Chapter 20

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 the named objects and remove unused named objects*

Key Terms

- *Group*
- *Explode*
- *Polyline Edit*
- *SPLINETYPE*
- *Undo*
- *Redo*
- *Rename*
- *Purge*

GROUPING SKETCHED OBJECTS USING THE OBJECT GROUPING DIALOG BOX

Ribbon: Home > Groups > Group Manager **Command:** CLASSICGROUP

In AutoCAD, you can group sketched objects using the **Object Grouping** dialog box, as shown in Figure 20-1. To invoke the **Object Grouping** dialog box, you need to choose the **Group Manager** tool from the **Groups** panel in the **Home** tab. You can use the **Object Grouping** dialog box to group AutoCAD objects and assign a name to the group. Once you have created groups, you can select the objects by the group name. The individual characteristics of an object are not affected by forming groups. Groups enable you to select all the objects in a group together for editing. It makes the object selection process easier and faster. Objects can be members of several groups. Also, a group can contain several smaller groups. This may be referred to as nested groups.

Although an object belongs to a group, you can still select an object as if it does not belong to any group. Groups can be selected by entering the group name or by selecting an object that belongs to the group. You can also highlight the objects in a group or sequentially highlight the groups to which an object belongs. The options in the **Object Grouping** dialog box are discussed next.

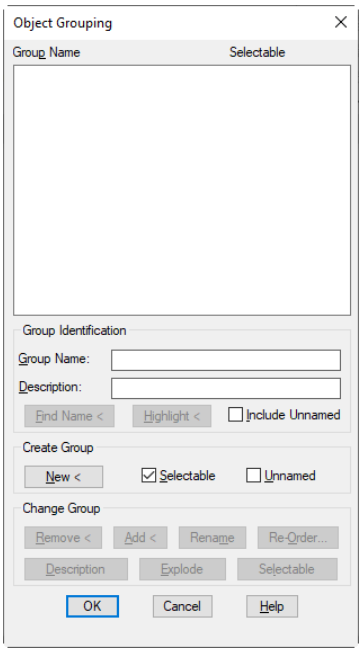

***Figure 20-1** The **Object Grouping** dialog box*

Group Name List Box

The **Group Name** list box in the **Object Grouping** dialog box displays the names of the existing groups. The list box also displays whether a group is selectable or not, under the **Selectable** column. A selectable group is the one in which all members of the group are selected on selecting a single member. However, the members that are in frozen or locked layers are not selected. You can also make a group non-selectable; which helps you to select the objects in the group individually.

Group Identification Area

The options under the **Group Identification** area are as follows:

Group Name

The **Group Name** edit box displays the name of the selected group. You can also use the **Group Name** edit box to enter the name of a new group. You can enter any name, but it is recommended that you use a name that reflects the type of objects in the group. For example, the group name WALLS can include all lines that form walls. Group names can be up to thirty-one characters long and can include letters, numbers, and special characters ($, _, and -). Group names cannot have spaces.

Description

The **Description** edit box displays the description of the selected group. It can be used to enter the description of a group.

Find Name

The **Find Name** button is used to find the group name or names associated with an object. When you choose this button, the dialog box temporarily disappears from the screen and AutoCAD prompts you to **Pick a member of a group**. Once you select the object, the **Group Member List** dialog box will be displayed, as shown in Figure 20-2, showing the group name(s) to which the selected object belongs.

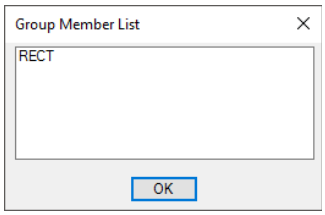

Figure 20-2 *The* ***Group Member List*** *dialog box*

Highlight

The **Highlight** button is used to highlight the objects in the selected group name. This button is available only when you select a group from the **Group Name** list box. When you choose the **Highlight** button, the objects that are members of the selected group are highlighted in the drawing and the **Object Grouping** message box is displayed, as shown in Figure 20-3. Choose the **Continue** button or press ENTER to return to the **Object Grouping** dialog box.

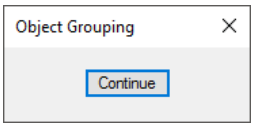

Figure 20-3 *The* ***Object Grouping*** *message box*

Include Unnamed

The **Include Unnamed** check box is used to display the names of the unnamed groups in the **Object Grouping** dialog box. The unnamed groups are created when you select the **Unnamed** check box in the **Create Group** area or when you copy the objects that belong to a group. AutoCAD automatically groups and assigns a name to the copied objects. The format of the name is A*n* (for example, *A1, *A2, *A3). If you select the **Include Unnamed** check box, the unnamed group names (*A1, *A2, *A3, ...) will also be displayed in the **Group Name** list box of the dialog box. The unnamed groups can also be created by not assigning a name to the group (refer to the **Unnamed** option in the next section, "**Create Group** Area").

Create Group Area

The options under this area are discussed next.

New

The **New** button is used to create a new group. To create a group, enter the name of the group in the **Group Name** edit box and then choose the **New** button; the **Object Grouping** dialog box will be temporarily closed and you will be prompted to select the objects to be included in the group. After selecting the objects, right-click; the dialog box will be displayed again and a new group will be created from the selected objects. The new group name is now displayed in the **Group Name** list box of the dialog box. The group names in the **Group Name** list box are listed alphabetically.

Selectable

The **Selectable** check box allows you to define a group as selectable. A selectable group has a property that if you select any one object in the group, the entire group will be selected. If you

clear this check box while defining a group, the new group created will not be selected at once. Therefore, you need to select all members of the group individually as the entire group will not be selected upon selecting the individual entities of the group.

Even if a group is defined as selectable, and the **PICKSTYLE** system variable is set to 0, you will not be able to select the entire group by selecting one of its members. You will be able to individually select members of the group. You can also turn the group selectability on or off by selecting or clearing the **Object grouping** check box in the **Selection modes** area of the **Selection** tab of the **Options** dialog box or by using the SHIFT+CTRL+A toggle keys.

Unnamed

When you create a group, you can assign it a name or leave it unnamed. If you select the **Unnamed** check box, AutoCAD will automatically assign a group name to the selected objects and these unnamed groups will be listed in the **Group Name** list box. To view the name of the unnamed group, the **Include Unnamed** check box should be selected. The format used for assigning a name is A*n* (*A1, *A2, *A3 ...), where *n* is a number which increments by 1 with each newly added unnamed group.

Change Group Area

All the options under this area, other than the **Re-Order** button, will be available only when you highlight a group name by selecting it in the **Group Name** list box of the dialog box. The options in this area are used to modify the properties of the existing group. These options are as follows.

Remove

The **Remove** button is used to remove objects from the selected group. Once you select the group name from the **Group Name** list box and then choose the **Remove** button; the dialog box will be temporarily closed and AutoCAD will display the following prompts at the command prompt.

