the ending widths of a polyline segment that follows the current vertex (Figure 18-18). By default, the ending width is equal to the starting width and hence, you can get a polyline segment of uniform width by pressing ENTER at the **Specify ending width for next segment <starting width>** prompt. You

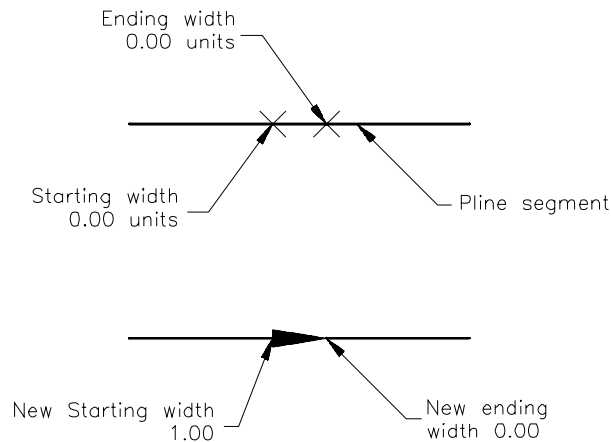


Figure 18-18 Using the **Width** option to change the width of a polyline

can also specify different starting and ending widths to get a varying-width polyline. The prompt sequence is given next.

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: E
 Enter a vertex editing option
 [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: *Move the marker to the starting vertex of the segment whose width is to be altered, using the **Next** or **Previous** option.*
 Enter a vertex editing option
 [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit] <N>: W
 Specify starting width for next segment <current>: *Enter the revised starting width.*
 Specify ending width for next segment <starting width>: *Enter the revised ending width or press ENTER to accept the default option of keeping the ending width of the segment equal to the starting width.*

If no difference is noticed in the appearance of the polyline, you may need to use the **Regen** option. The segment with the revised widths is redrawn after invoking the **Regen** option or when you exit the vertex mode editing.

eXit option. This option lets you exit the vertex mode editing and return to the **PEDIT** prompt.

Fit Option

The **Fit** or **Fit curve** option generates a curve that passes through all the corners (vertices) of the polyline, using the tangent directions of the vertices (Figure 18-19). The curve is composed of a series of arcs passing through the corners (vertices) of the polyline. This option is used when you draw a polyline with sharp corners and need to convert it into a series of smooth curves. An example of this is a graph. In a graph, we need to show a curve by joining a series of plotted points. The process involved is called curve fitting; therefore, the name of this option. The vertices of the polyline is also known as the control points. The closer together these control

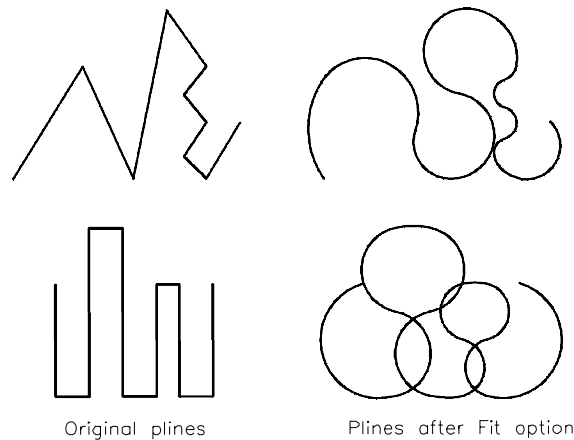


Figure 18-19 The **Fit** option

points are, the smoother the curve. Therefore, if the **Fit** option does not give optimum results, insert more vertices into the polyline or edit the tangent directions of vertices and then use the **Fit** option on the polyline. Before using this option, you may give each vertex a tangent direction. The curve is then constructed, according to the tangent directions you have specified. The following prompt sequence illustrates the **Fit** option after you invoke the **PEDIT** command.

Select polyline or [Multiple]: *Select the polyline to be edited.*
Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **F**

If the tangent directions need to be edited, use the **Edit vertex** option of the **PEDIT** command. Move the X marker to each of the vertices that need to be changed. Now, you can invoke the **Tangent** option, and either enter the tangent direction in degrees or select points. The chosen direction is expressed by an arrow placed at the vertex. The prompt sequence, after you invoke the **PEDIT** command, is given next.

Select polyline or [Multiple]: *Select the polyline to be edited.*
Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **E**
Enter a vertex editing option
[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit]<N>: **T**
Specify direction of vertex tangent: *Specify a direction in + or - degrees, or select a point in the desired direction. Press ENTER.*

Once you specify the tangent direction, use the **eXit** option to return to the previous prompt and use its **Fit** option.

Spline Option

The **Spline** option (Figure 18-20) also smoothens the corners of a straight segment polyline, as does the **Fit** option, but the curve passes through only the first and the last control points (vertices), except in the case of a closed polyline. The spline curve is stretched toward the other

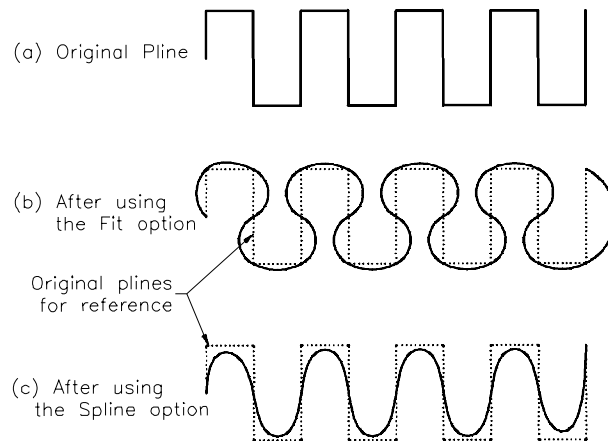


Figure 18-20 The *Spline* option

control points (vertices) but does not pass through them, as in the case of the **Fit** option. The greater the number of control points, the greater the force with which the curve is stretched toward them. The prompt sequence after you invoke the **PEDIT** command is as follows:

Select polyline or [Multiple]: *Select the polyline.*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **S**

The generated curve is a B-spline curve. The **frame** is the original polyline without any curves in it. If the original polyline has arc segments, these segments are straightened when the spline's frame is formed. A frame that has width produces a spline curve that tapers smoothly from the width of the first vertex to that of the last. Any other width specification between the first width specification and the last is neglected. When a spline is formed from a polyline, the frame is displayed as a polyline with a zero width and a continuous linetype. Also, AutoCAD LT saves its frame so that it may be restored to its original form. Tangent specifications on control point vertices do not affect spline construction.

By default, the spline frames are not shown on the screen, but you may want them displayed for reference. In this case, the system variable **SPLFRAME** needs to be manipulated. The default value for this variable is zero. If you want to see the spline frame as well, set it to 1.

Now, whenever the **Spline** option is used on a polyline, the frame will also be displayed. Most editing commands such as **MOVE**, **ERASE**, **COPY**, **MIRROR**, **ROTATE**, and **SCALE** work similarly for both the Fit curves and Spline curves. They work on both the curve and its frame, whether the frame is visible or not. The **EXTEND** command changes the frame by adding a vertex at the point the last segment intersects with the boundary. If you use any of the previously mentioned commands and then use the **Decurve** option to decurve the spline curve and later use the **Spline** option again, the same spline curve is generated. The **BREAK**, **TRIM**, and **EXPLODE** commands delete the frame. The **DIVIDE**, **MEASURE**, **FILLET**, **CHAMFER**, **AREA**, and **HATCH** commands recognize only the spline curve and do not consider the frame. The **STRETCH** command first stretches the frame and then fits the spline curve to it.