Select objects to remove from group...
Remove objects: *(Select the objects that you want to remove from the selected group.)*

Also, all objects belonging to the selected group will be highlighted. As you select the objects to be removed from the group, they no longer appear highlighted. Note that even if you remove all objects from the group, the group name will still exist, unless you use the **Explode** option to remove the group definition (discussed later in this section). If you remove objects and then add them later in the same drawing session, the objects retain their order in the group.

Add

The **Add** button is used to add objects to the selected group. When you select this option, AutoCAD will prompt you to select the objects you want to add to the selected group. The prompt sequence in this case will be similar to that of the **Remove** option. Similar to removing an object, all objects belonging to the selected group are highlighted and as soon as you select an object to be added to the group, it also gets highlighted.

Rename

The **Rename** button is used to rename the selected group. To rename a group, first select the group name from the **Group Name** list box, enter the new name in the **Group Name** edit box,

and then choose the **Rename** button. AutoCAD will rename the specified group. You can use this option to rename unnamed groups that were discussed earlier.

Re-Order

The **Re-Order** button is used to change the order of the objects in the selected group. The objects are numbered in the order in which you select them when selecting objects for the group. Sometimes, while creating a tool path, you may want to change the order of these objects to get a continuous tool motion. You can do this using the **Re-Order** button. This option is also useful in some batch operations where one object needs to be on top of another object for display reasons. When you choose the **Re-Order** button, AutoCAD displays the **Order Group** dialog box, as shown in Figure 20-4, where you can change the order of the group members.

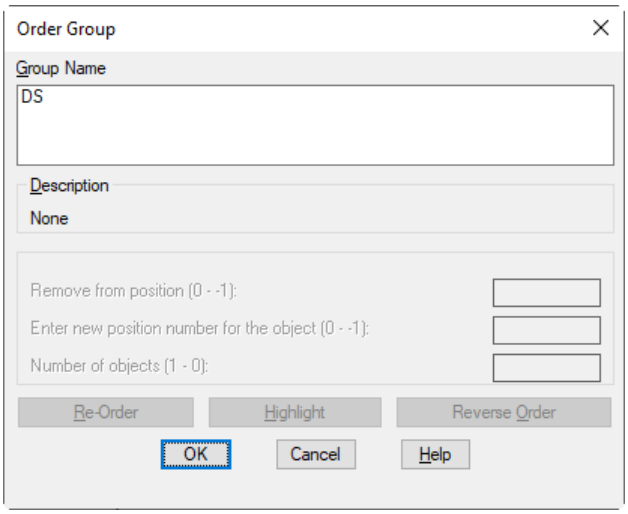

Figure 20-4 *The* ***Order Group*** *dialog box*

Description

The **Description** button is used to change the description of the selected group. To change the description of a group, first select the group name from the **Group Name** list box, enter the new description in the **Description** text box, and then choose the **Description** button. AutoCAD will update the description of the specified group.

Explode

The **Explode** option is used to delete group definition of the selected group. The objects that were in the group become regular objects without a group reference. If you had made copies of the group that was exploded in a drawing, they shall remain as unnamed groups in the drawing. You can select the **Include Unnamed** check box in the **Group Identification** area of the **Object Grouping** dialog box to view the names of the unnamed groups in the **Group Name** list box. You can then select them in the list box and explode them or rename them as required.

Selectable

The **Selectable** button is used to change the selectable status of the selected group. To change the selectable status of the group, first select the group name from the **Group Name** list box,

and then choose the **Selectable** button. This button acts like a toggle key between the options of selectable and non-selectable. If the selectable status for a group is displayed as "**Yes**", you can choose the **Selectable** button to change it to "**No**". Selectable implies that the entire group gets selected when one object in the group is selected. If the selectable status is "**No**", you cannot select the entire group by selecting one object in the group.

Example 1 *Reorder Group*

In this example, you will draw a rectangle representing a toolpath. The rectangle should comprise of four individual lines. Group all lines by selecting them in the order shown in Figure 20-5(a). Then, highlight the order, and re-order the group, as shown in Figure 20-5(b).

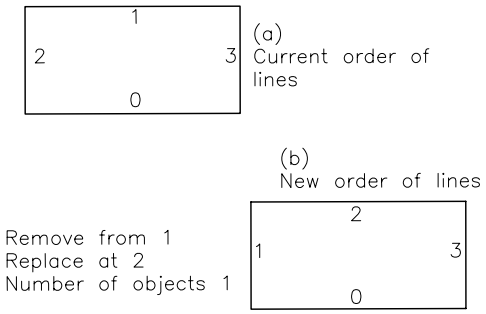

Figure 20-5 *Changing the order of group objects*

1. Draw four lines that are connected at the end points to form a rectangle.

2 Choose the **Group Manager** tool from the **Groups** panel; the **Object Grouping** dialog box is displayed.

3. Type **G1** in the **Group Name** edit box and choose the **New** button; the **Object Grouping** dialog box will disappear and you will be prompted to select the objects for grouping.

4. Select the lines in the order specified in Figure 20-5(a) and right-click; the **Object Grouping** dialog box is displayed again.

5. Make the group (G1) current by selecting it in the **Group Name** list box and choose the **Re-Order** button in the dialog box; the **Order Group** dialog box is displayed.

6. Select the group G1 in the **Group Name** list box, if not already selected, and choose the **Highlight** button; the **Object Grouping** dialog box is displayed, as shown in Figure 20-6. You can use the **Next** and **Previous** buttons to highlight the grouped objects in the order of selection.

   You will notice that the order of selection is not proper. As these lines represent a tool path, you need to reorder the objects, as shown in Figure 20-5(b).

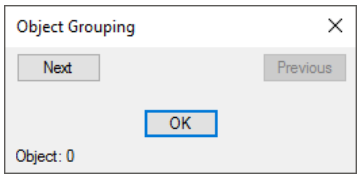

Figure 20-6 *The* ***Object Grouping*** *dialog box*

7. To get a clockwise tool path, you must switch object numbers 1 and 2. You can do so by entering the necessary information in the **Order Group** dialog box. Enter **1** in the **Remove from position (0-3)** edit box and **2** in the **Enter new position number for the object (0-3)** edit box. Enter **1** in the **Number of objects (1-4)** edit box because there is only one object to be replaced.

8. After entering the information, choose the **Re-Order** button to define the new order of the objects. You can confirm the change by choosing the **Highlight** button again and cycling through the objects.

GROUPING SKETCHED OBJECTS USING THE GROUP BUTTON

Ribbon: Home > Groups > Group **Command:** GROUP/G
Toolbar: Group > Group



In AutoCAD, you can group objects by using the **Group** tool from the **Groups** panel of the **Home** tab. The prompt sequence to create a group is given next:

*Choose the **Group** tool.*
Select objects or [Name/Description]: **N** Enter
Enter a group name or [?]: *Enter the group name* Enter
Select objects or [Name/Description]: *Select the objects to be grouped.*
Select objects or [Name/Description]: Enter
Group "*group name*" has been created.