When you use the **Join** option of the **PEDIT** command, the spline curve is decurved and the original spline information is lost. The **Next** and **Previous** options of the **Edit vertex** option of the **PEDIT** command move the marker only to the points on the frame, whether visible or not. The **Break** option discards the spline curve. **Insert**, **Move**, **Straighten**, and **Width** options refit the spline curve. Object Snaps consider the spline curve and not the frame; therefore, if you wish to snap to the frame control points, restore the original frame.

There are two types of spline curves:

Quadratic B-spline

Cubic B-spline

Both of them pass through the first and the last control points, which is characteristic of the spline curve. Cubic curves are very smooth. A cubic curve passes through the first and last control points, and the curve is closer to the other control points. Quadratic curves are not as smooth as the cubic ones, but they are smoother than the curves produced by the **Fit curve** option. A quadratic curve passes through the first and last control points, and the rest of the curve is tangent to the polyline segments between the remaining control points (Figure 18-21).

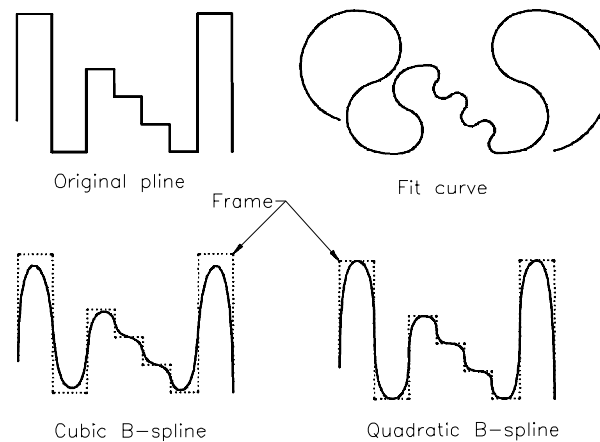


Figure 18-21 Comparison of fit curve, quadratic B-spline, and cubic B-spline

Generation of different types of spline curves. If you want to edit a polyline into a B-spline curve, you are first required to enter a relevant value in the **SPLINETYPE** system variable. A value of 5 produces the quadratic curve, whereas a value of 6 produces a cubic curve. The default value is 6, which implies that when we use the **Spline** option of the **PEDIT** command, a cubic curve is produced by default. You can change the value of the **SPLINETYPE** variable using the command line.

SPLINESEGS. The system variable **SPLINESEGS** governs the number of line segments used to construct the spline curves, and so you can use this variable to control the smoothness of the curve. The default value for this variable is 8. With this value, a reasonably smooth curve that does not need much regeneration time is generated. The greater the value of this

variable, the smoother the curve, but greater the regeneration time, the more space occupied by the drawing file.

Figure 18-22 shows cubic curves with different values for the **SPLINESEGS** parameter.

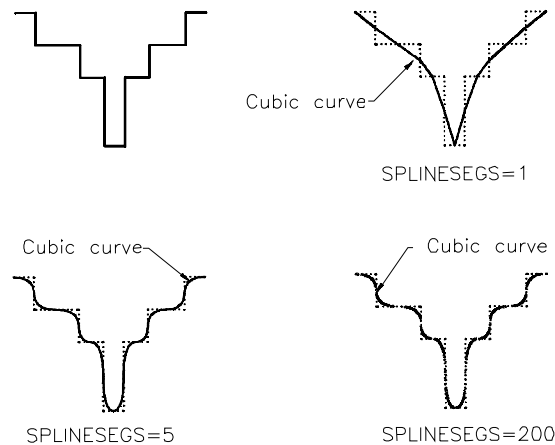


Figure 18-22 Using the **SPLINESEGS** variable

Decurve Option

The **Decurve** option straightens the curves generated after using the **Fit** or **Spline** option on a polyline. They return to their original shape (Figure 18-23). The polyline segments are straightened using the **Decurve** option. The vertices inserted after using the **Fit** or **Spline** option are also removed. Information entered for the tangent reference is retained for use in future fit curve operations. You can also use this command to straighten out any curve drawn with the help of the **Arc** option of the **PLINE** command.

Enter **D** at the **Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]** prompt to invoke the **Decurve** option of the **PEDIT** command.

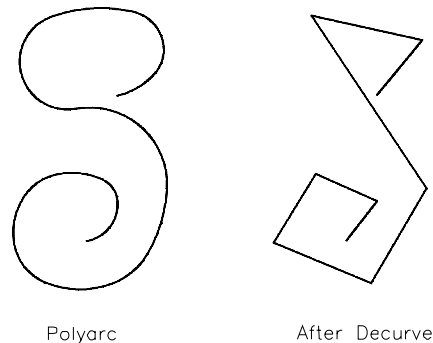


Figure 18-23 The **Decurve** option



Note

If you use edit commands such as the **BREAK** or **TRIM** commands on spline curves, the **Decurve** option cannot be used.

Ltype gen Option

You can use this option to control the linetype pattern generation for linetypes other than Continuous with respect to the vertices of the polyline. This option has two modes ON and OFF. If turned off, the break in the noncontinuous linetypes will be avoided at the vertices of the

polyline and a continuous segment will be displayed at the vertices (Figure 18-24). If turned on, this option generates the linetype in a continuous pattern such that the gaps may be displayed at the vertices. This option is not applicable to polylines with tapered segments.

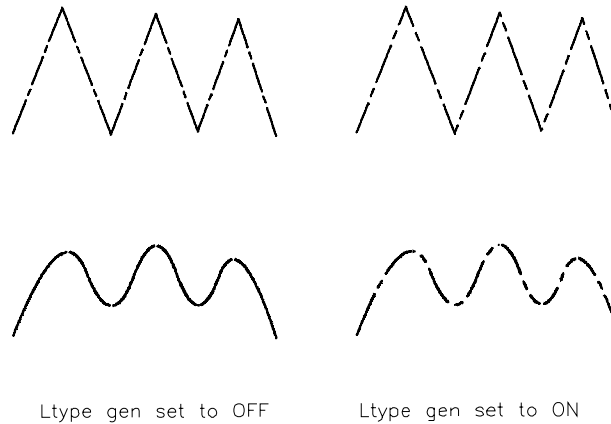


Figure 18-24 Using the **Ltype gen** option

The command prompt displayed when you select a polyline and invoke the **Ltype gen** option of the **PEDIT** command is as follows.

Enter polyline linetype generation option [ON/OFF] <Off>: Enter ON or press ENTER to accept the default value.

The linetype generation for new polylines can also be controlled with the help of the **PLINEGEN** system variable, which acts as a toggle. The default value is 0 (off). Entering a value of 1 turns on the **Ltype gen** option.

Undo (U) Option

The **Undo** option negates the effect of the most recent **PEDIT** operation. You can go back as far as you need to in the current **PEDIT** session by using the **Undo** option repeatedly until you get the desired screen. If you started editing by converting an object into a polyline, and you want to change the polyline back to the object from which it was created, the **Undo** option of the **PEDIT** command will not work. In this case, you will have to exit to the Command prompt and use the **UNDO** command to undo the operation.

Editing Multiple Polylines

Selecting the **Multiple** option of the **PEDIT** command allows you to select more than one polyline for editing. You can select the polylines using any of the objects selection techniques. After the objects to be edited are selected, press ENTER or right-click to proceed with the command. The prompt sequence that will follow is given next.

Enter an option [Close/Open/Join/Width/Fit/Spline/Decurve/Ltype gen/Undo]:

Join

This option is used to join more than one polylines that may or may not be in contact with each other. Even the polylines that are not coincident can be joined using this option. The polylines are joined using a specified distance value called the **Fuzz distance**. After you select the **Join** option, the sequence of prompts is as follows.

Join Type = Extend

Enter fuzz distance or [Jointype] <current> : *Enter a distance or J for changing the Jointype.*

The prompt sequence to follow when you enter **J** at the above prompt is given next.

Enter join type [Extend/Add/Both] <Extend>:

Extend. This option extends or trims the selected polylines at the endpoints that are nearest to each other; see Figure 18-25. Keep in mind that the segments of the selected polylines that are nearest to each other should not be parallel. This means that the segments that are nearest to each other should intersect at some point when extended.

Add. This option joins the selected polylines by drawing a straight line between the nearest endpoints, see Figure 18-26. The fuzz distance should be greater than the actual distance between the two endpoints to be joined.

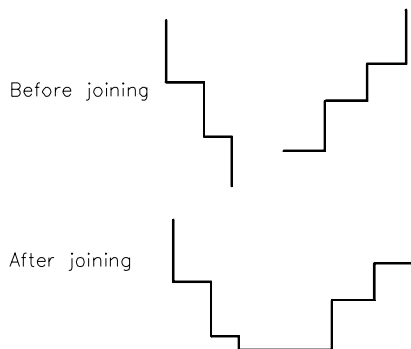


Figure 18-25 Joining two polylines using the **Extend** option

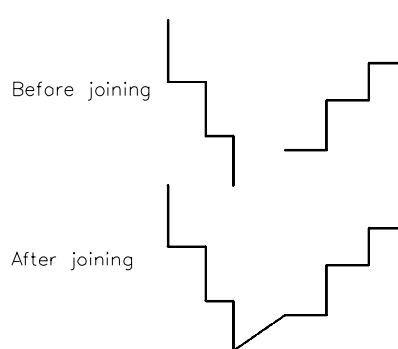


Figure 18-26 Joining two polylines using the **Add** option

Both. This option joins the endpoints by extending or trimming if possible; otherwise, it adds a line segment between the two endpoints.

Exercise 2*General*

- a. Draw a line from point (0,0) to point (6,6). Convert the line into a polyline with a starting width 0.30 and a ending width 0.00.
- b. Draw different polylines of varying width segments and then use the **Join** option to join all the segments into one polyline. After joining the segments, make their width uniform, with the **Width** option.

Exercise 3*General*

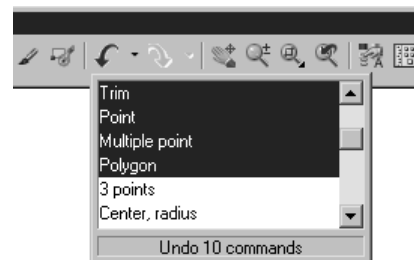
- a. Draw a staircase-shaped polyline, then use the different options of the **PEDIT** command to generate a fit curve, a quadratic B-spline, and a cubic B-spline, and then convert the curves back to the original polyline.
- b. Draw square-wave-shaped polylines, and use different options of the **Edit vertex** option to navigate around the polyline, split the polyline into two at the third vertex, insert more vertices at different locations in the original polyline, and convert the square-wave-shaped polyline into a straight-line polyline.

UNDOING PREVIOUS COMMANDS**Command:** UNDO

The **LINE** and **PLINE** commands have the **Undo** option, which can be used to undo (nullify) the changes made within these commands. The **UNDO** command is used to undo a previous command or to undo more than one command at one time. This command can be invoked by entering **UNDO** at the Command prompt. **Undo** is also available from the **Edit** menu and the **Standard** toolbar. The **Undo** command in the **Edit** menu can only undo the previous command and only one command at a time. If you right-click in the drawing area a shortcut menu is displayed. Choose **Undo**. It is equivalent to **UNDO 1** or to enter **U** at the Command prompt. The **U** command can undo only one operation at a time. You can use **U** as many times as you want until AutoCAD LT displays the message: **Nothing to undo**. When an operation cannot be undone, the message name is displayed but no action takes place. External commands such as **PLOT** cannot be undone.

Undoing Previous Commands using the Standard Toolbar

Click the **Undo list** arrow in the **Standard** toolbar. The **Undo List** box is displayed. All the commands recently used are listed in the **Undo List** box. Select the commands that you want to undo by placing the cursor in the list box and moving downward. The Commands selected are highlighted in blue color, as shown in Figure 18-27. The

*Figure 18-27 The Undo List box*

Undo List box **Status bar** displays the number of commands selected. Click the left mouse button to confirm the selections.

Undoing Previous Commands using the Command Line

To **Undo** previous commands using the Command Line, enter **UNDO** at the commands line (if you are working in dynamic mode the command will be entered at **Pointer Input**). The prompt sequence for the **UNDO** command is given next.

Command: **UNDO**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>:

Enter a positive number, an option, or press ENTER to undo a single operation.

The various options of this command are discussed next.

Number (N) Option

This is the default option. This number represents the number of previous command sequences to be deleted. For example, if the number entered is 1, the previous command is undone; if the number entered is 4, the previous four commands are undone, and so on. This is identical to invoking the **U** command four times, except that only one regeneration takes place. AutoCAD LT lets you know which commands were undone by displaying a message after you press **ENTER**. The prompt sequence is given next.

Command: **UNDO**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: 3

PLINE LINE CIRCLE

Control (C) Option

This option lets you determine how many of the options are active in the **UNDO** command. You can disable the options you do not need. With this option, you can even disable the **UNDO** command. To access this option, type **C**. You will get the following prompt.

Enter an UNDO control option [ALL/None/One] <All>: *Enter an option or press ENTER to accept the default.*

ALL

The **All** option activates all the features (options) of the **UNDO** command.

None (N)

This option turns off **UNDO** and the **U** command. If you have used the **BEGIN** and **End** options or **Mark** and **Back** options to create **UNDO** information, all of that information is undone. The prompt sequence for invoking the **Control** option is as follows.

Command: **UNDO**

Enter the number of operations to undo [Auto/Control/BEGIN/End/Mark/Back] <1>: **C**

Enter an UNDO control option [ALL/None/One] <All>: **N**

If you try to use the **U** command now, while the **UNDO** command has been disabled, AutoCAD LT gives you the following message.

Command: **U**

U command disabled. Use **UNDO** command to turn it on.

The prompt sequence for the **UNDO** command after issuing the **None** option is given next.