You can also group the objects by using the shortcut menu. Select the objects to be grouped, and right-click in the drawing area; a shortcut menu will be displayed. Choose **Group > Group** from the shortcut menu; the selected objects will be grouped as *Unnamed group*.

Later on, you can ungroup them using **Ungroup** option from the same shortcut menu.

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** tool from the **Modify** panel in the **Home** tab and then enter the following prompt sequence:

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

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, choose the **Erase** tool from the **Modify** panel. The prompt sequence for erasing the group is given next.

Select objects: *Press 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 keyboard shortcut SHIFT+CTRL+A is used as a toggle key to turn the group selection on or off.

CHANGING PROPERTIES OF AN OBJECT

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

Using the PROPERTIES Palette

Ribbon: View > Palettes > Properties **Toolbar:** Standard > Properties
Quick Access Toolbar: Properties (*Customize to Add*) **Command:** PROPERTIES/PR/CH



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. The procedure to change the general properties of objects using the **PROPERTIES** palette has already been discussed earlier in Chapter 4. In this chapter, the **Geometry** category containing 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 drop-down list at the top of the **PROPERTIES** palette will display all the type of selected objects. If you select a type of object from the drop-down list, the corresponding categories of the object properties will be displayed in the palette. You can also invoke the **PROPERTIES** palette from the shortcut menu displayed on selecting the object.

Selecting a Line

If you have selected a line in the drawing, the **PROPERTIES** palette appears similar to what is shown in Figure 20-7, 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

choose the Pick Point button in the respective fields and select a new location for the endpoint of the selected line. The Pick Point button appears in the fields when you click on 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 a Circle

When you select a circle, the **Geometry** category displays the categories as shown in Figure 20-8. You can modify the values in these fields. The effects of modifying the values are discussed next.

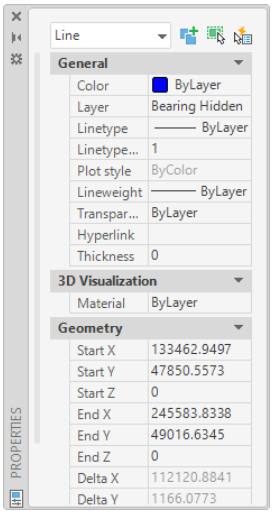

*Figure 20-7 The **PROPERTIES** palette for a line*

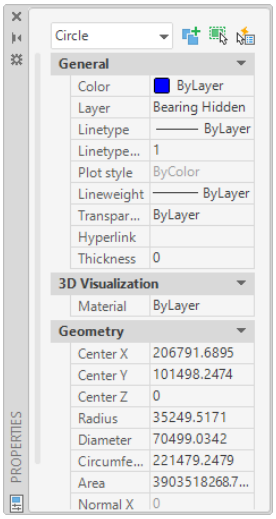

*Figure 20-8 The **PROPERTIES** palette for a circle*

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 specified before. It is as if the circle has moved from the old location to a new one.

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 the Multiline Text

If you select multiline text written in a drawing and invoke the **PROPERTIES** palette, refer to Figure 20-9, apart from the **General** category, three more categories of properties are displayed: **Text**, **3D Visualization**, and **Geometry**. The properties in the **Text** and **Geometry** categories and how they can be modified are discussed next.

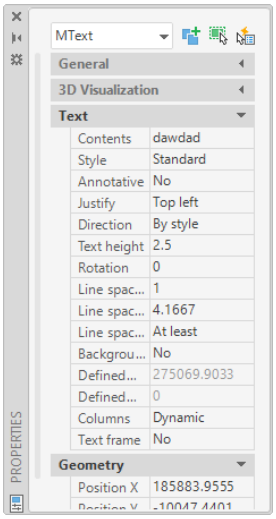

***Figure 20-9** The **Text** and **Geometry** categories of the **PROPERTIES** palette*

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 this button to display the **Text Editor** where you can make the necessary modifications using the options in the dialog box. Once you have made the changes, choose the **Close Text Editor** button from the **Text Editor** contextual tab 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.

Annotative: You can convert the non-annotative text to annotative and vice-versa by selecting the **Yes** and **No** options respectively from the **Annotative** drop-down list.

Justify: All the text justification options are available in the **Justify** drop-down list. This drop-down list will be 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.

Text height: You can enter a new value for the height of 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 **Text 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.

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.

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

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

Columns: Columns are of three types, None, Static, or Dynamic. To edit the column settings, choose the button; the **Column Settings** dialog box will be displayed. You can change the column settings through this dialog box.

Defined width/Defined height: These fields display the defined width/height of the text window in the **In-Place Text Editor**. These fields will be available, when you choose the **None** option from the **Columns** field. To change the values of these fields, click in the field and enter the new value in it.

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 the Annotative Block Reference

When you select a block in the drawing and invoke the **PROPERTIES** palette, apart from the **General** category, three more categories containing the properties of the block are available. They are **Geometry**, **3D Visualization**, and **Misc**. The properties under these categories are discussed next.

Position X/Position Y/Position Z: In the **Position X/Position Y/Position Z** fields of the **Geometry** category, you can enter new *X*, *Y*, and *Z* coordinates respectively 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.

Annotative: This field specifies whether the block is annotative or non-annotative. This information is read-only.

Annotative scale: This field displays the current annotation scale of the drawing. You can add more annotation scales to the drawing in the **Annotation Object Scale** dialog box that will be displayed when you choose the button from the **Annotative scale** field.

Match orientation to layout: This field displays whether or not the orientation of the text used in the block is forced to align with the orientation of the floating viewports. This information is read-only.

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 the Annotative 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**, **3D Visualization**, and **Misc**. Editing of an attribute has already been discussed in Chapter 17.

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.

Annotative: This field specifies whether the attribute is annotative or non-annotative. Select the appropriate option from the **Annotative** drop-down list to change the annotative property of the attributes.

Annotative scale: This field displays the current annotation scale of the drawing. You can add more annotation scales to the drawing in the **Annotation Object Scale** dialog box that will be displayed when you choose the button from the **Annotative scale** field.

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 **Justify** drop-down list.

Paper text height: This field displays the height of the annotative text that is to be maintained on the paper.

Model text height: This field displays the height of the text that is to be displayed in the model space. The **Model text height** value depends on the **Paper text height** value and the **Annotative scale**.

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

Rotation: You can rotate the selected attribute text by specifying an angle 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 angle value in the **Obliquing** field of the **PROPERTIES** palette.

Direction: This field displays the flow direction of the attribute text.

Boundary Width: This field displays the width of the text window in the **In-Place Text Editor**. The value displayed here is the relative value, specifying a value less than **1** will decrease the boundary width and a value more than **1** will increase the boundary width. The value in the **Boundary Width** will be available only if the **Multiple lines** check box is selected in the **Mode** area of the **Attribute Definition** dialog box. You can also select the **Yes** option from the **Multiple lines** drop-down list in the **Misc** area 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 **Text** 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 this attribute, the remaining attributes will also have the ***Text Alignment*** *fields active.*

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 the ***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.*

Exercise 1 *Properties*

Draw a hexagon on OBJ layer in red color. Let the linetype be hidden. Now, use the **PROPERTIES** palette to change the layer of the hexagon 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 lineweight to each of the hexagon segments between the specified vertices. Use the **Tools > Inquiry > List** tool from the menu bar to verify that the changes have taken place.