Command: **UNDO**

Enter an UNDO control option [All/None/One] <All>: *Enter O or press ENTER to accept the default.*

To enable the **UNDO** options again, you must enter the **All** or **One** (one mode) option.

One (O)

This option restrains the **UNDO** command to a single operation. All **UNDO** information saved earlier during editing is scrapped. The prompt sequence is as follows.

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: **C**
Enter an UNDO control option [All/None/One] <All>: **O**

If you then enter the **UNDO** command, you will get the following prompt.

Command: **UNDO**

Enter an option [Control] <1>: *Press ENTER to undo the last operation or enter C to select another control option.*

In response to this last prompt, you can now either press ENTER to undo only the previous command, or go into the **Control** options by entering C. AutoCAD LT acknowledges undoing the previous command with messages such as given next.

Command: **UNDO**

Enter an option [Control] <1>: 

CIRCLE

Everything has been undone



Tip

*AutoCAD LT stores all information about all the **UNDO** operations taking place in a drawing session. When you use the **None** control option of the **UNDO** command, this information is removed and valuable disk space is made available. In case of limited disk space availability, it may be a good idea to use the **One** suboption of the **UNDO Control** command. This allows you to use both the **U** and **UNDO** commands and at the same time makes the disk space available.*

Auto (A) Option

Enter A to invoke this option. The following prompt is displayed.

Enter UNDO Auto mode [ON/OFF] <current>: *Select ON or OFF or press ENTER to accept the default.*

This option is on by default and every menu item is a single operation. When you use the **B**egin and **E**nd options to group a series of commands as a single operation, entering **U** at the Command prompt removes the objects individually and not as a group, if the **Auto** option is on. But if **Auto** is off, entering **U** at the Command prompt will remove the entire group of objects you had grouped using the **B**egin and **E**nd options in one single operation.

Also, if you have put a marker in between the start and end operations and if **Auto** is On, the group of commands till the marker, is undone altogether on using the **B**ack option. This group of commands is considered to be a single group and is undone as a single command, although, entering **U** at the Command prompt removes the objects individually until the marker is encountered. But, if the **Auto** option is off, entering **U** at the Command prompt, or using the **B**ack option of the **UNDO** command, undoes the entire group of operations irrespective of the marker being there.

B

A group of commands is treated as one command for the **U** and **UNDO** commands (provided **Auto** option is off) by embedding the commands between the **B**egin and **E**nd options of the **UNDO** command. If you anticipate the removal of a group of successive commands later in a drawing, you can use this option, since all of the commands after the **B**egin option and before the **E**nd option are treated as a single command by the **U** command (if the **Auto** option is Off). For example, the following sequence illustrates the possibility of removal of two commands.

Command: **UNDO**

Enter the number of operations to undo or [Auto/Control/B

Command: **CIRCLE**

Specify center point for Circle or [3P/2P/Ttr (tan tan radius)]: *Specify the center.*

Specify radius of circle or [Diameter] <current>: *Specify the radius of the circle.*

Command: **PLINE**

Specify start point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Select first point.*

Specify next point: *Select the next point.*

Specify start point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Press ENTER.*

Command: **UNDO**

Enter the number of operations to UNDO or [Auto/Control/B

Command: **U**

To start the next group once you have finished specifying the current group, use the **E**nd option to end this group. Another method is to enter the **B**egin option to start the next group while the current group is active. This is equivalent to issuing the **E**nd option followed by the **B**egin option. The group is complete only when the **E**nd option is invoked to match a **B**egin option.

If **U** or the **UNDO** command is issued after the **BEGIN** option has been invoked and before the **End** option has been issued, only one command is undone at a time until it reaches the juncture where the **BEGIN** option has been entered. If you want to undo the commands issued before the **BEGIN** option was invoked, you must enter the **End** option so that the group is complete. This is demonstrated in the following example.

Example 2

General

Enter the following commands in the same sequence as given, and notice the changes that take place on screen.

```
CIRCLE
POLYGON
UNDO BEGIN (Make sure that the Auto is OFF)
PLINE
TRACE
U
DONUT
UNDO End
TEXT
U
U
U
U
```

The first **U** command will undo the **TRACE** command. If you repeat the **U** command, the **PLINE** command will be undone. Any further invoking of the **U** command will not undo any previously drawn object (**POLYGON** and **CIRCLE**, in this case), because after the **PLINE** is undone, you have an **UNDO BEGIN**. Only after you enter **UNDO End** can you undo the **POLYGON** and the **CIRCLE**. In the example, the second **U** command will undo the **TEXT** command, the third **U** command will undo the **DONUT** and **PLINE** commands (these are enclosed in the group), the fourth **U** command will undo the **POLYGON** command, and the fifth **U** command will undo the **CIRCLE** command. Whenever the commands in a group are undone, the name of each command or operation is not displayed as it is undone. Only the name, **GROUP**, is displayed.



Tip

*You can use the **BEGIN** and **End** options only when the **UNDO Control** is set to **All** and the **Auto** option is off.*

Mark (M) and Back Options

The **Mark** option installs a marker in the Undo file. The **Back** option lets you undo all the operations until the mark. In other words, the **Back** option returns the drawing to the point where the previous mark was inserted. For example, if you have completed a portion of your drawing and do not want anything up to this point to be deleted, you can insert a marker and then proceed. Then, even if you use the **UNDO Back** option, it will work only until the marker. You

can insert multiple markers, and with the help of the **Back** option, you can return to the successive mark points. The following prompt sequence is used to introduce a mark point.

Command: **UNDO**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back]: **M**

The prompt sequence for using the **Back** option is given next.

Command: **UNDO**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back]: **B**

Once all the marks have been exhausted with the successive **Back** options, any further invoking of the **Back** option displays the message: **This will undo everything. OK? <Y>**. If you enter Y (Yes) at this prompt, all operations carried out since you entered the current drawing session will be undone. If you enter N (No) at this prompt, the **Back** option will be disregarded.

You cannot undo certain commands and system variables, for example, **DIST**, **LIST**, **DELAY**, **NEW**, **OPEN**, **QUIT**, **AREA**, **HELP**, **PLOT**, **QSAVE** and **SAVE**, among many more. Actually, these commands have no effect that can be undone. Commands that change operating modes (**GRID**, **UNITS**, **SNAP**, **ORTHO**) can be undone, though the effect may not be apparent at first. This is the reason why AutoCAD LT displays the command names as they are undone.

REVERSING THE UNDO OPERATION

| | |
|-----------------|-----------------|
| Toolbar: | Standard > Redo |
| Menu: | Edit > Redo |
| Command: | REDO |



If you right-click in the drawing area, a shortcut menu is displayed. Choose **Redo** to invoke the **REDO** command. The **REDO** command brings back the process you removed previously using the **U** and **UNDO** commands. This command undoes the last **UNDO** command, but it must be entered immediately after the **UNDO** command and the objects previously undone reappear on the screen. Click the **Redo list** arrow in **Standard** toolbar. The **Redo List** box (Fig 18-28) is displayed; it lists the commands that have been recently undone. Select the commands that you want to **Redo** and click the left mouse button to confirm the selection.

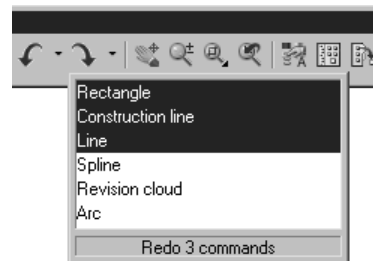


Figure 18-28 The Redo List box



Note

With this release of AutoCAD LT, consecutive **Zoom** and **Pan** operations can be undone or redone by using the **Undo** or **Redo** command only once so that the drawing returns to the initial zoom state. To disable this feature, clear the **Combine zoom and pan commands** check box in the **Undo/Redo** area of the **User Preferences** tab of the **Options** dialog box.

RENAMING NAMED OBJECTS

Menu: Format > Rename
Command: RENAME

You can edit the names of the named objects such as blocks, dimension styles, layers, linetypes, styles, UCS, views, and viewports using the **Rename** dialog box. You can select the named object from the list in the **Named Objects** area of the dialog box. The corresponding names of all the objects of the specified type that can be renamed are displayed in the **Items** area. For example, if you want to rename the layer named LOCKED to HID, the process will be as follows.

1. Select **Layers** in the **Named Objects** list box. All the layer names in the current drawing that can be renamed are displayed in the **Items** list box.
2. Select LOCKED in the **Items** list box to highlight it. LOCKED is displayed in the **Old Name** edit box.
3. Enter HID in the **Rename To** edit box (Figure 18-29), and choose the **OK** button.

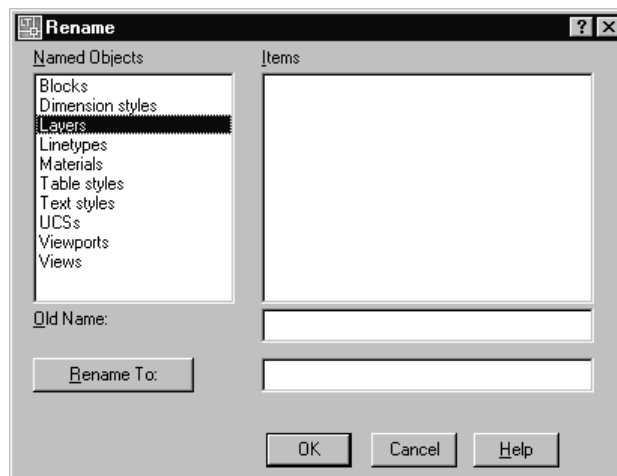


Figure 18-29 The **Rename** dialog box

Now, the layer named LOCKED is renamed to HID. You can invoke the **Rename** dialog box again, view the **Layer Control** drop-down list in the **Layers** toolbar, or invoke the **Layer Properties Manager** dialog box to notice this change. You can rename blocks, dimension styles, linetypes, styles, UCS, views, and viewports in the same way.

**Note**

The **RENAME** command cannot be used to rename drawing files created using the **WBLOCK** command. You can change the name of a drawing file by using the **Rename** option of your Operating System.

The Layer **0** and the **Continuous** linetype cannot be renamed and, therefore, do not appear in the **Items** list box of the **Rename** dialog box.

REMOVING UNUSED NAMED OBJECTS

| | |
|-----------------|----------------------------------|
| Menu: | File > Drawing Utilities > Purge |
| Command: | PURGE |

This is another editing operation used for deletion and it was discussed earlier in relation to blocks. You can delete unused named objects such as blocks, layers, dimension styles, linetypes, text styles, and shapes with the help of the **PURGE** command. When you create a new drawing or open an existing one, AutoCAD LT records the named objects in that drawing and notes other drawings that reference the named objects. Usually only a few of the named objects in the drawing (such as layers, linetypes, and blocks) are used. For example, when you create a new drawing, the prototype drawing settings may contain various text styles, blocks, and layers which you do not want to use. Also, you may want to delete particular unused named objects such as unused blocks in an existing drawing. Deleting inactive named objects is important and useful because doing so reduces the space occupied by the drawing. With the **PURGE** command, you can select the named objects you want to delete. You can use this command at any time in the drawing session. When you invoke the **Purge** command, the **Purge** dialog box is displayed (Figure 18-30).

View items you can purge

When this radio button is selected, AutoCAD LT displays the tree view of all the named objects that are in the current drawing and those can be purged. When this radio button is chosen, **Items not used in drawing** area, and the two check boxes, **Confirm each item to be purged** and **Purge nested items**, are displayed in the dialog box. These are as discussed as follows.

Items not used in drawing

This area lists all the named objects that can be purged. You can list the items of any object type by choosing the plus (+) sign or by double-clicking on the named object in the tree view. Select them to be purged. You can select more than one item by holding down the SHIFT or the CTRL key and selecting the items.

Confirm each item to be purged

This check box is selected to confirm before purging the selected item. The **Confirm Purge** dialog box is displayed when items are selected to be purged and after the **Purge** or **Purge All** buttons are chosen. You can confirm or cancel the items to be purged in this dialog box.

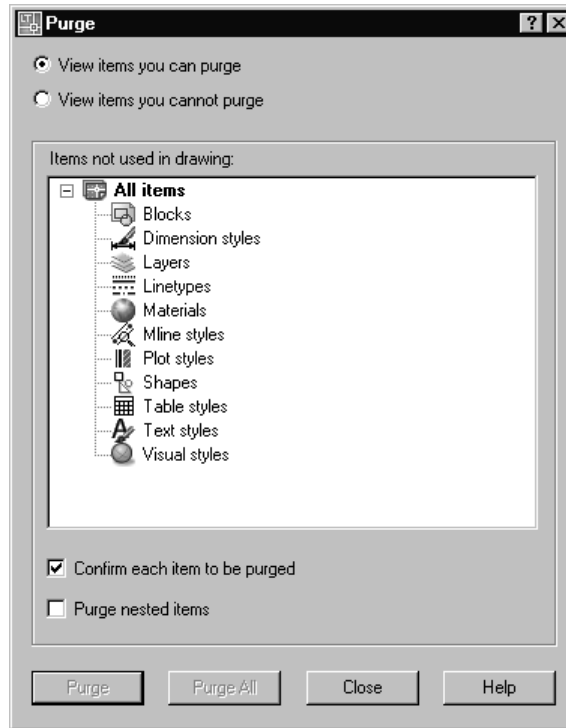


Figure 18-30 The **Purge** dialog box with items that can be purged

Purge nested items

This check box, when selected, removes all the nested items not in use. This removes the nested items only when you select the **Purge All** button or select blocks.

View items you cannot purge

This radio button is selected to display the tree view of the items you cannot purge. These are the items that are used in the current drawing or are default items that cannot be removed (Figure 18-31). Choosing this radio button displays the tree view in the **Items currently used in drawing** area of the dialog box. When you select an object in this tree view, information about why you cannot purge this object is displayed below the tree view.



Note

The **Entire Drawing** option of the **Write Block** dialog box that is invoked on using the **WBLOCK** command or the **-WBLOCK asterisk** option that have been discussed earlier have the same effect as the **PURGE All** command. The only difference is that the unused named objects are removed automatically without any verifications, though these methods are much faster.