EXPLODING COMPOUND OBJECTS

Ribbon: Home > Modify > Explode **Toolbar:** Modify > Explode
Menu Bar: Modify > Explode **Command:** EXPLODE/X

The **Explode** tool 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, refer to Figure 20-10. 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. Polyface meshes are turned into 3D faces, points, and lines. Upon exploding 3D solids, the planar surfaces of the 3D solid turn into regions, and non-planar surfaces turn into bodies, multilines are changed to lines. Regions turn into lines, ellipses, splines, or arcs. On exploding, 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 a single line text. This tool is especially useful when you have inserted an entire drawing and you need to alter a small detail. After you invoke the **Explode** tool, 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.

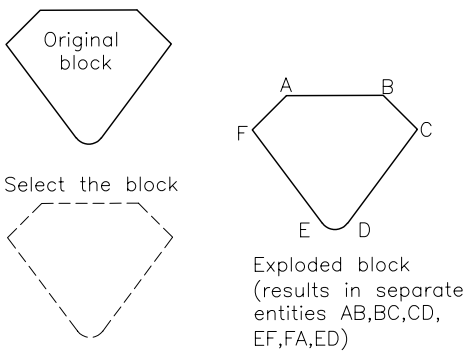

Figure 20-10 *Use of the* ***Explode*** *tool*

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** tool, 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 blocks inserted using the **MINSERT** command and x-refs cannot be exploded.*



Tip

*If you want to insert a block in the form of its separate components while using the **INSERT** tool, select the **Explode** check box in the **Insert** dialog box. Also, if you are using the **-INSERT** command, type * prior to the block name at the **Enter block name or [?]:** prompt, the block will be inserted in the drawing as separate component object, 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 **Edit Polyline** tool. In this section, you will learn how to edit simple 2D polylines. The following are the editing operations that can be performed on an existing polyline using the **Edit Polyline** tool. These operations 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. After invoking the **Edit Polyline** tool, if you select an entity that is not a polyline, 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 Single Polyline

Ribbon: Home > Modify > Edit Polyline **Toolbar:** Modify II > Edit Polyline
Menu Bar: Modify > Object > Polyline **Command:** PEDIT/PE



You can edit a single polyline by using the **Edit Polyline** tool. To invoke this tool, choose the **Edit Polyline** tool from the expanded **Modify** panel of the **Home** tab; the following prompt sequence is displayed:

Select polyline or [Multiple]:

If the selected entity is not a polyline, the following message will be displayed at the command prompt.

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 or by simply pressing ENTER. To let the object be as it is, enter N. If you enter N, then AutoCAD will 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 provides you the option of either selecting a single polyline or multiple polylines. In this case, a single 2D polyline is selected, and its prompt sequence is given next.

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



Note

*Depending on the type of polyline selected, the options of **Edit Polyline** tool get changed. In this chapter, the **2D polyline** options have been discussed.*

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

Close (C) Option

This option is used to close an open polyline. The **Close** option 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 20-11 illustrates this option.

Open (O) Option

If you close an open polyline, the **Close** option will be replaced by the **Open** option. Enter **O** for open; the closing segment will be removed, refer to Figure 20-11.

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, refer to Figure 20-12. This option can be used only if a polyline is open. After this option has been selected, AutoCAD prompts you to select objects. Once you have chosen the objects to be joined to the original polyline, AutoCAD 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 depends upon the sequence of selecting the lines. If you select the two lines, the first selected line will get joined to the polyline.

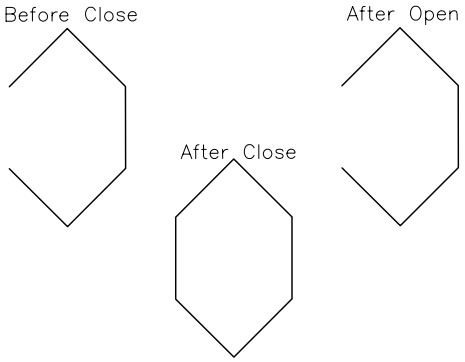

Figure 20-11 *The **Close** and **Open** options*

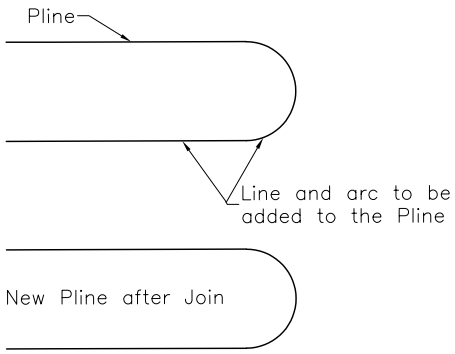

Figure 20-12 *Using the **Join** option*

Width (W) Option

The **W** option allows you to define a new, unvarying width for all segments of a polyline, refer to Figure 20-13. 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 using 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 figure from 0.20 to 0.5, refer to Figure 20-14. On invoking the **Edit Polyline** tool, the following prompt is displayed:

Select polyline or [Multiple]: *Select a polyline.*
Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Reverse/Undo]: **W** Enter
Specify new width for all segments: **0.5** Enter

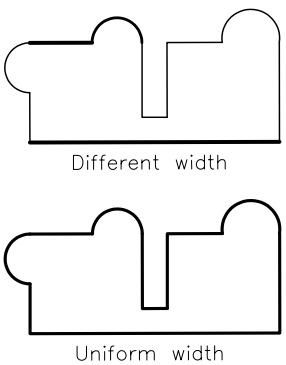

Figure 20-13 *Making the width of a polyline uniform*

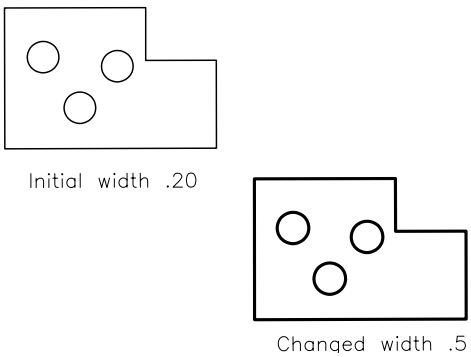

Figure 20-14 *Changing the width of all segments*



Note

*Circles drawn using the **Circle** tools cannot be changed to polylines. However, for polycircles drawn by using the **Arc** option of the **Polyline** tool (by drawing two semicircular polyarcs) or by using the **Donut** tool, you can modify the thickness using the **Width** option of the **Edit Polyline** tool.*

Edit vertex (E) Option

The **Edit vertex** option allows you select 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 **Edit Polyline** tool, 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/Reverse/Undo]: **E** Enter
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 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 the first and last vertices, but cannot move past the first or last vertex, even if the polyline is closed, refer to Figure 20-15.