Standard objects created by AutoCAD LT (such as layer 0, STANDARD text style, and linetype CONTINUOUS) cannot be removed by the **PURGE** command, even if these objects are not used.

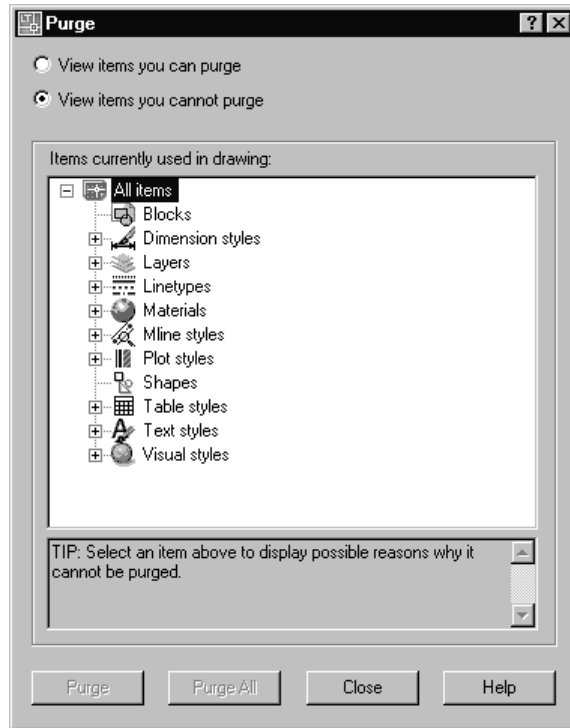


Figure 18-31 The **Purge** dialog box with items that cannot be removed

SETTING SELECTION MODES USING THE OPTIONS DIALOG BOX

Menu: Tools > Options
Command: OPTIONS

When you select a number of objects, the selected objects form a **selection set**. Selection of the objects is controlled in the **Options** dialog box (Figure 18-32) that is invoked by the **OPTIONS** command. Six selection modes are provided in the **Selection** tab of this dialog box. You can select any one of these modes or a combination of various modes.

Noun/verb selection

By selecting the **Noun/verb selection** check box, you can select the objects (noun) first and then specify the operation (verb) (command) to be performed on the selection set. This mode is active by default. For example, if you want to move some objects when the mode **Noun/verb selection** is enabled, first select the objects to be moved, and then invoke the **MOVE** command. The objects selected are highlighted automatically when the **MOVE** command is invoked, and AutoCAD LT does not issue any **Select objects** prompt. The following are some of the commands that can be used on the selected objects when the **Noun/verb selection** mode is active.

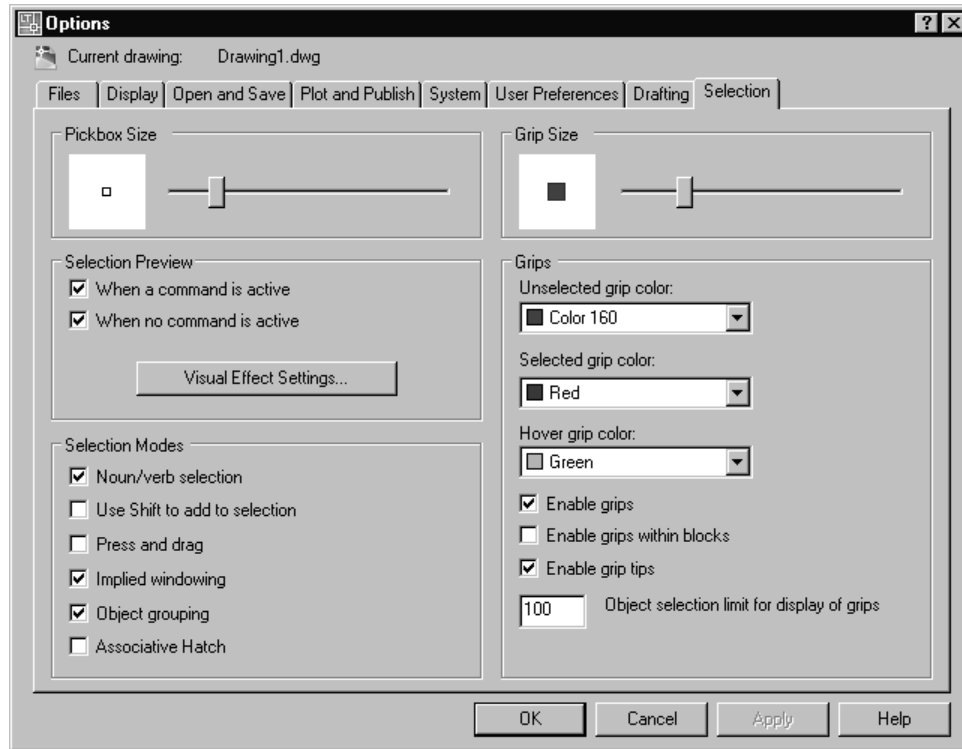


Figure 18-32 The Selection tab of the Options dialog box

| | | | | |
|-------|------------|--------|--------|--------|
| ARRAY | BLOCK | CHANGE | CHPROP | COPY |
| DVIEW | EXPLODE | ERASE | LIST | MIRROR |
| MOVE | PROPERTIES | ROTATE | SCALE | WBLOCK |

The following are some of the commands that are not affected by the **Noun/verb selection** mode. You are required to specify the objects (noun) on which an operation (command/verb) is to be performed, after specifying the command (verb).

| | | | | |
|---------|---------|--------|----------|----------|
| BREAK | CHAMFER | DIVIDE | EDGESURF | EXTEND |
| FILLET | MEASURE | OFFSET | REVSURF | RULESURF |
| TABSURF | TRIM | | | |

When the **Noun/Verb selection** mode is active, the **PICKFIRST** system variable is set to 1 (On). In other words, you can also activate the **Noun/verb selection** mode by setting the **PICKFIRST** system variable to 1 (On).

Use Shift to add to selection Option

The next option in the **Selection Modes** area of the **Selection** tab of the **Options** dialog box is **Use Shift to add to selection**. Selecting this option establishes the additive selection mode, which is the normal method of most Windows programs. In this mode, you have to hold down

the SHIFT key when you want to add objects to the selection set. For example, suppose X, Y, and Z are three objects on the screen and you have selected the **Use Shift to add to selection** check box in the **Selection Modes** area of the **Options** dialog box. Select object X. It is highlighted and put in the selection set. After selecting X, and while selecting object Y, if you do not hold down the SHIFT key, only object Y is highlighted and it replaces object X in the selection set. On the other hand, if you hold down the SHIFT key while selecting Y, it is added to the selection set (which contains X), and the resulting selection set contains both X and Y. Also, both X and Y are highlighted. To summarize the concept, objects are added to the selection set only when the SHIFT key is held down while objects are selected. Objects can be discarded from the selection set by reselecting these objects while the SHIFT key is held down. If you want to clear an entire selection set quickly, draw a blank selection window anywhere in a blank drawing area. You can also right-click to display a shortcut menu and choose **Deselect All**. All selected objects in the selection set are discarded from it.

When the **Use Shift to add to selection** mode is active, the **PICKADD** system variable is set to 0 (Off). In other words, you can activate the **Use Shift to add to selection** mode by setting the **PICKADD** system variable to off.

Press and drag

This selection mode is used to govern the way you can define a selection window or a crossing window. When you select this option, you can create the window by pressing the pick button to select one corner of the window and continuing to hold down the pick button and dragging the cursor to define the other diagonal point of the window. When you have the window you want, release the pick button. If the **Press and drag** mode is not active, you have to select twice to specify the two diagonal corners of the window to be defined. This mode is not active by default. This implies that to define a selection window or a crossing window, you have to select twice to define their two opposite corner points.

When the **Press and drag** mode is active, the **PICKDRAG** system variable is set to 1 (On). In other words, you can activate the **Press and drag** mode by setting **PICKDRAG** to On.

Implied windowing

By selecting this option, you can automatically create a Window or Crossing selection when the **Select objects** prompt is issued. The selection window or crossing window, in this case, is created in the following manner: At the **Select objects** prompt, select a point in the empty space on the screen. This becomes the first corner point of the selection window. After this, AutoCAD LT asks you to specify the other corner point of the selection window. If the first corner point is to the right of the second corner point, a Crossing selection is defined; if the first corner point is to the left of the second corner point, a Window selection is defined. The **Implied windowing** check box is selected by default.

If this option is not active, or if you need to select the first corner in a crowded area where selecting would select an object, you need to specify Window or Crossing at the **Select objects** prompt, depending on your requirement.

When the **Implied windowing** mode is active, the **PICKAUTO** system variable is set to 1 (On). In other words, you can activate the **Implied windowing** mode by setting the value of the **PICKAUTO** system variable to 1 (On).

Object grouping

This turns the automatic group selection on and off. When this option is on and you select a member of a group, the whole group is selected. You can also activate this option by setting the value of the **PICKSTYLE** system variable to 1. (Groups were discussed earlier in this chapter.)



Tip

*You can clear the **Object grouping** check box in the **Selection Modes** area of the **Selection** tab of the **Options** dialog box to be able to select the objects of a group individually for editing without having to explode the group.*

Associative Hatch

If the **Associative Hatch** check box is selected in the **Selection modes** area of the **Selection** tab of the **Options** dialog box, the boundary object is also selected when an associative hatch is selected. You can also select this option by setting the value of the **PICKSTYLE** system variable to 2 or 3. Hatching and boundaries have been discussed in Chapter 13, Hatching Drawings. It is recommended that you select the **Associative Hatch** check box for most drawings.

Pickbox Size

The **Pickbox** slider bar controls the size of the pickbox. The size ranges from 0 to 20. The default size is 3. You can also use the **PICKBOX** system variable.

Self-Evaluation Test

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

1. Even if a group is defined as selectable, if the **PICKSTYLE** system variable is set to 0, you will not be able to select the entire group by selecting one of its members. (T/F)
2. Only the **CHANGE** command can be used to change the properties associated with an object. (T/F)
3. The **PROPERTIES** palette can be used to modify the geometry of objects apart from their general properties. (T/F)
4. The **RENAME** command can be used to change the name of a drawing file created by the **WBLOCK** command. (T/F)
5. A group of commands is treated as one command for the **U** and **UNDO** commands by embedding them between the _____ and _____ options of the **UNDO** command.

6. If the **Press and Drag** mode is not active, you have to select _____ to specify the two diagonal points of a selection window.
7. The _____ option of the **PEDIT** command's main prompt can be used to change the starting and ending widths of a polyline separately to a desired value.
8. The _____ option of the **PEDIT** command allows you to select more than one polyline.
9. You can move past the first and last vertices in a closed polyline by using either the _____ option or the _____ option.
10. While using the _____ command, if you select a line or an arc, AutoCAD LT provides you with the option of converting them into a polyline first.

Review Questions

Answer the following questions:

1. When you use the **GROUP** command to form object groups, AutoCAD LT lets you sequentially highlight the groups of which the selected object is a member. (T/F)
2. The color, linetype, lineweight, and layer on which a block is drawn can be changed with the help of the **CHANGE** command. (T/F)
3. After exploding an object, the object remains identical except that the color and linetype may change because of floating layers, colors, or linetypes. (T/F)
4. The **PURGE** command has the same effect as the **WBLOCK Entire drawing** or the **-WBLOCK asterisk** method. The only difference is that with the **PURGE** command, deletion takes place automatically. (T/F)
5. If you have made a copy of a group without naming the copy then its name is displayed in which of the following notations in the **Group Name** list box?
 - (a) \$A1
 - (b) @A1
 - (c) %A1
 - (d) *A1
6. Which of the following options deletes the selected group from the drawing?
 - (a) **Remove**
 - (b) **Reorder**
 - (c) **Explode**
 - (d) **Rename**

7. Which of the following system variables controls the smoothness of the curve?
- (a) **SPLINETYPE** (b) **SPLINESEGS**
(b) **PLINEGEN** (c) **PICKFIRST**
8. Which option of the **PEDIT** command can straighten a curve drawn with the help of **PLINE** command?
- (a) **Join** (b) **Close**
(c) **Decurve** (d) None of the above
9. Which of the following properties cannot be changed using the **CHANGE** command?
- (a) **Lineweight** (b) **Color**
(c) **Plotstyle** (d) **Thickness**
10. The _____ command can be used to change the name of a block.
11. If the _____ option of the **UNDO** command is off, any group of commands grouped together using the **Begin** and **End** options of the **UNDO** command are undone together.
12. Circles drawn using the **CIRCLE** command can be changed to _____.
13. The _____ option in the **Group** command lets you change the order of the objects in the selected group.
14. A group can be selected by entering _____ at the AutoCAD LT **Select objects:** prompt.
15. The _____ option of the **UNDO** command disables the **UNDO** and **U** commands entirely.

Exercises

Exercise 4

Graphics

Draw part (a) in Figure 18-33 and then using the **PROPERTIES** and relevant **PEDIT** command options, convert it into parts (b), (c), and (d). The linetype used in part (d) is HIDDEN.

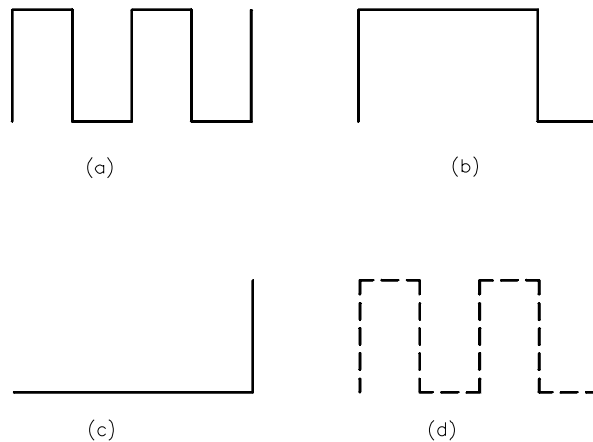


Figure 18-33 Drawing for Exercise 4

Exercise 5

Mechanical

Draw the sketch shown in Figure 18-34 using the **LINE** command. Change the object to a polyline with a width of 0.05.