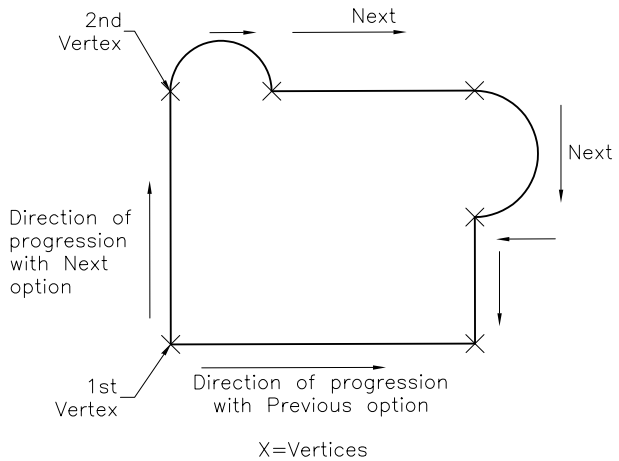

***Figure 20-15** The **Next** and **Previous** options*

The prompt sequence for using the **Edit vertex** option, after you invoke the **Edit Polyline** tool, 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/Reverse/Undo]: **E** Enter
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 20-16 or break it at a single point. You can specify the vertices within which the polyline would be removed. 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 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 the endpoint (last vertex) of the polyline. 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, which acknowledges this by displaying the message ***Invalid***.

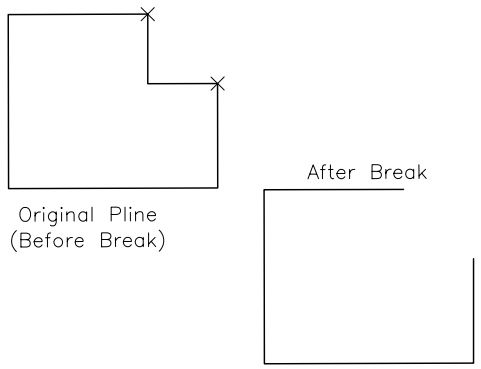

Figure 20-16 *Using the* ***Break*** *option*

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 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 **G** 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 **G**. On doing so, the polyline segments between the two selected vertices will be erased. The prompt sequence for using this option, after you have invoked the **Edit Polyline** tool, 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/Reverse/Undo]: **E** Enter

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** Enter

Once you invoke the **Break** option, AutoCAD 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 is used to define a new vertex and add it to the polyline, refer to Figure 20-17. 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 invoke the **Edit Polyline** tool, the prompt sequence for using the **Insert** option will be 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/Reverse/Undo]: **E** Enter
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** Enter
Specify location for new vertex: *Move the cursor and click to specify the location of the new vertex or enter the coordinates of the new location.*

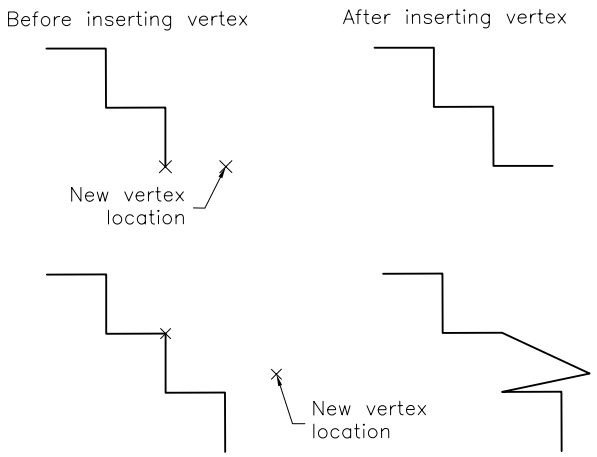

Figure 20-17 *Using the* ***Insert*** *option to define new vertex points*



Tip

*In AutoCAD, you can insert vertex by using the shortcut menu. To do so, select a polyline; the grips will be displayed. Hold the cursor on one of the grips; a shortcut menu will be displayed. Choose the **Add Vertex** option and click on the drawing area to add new vertex. You can also stretch the selected arc or convert the line to arc or vice versa by selecting appropriate option from the shortcut menu.*

Move option: This option is used to move the X-marked vertex to a new position, refer to Figure 20-18. Before invoking the **Move** option, you must move the X marker to the vertex you want to relocate by selecting the **Next** or **Previous** option. The prompt sequence for relocating a vertex after you invoke the **Edit Polyline** tool 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/Reverse/Undo]: **E** Enter
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** Enter
Specify new location for marked vertex: *Specify the new location for the selected vertex by 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.*

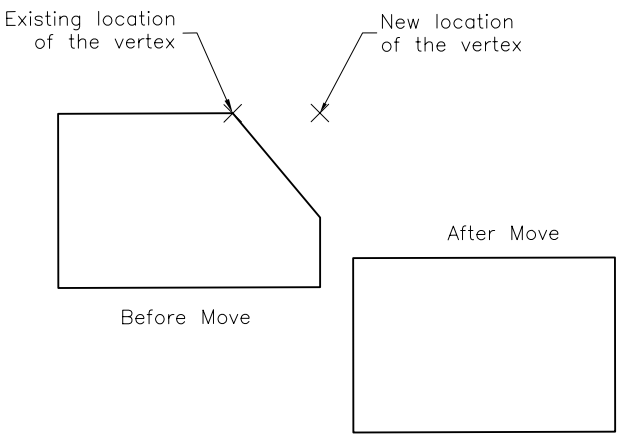

***Figure 20-18** The **Move** option*

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 is used to straighten polyline segments or arcs between specified vertices, refer to Figure 20-19. 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 invoke the **Edit Polyline** tool 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/Reverse/Undo]: **E** Enter
Enter a vertex editing option [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit]<N>: *Move the 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
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** Enter

The polyline segments between 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.

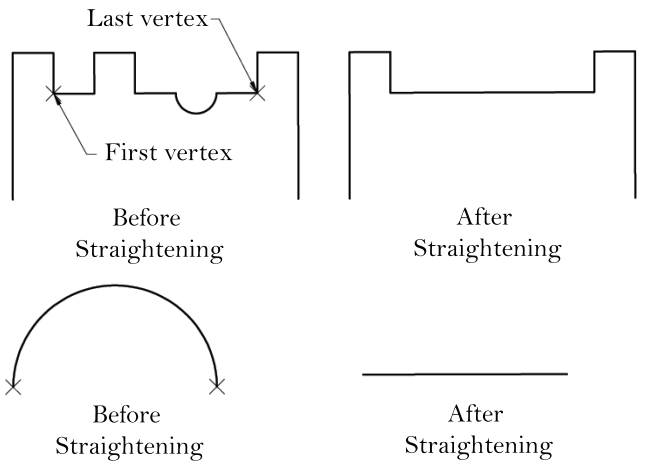

Figure 20-19 *Using the* ***Straighten*** *option to straighten polylines*

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 **Edit Polyline** tool. 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 allows you to change the width at the start and end points of a polyline segment that follows the current vertex, refer to Figure 20-20. 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 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/Reverse/Undo]: **E** Enter
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** Enter
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.*

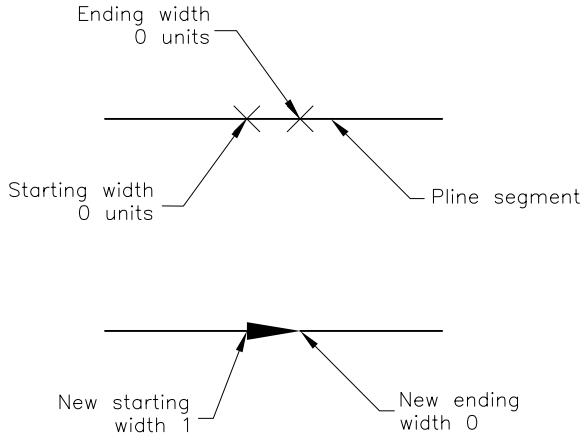

Figure 20-20 *Using the* ***Width*** *option to change the width of a polyline*

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 **Edit Polyline** prompt.

Fit Option

The **Fit** or **Fit curve** option is used to generate a curve that passes through all the corners (vertices) of the polyline, using the tangent directions of the vertices, refer to Figure 20-21.

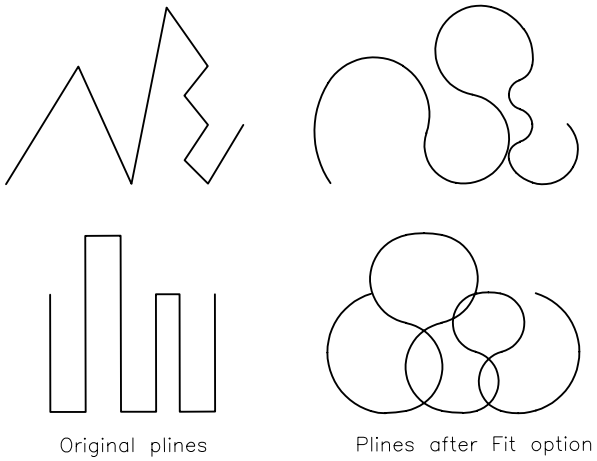

Figure 20-21 *Curves generated using the* ***Fit*** *option*

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 you 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, this option is known as **Fit**. The vertices of the polyline are also known as the control points. The closer these control points are, the smoother is 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 **Edit Polyline** tool.

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

If the tangent directions need to be edited, use the **Edit vertex** option of the **Edit Polyline** tool. 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 to use the **Tangent** option of the **Edit Polyline** tool 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/Reverse/
Undo]: **E** Enter
Enter a vertex editing option [Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/
Width/eXit]<N>: **T** Enter
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.



Tip

*In AutoCAD, after applying the **Fit** option to a polyline, you can change the tangent direction by using the tool tip. To do so, select a polyline; the grips will be displayed. Move the cursor near one of the grips; a tooltip will be displayed. Choose the **Tangent Direction** option and specify the new direction in the drawing area.*

Spline Option

The **Spline** option, refer to Figure 20-22, 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 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 on invoking the **Edit Polyline** tool is as follows:

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

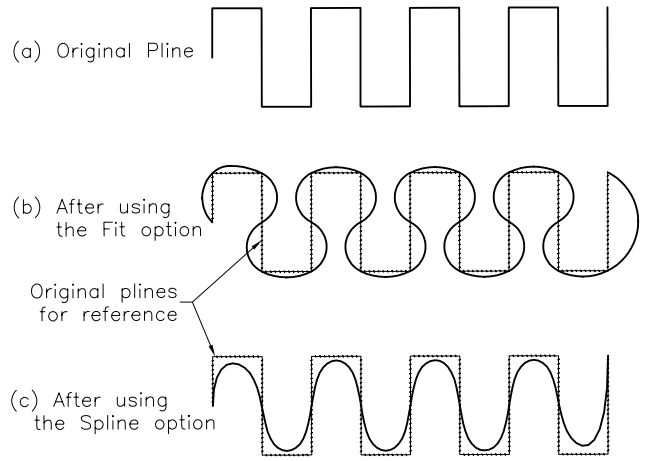

Figure 20-22 *Smoothening the corners using the **Spline** option*

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 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 tools 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** tool 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 tools 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** tools delete the frame. The **Divide**, **Measure**, **Fillet**, **Chamfer**, **Area**, and **Hatch** tools recognize only the spline curve and do not consider the frame. The **Stretch** tool first stretches the frame and then fits the spline curve to it.

When you use the **Join** option of the **Edit Polyline** tool, the spline curve is decurved and the original spline information is lost. The **Next** and **Previous** options of the **Edit vertex** option of the **Edit Polyline** tool move the marker only to the points on the frame, whether visible or not. The **Break** option discards the spline curve. The **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** and **Cubic B-spline**. These types of curves pass through the first and the last control points, which is the 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, refer to Figure 20-23.

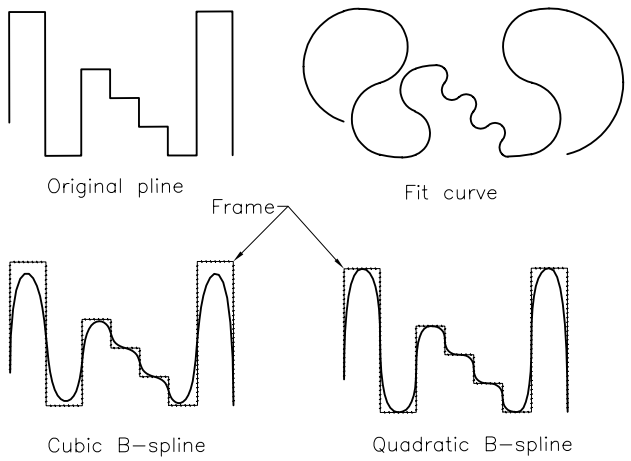

Figure 20-23 *Comparison of fit curve, quadratic B-spline, and cubic B-spline*

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 **Edit Polyline** tool, a cubic curve is produced by default. You can change the value of the **SPLINETYPE** variable using the command prompt.

In case you are editing 3D polygon meshes, the smoothness of the meshes is controlled by the **SURFTYPE** variable. The value of the variable can be set on the command prompt. You can select the type of polygon mesh you want when you are using the **Smooth Surface** option of the **Edit Polyline** tool.

SPLINESEGS. The system variable **SPLINESEGS** governs the number of line segments used to construct the spline curves, and therefore 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 will be the curve. But greater the regeneration time, the more space will be occupied by the drawing file.

Figure 20-24 shows cubic curves with different values for the **SPLINESEGS** variable.