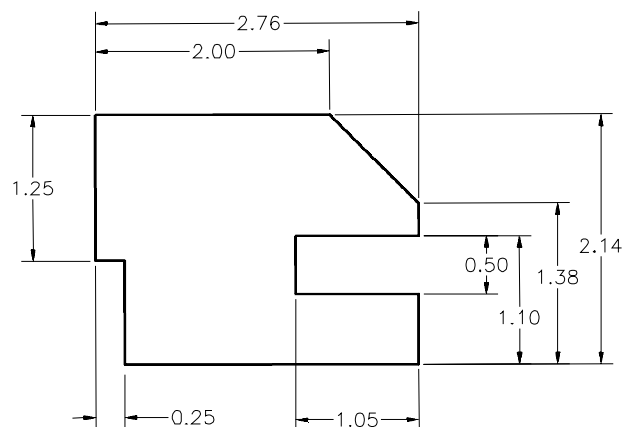


Figure 18-34 Drawing for Exercise 5

Exercise 6

Graphics

Draw part (a) in Figure 18-35; then, using the relevant **PEDIT** command options, convert it into drawing (b).

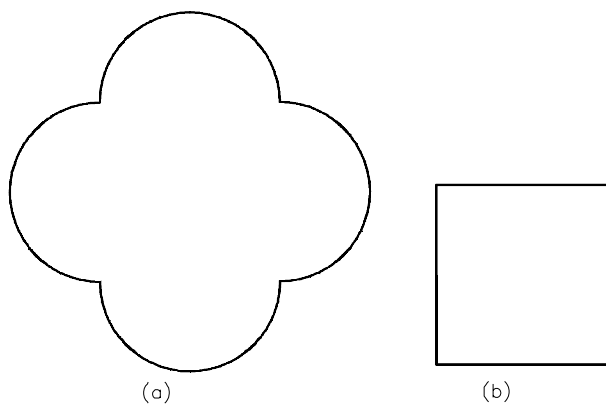


Figure 18-35 Drawing for Exercise 6

Exercise 7

Graphics

Draw part (a) in Figure 18-36; then, using the relevant **PEDIT** options, convert it into drawings (b), (c), and (d). Identify the types of curves.

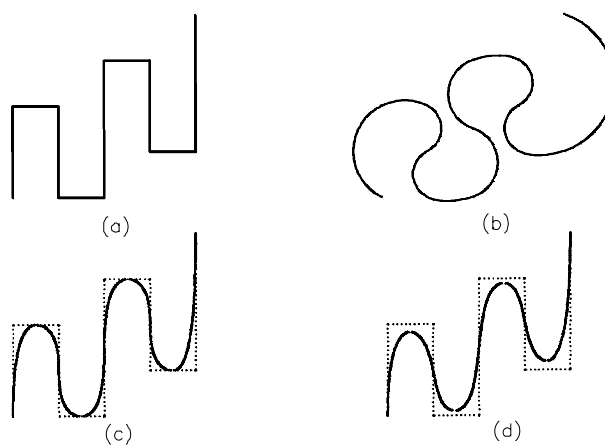


Figure 18-36 Drawing for Exercise 7

Exercise 8*Graphics*

Draw part (a) in Figure 18-37; then, using the relevant **PEDIT** options, convert it into drawing (b).

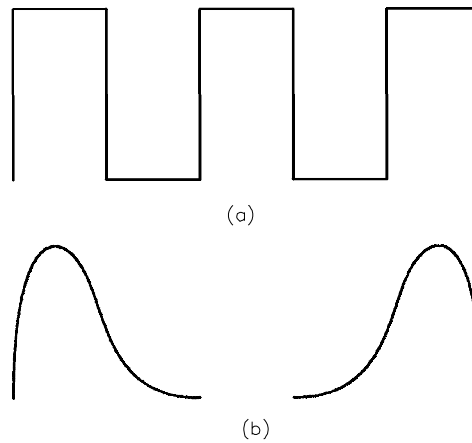


Figure 18-37 Drawing for Exercise 8

Exercise 9*Graphics*

Draw part (a) in Figure 18-38; then, using the relevant **PEDIT** options, convert it into part (b) of the drawing.

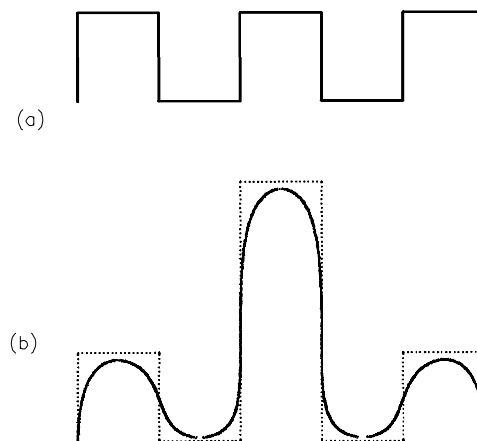


Figure 18-38 Drawing for Exercise 9

Exercise 10

Mechanical

Draw the illustration shown at the top of Figure 18-39; then use the required commands to obtain the illustration shown at the bottom. You can also use grips to obtain this illustration. Assume the missing dimensions.

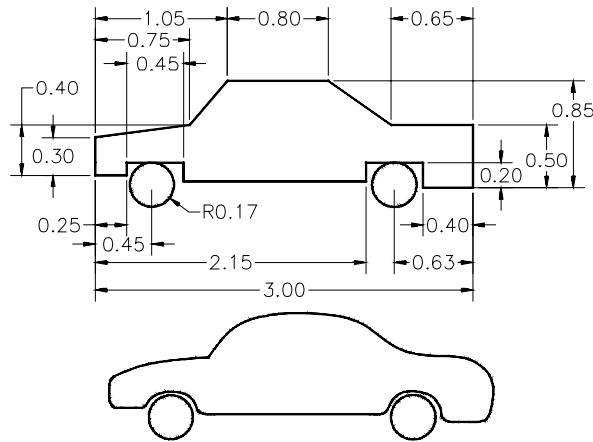


Figure 18-39 Drawing for Exercise 10

Exercise 11

Mechanical

Create the drawing shown in Figure 18-40. Use the splined polylines to draw the break lines. Dimension the drawing as shown.

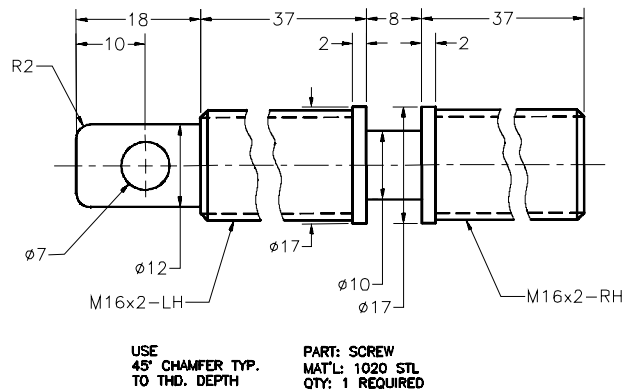


Figure 18-40 Drawing for Exercise 11

Answers to Self-Evaluation Test

1 - T, 2 - F, 3 - T, 4 - F, 5 - Begin, End, 6 - Twice, 7 - Width, 8 - Multiple, 9 - Next, Previous, 10 - PEDIT