Decurve Option

The **Decurve** option straightens the curves generated after using the **Fit** or **Spline** option on a polyline. On using this option, the curves return to their original shape, refer to Figure 20-25. 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 tool to straighten out any curve drawn with the help of the **Arc** option of the **Polyline** tool. Enter **D** at the **Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Reverse/Undo]** prompt to invoke the **Decurve** option of the **Edit Polyline** tool.

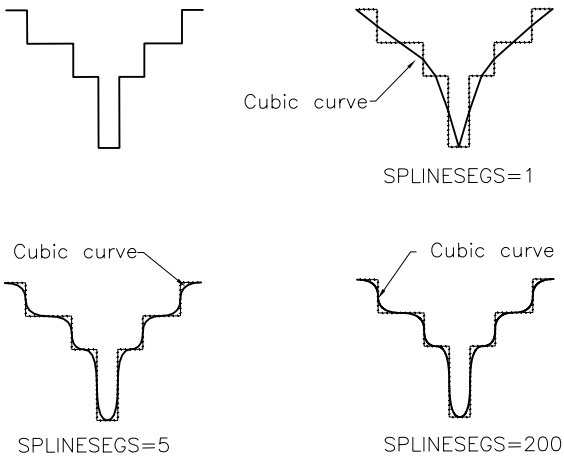

Figure 20-24 *Using the* ***SPLINESEGS*** *variable*

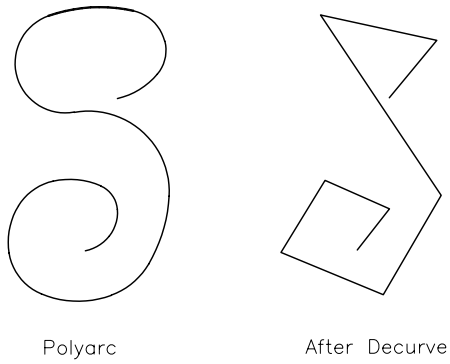

Figure 20-25 *Using the* ***Decurve*** *option*

Note

If you use editing tools such as the ***Break*** *or* ***Trim*** *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, refer to Figure 20-26. 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.

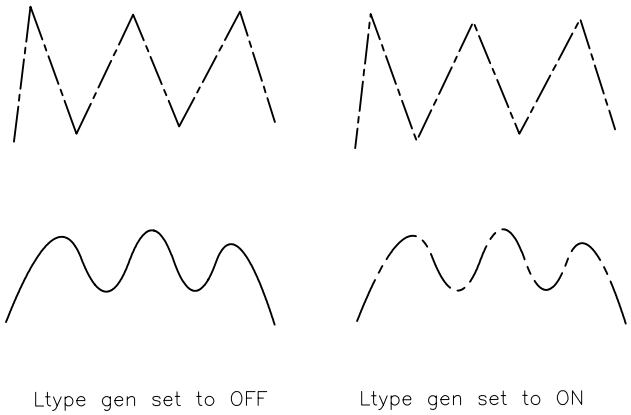

Figure 20-26 *Using the* ***Ltype gen*** *option*

The command prompt displayed when you select a polyline and invoke the **Ltype gen** option of the **Edit Polyline** tool is given next.

> 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 function. The default value of this variable is off. Entering the value **ON** turns on the **Ltype gen** option. Alternatively, select the appropriate option in the **Linetype generation** field in the **Misc** area of the **PROPERTIES** palette with the polyline selected.

Reverse Option

The **Reverse** option is used to change the start point of the polyline. On entering **R** and pressing ENTER at the **Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Reverse/Undo]** command prompt, the start point of the polyline will be changed. This can be confirmed by using the **Edit vertex** option of the **Edit Polyline** tool before and after using the **Reverse** option.

Undo (U) Option

The **Undo** option negates the effect of the most recent polyline editing operation. You can go back as far as you need to in the current polyline editing 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 **Edit Polyline** tool will not work. In this case, you will have to exit to the command prompt and use the **Undo** tool to undo the operation.

Editing Multiple Polylines

Selecting the **Multiple** option of the **Edit Polyline** tool 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/Reverse/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 prompt is as follows.

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 is used to extend or trim the selected polylines at the endpoints that are nearest to each other, refer to Figure 20-27. 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 is used to join the selected polylines by drawing a straight line between the nearest endpoints, refer to Figure 20-28. The fuzz distance should be greater than the actual distance between the two endpoints to be joined.

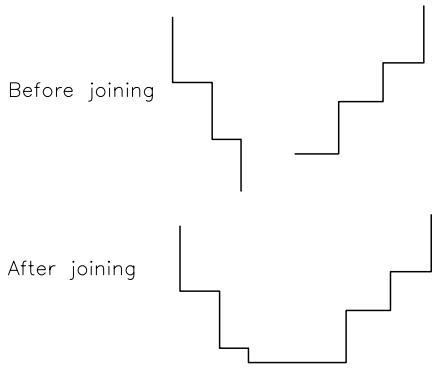

***Figure 20-27** Joining two polylines using the **Extend** option*

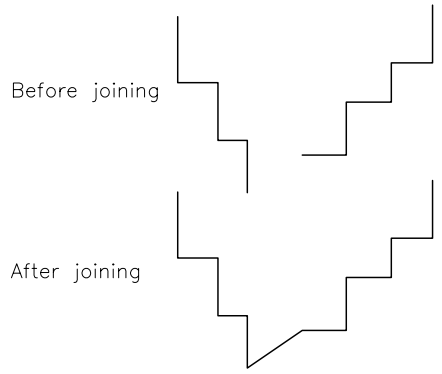

***Figure 20-28** Joining two polylines using the **Add** option*

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

Exercise 2 *Edit Polyline*

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 an 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 *Edit Polyline*

a. Draw a staircase-shaped polyline and use the different options of the **Edit Polyline** tool 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

Quick Access Toolbar: Undo **Toolbar:** Standard > Undo **Command:** UNDO/U



In AutoCAD, there are many tools with the **Undo** option, which can be used to undo (nullify) the changes made within these commands. The **Undo** option is used to undo a previous command or to undo more than one command at a time. This command can be invoked by entering **UNDO** at the command prompt. **Undo** is also available in the **Quick Access Toolbar**, the **Edit** menu, and the **Standard** toolbar. The **Undo** tool in the **Quick Access Toolbar** can only undo the previous command and only one command at a time. If you right-click in the drawing area; a shortcut menu will be 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 displays the message: **Nothing to undo**. When an operation cannot be undone, a message is displayed but no action takes place. External commands such as **PLOT** cannot be undone.

Undoing Previous Commands using the Quick Access Toolbar

Click the arrow on the **Undo** tool in the **Quick Access Toolbar**; the **Undo** list box will be displayed. All commands used recently will be 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 20-29. The Status Bar below the **Undo** list box displays the number of commands selected. Click the left mouse button to confirm the selections.

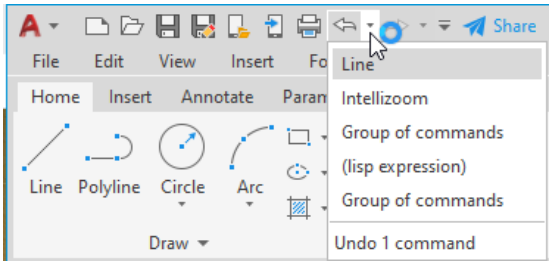

Figure 20-29 *The* ***Undo*** *list box*

Undoing Previous Commands using the Command Prompt

To Undo previous commands using the command Line, enter **UNDO** at the command prompt (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.*

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 allows you to 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 Enter
PLINE LINE CIRCLE

Control (C) Option

This option allows you to determine how many of the options are active for the **Undo** tool. You can disable the options you do not need. With this option, you can even disable the **Undo** tool. To access this option, type C. You will get the following prompt.

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

ALL

The **All** option activates all the features (options) of the **Undo** tool.

None (N)

This option turns off the **Undo** tool. If you have used the **BEgin** and **End** options or the **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
Enter an UNDO control option [ALL/None/One/Combine/Layer] <All>: **N** Enter

If you try to use the **Undo** tool now, AutoCAD gives you the following message.

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/Combine/Layer] <All>: *Enter O or press ENTER to accept the default option.*

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
Enter an UNDO control option [All/None/One/Combine/Layer] <All>: **O** Enter

If you then enter the **UNDO** command, you will get the prompt that is given next.

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 acknowledges undoing the previous command with messages given next.

Command: **UNDO**
Enter an option [Control] <1>: Enter
CIRCLE
Everything has been undone



Tip

*AutoCAD stores all information about all the **UNDO** operations performed in a drawing session. When you use the **None** control option of the **UNDO** command, this information is removed and the disk space is made available. In case of limited disk space, it may be a good idea to use the **One** suboption of the **UNDO Control** command. Using **One** control option, you can **UNDO** commands and at the same time make the disk space available.*

Combine

This option allows you to combine all the consecutive **Zoom** and **Pan** operations as a single operation while using the **UNDO** and **REDO** commands.

Layer

This option allows you to combine layer property actions as a single action.

Auto (A) Option

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

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

This option is turned on by default and every menu item is a single operation. When you use the **Begin** and **End** 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 turned on. But if the **Auto** option is turned off, entering **U** at the command prompt will remove the entire group of objects you had grouped using the **Begin** and **End** 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 **Back** 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 turned off, entering **U** at the command prompt, or using the **Back** option of the **UNDO** command, undoes the entire group of operations irrespective of the marker being there.

BEgin (BE) and End (E) Options

A group of commands is treated as one command for the **U** and **UNDO** commands (provided the **Auto** option is turned off) by embedding the commands between the **BEgin** and **End** 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 **BEgin** option and before the **End** option are treated as a single command by the **U** command (if the **Auto** option is turned 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/BEgin/End/Mark/Back] <1>: **BE** Enter

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: *Select the first point.*
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: *Select the next point.*
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: *Press ENTER.*
Command: **UNDO**
Enter the number of operations to UNDO or [Auto/Control/BEgin/End/Mark/Back]: **E** Enter
Command: **U**

To start the next group after having finished specifying the current group, use the **End** option to end the operation. Another method is to enter the **BEgin** option to start the next group while the current group is active. This is equivalent to issuing the **End** option followed by the **BEgin** option. The group is complete only when the **End** option is invoked to match a **BEgin** 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.

Mark (M) and Back (B) Options

The **Mark** option installs a marker in the Undo file. The **Back** option allows you to 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 **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** Enter

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** Enter

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 displays the command names as they are undone.

REVERSING THE UNDO OPERATION

Quick Access Toolbar: Redo **Toolbar:** Standard > Redo **Command:** REDO

If you right-click in the drawing area, a shortcut menu is displayed with **Redo(x)** as one of the commands,where x represents the last used command.Choose the **Redo(x)** command to redo the undo operation. The **REDO** command brings back the process you removed previously using the **U** and **UNDO** commands. This command undoes the last **UNDO** command performed, provided it is entered immediately after the **UNDO** command. On using this command, the objects previously undone reappear on the screen. Click on the **Redo** button in the **Quick Access Toolbar**; the **Redo** drop-down is displayed, refer to Figure 20-30. 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.

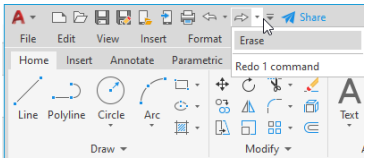

Figure 20-30 *The **Redo** drop-down*

Note

*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 Bar: 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 HID to HIDDEN, the process will be as follows:

1. Enter **RENAME** at the command prompt and then press ENTER; the **Rename** dialog box will be displayed, as shown in Figure 20-31.

2. Select **Layers** in the **Named Objects** list box, if not already selected. All the layer names in the current drawing that can be renamed are displayed in the **Items** list box.

3. Select **HID** in the **Items** list box to highlight it. HID will be displayed in the **Old Name** edit box.

4. Enter **HIDDEN** in the **Rename To** edit box, and choose the **OK** button.

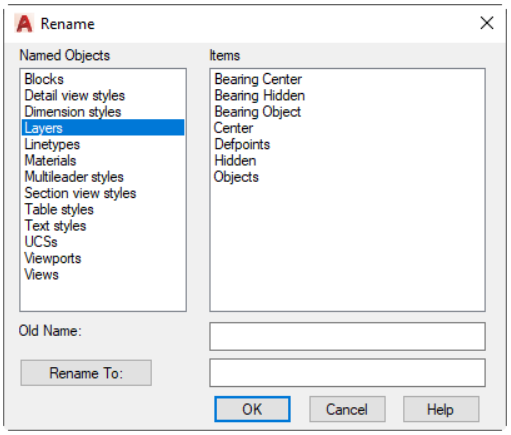

Figure 20-31 *The* ***Rename*** *dialog box*

Now, the layer named **HID** will be renamed to **HIDDEN**. You can view the change in the **Layer** drop-down list in the **Layers** panel of the **Home** tab or invoke the **LAYER PROPERTIES MANAGER** to notice this change. You can rename blocks, dimension styles, linetypes, styles, UCS, views, and viewports in the same way.



Note

1. The ***RENAME*** *command cannot be used to rename drawing files created using the* ***Insert > Block Definition > Create Block > Write Block*** *method. You can change the name of a drawing file by using the* ***Rename*** *option of your Operating System.*

2. 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

Application Menu: Drawing Utilities > Purge
Menu Bar: 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, shapes, and so on with the help of the **PURGE** tool. When you create a new drawing or open an existing one, AutoCAD 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** tool, you can select the named objects you want to delete. You can invoke this tool at any time in the drawing session. On invoking the **PURGE** command, the **Purge** dialog box is displayed, as shown in Figure 20-32. The options in this dialog box are discussed next.

Purgeable Items Tab

In this tab, AutoCAD displays the tree view of all the named items that are in the current drawing and can be purged. By default, this tab is chosen, and the **Named Item Not Used** and **Options** areas are displayed in the **Purge** dialog box, refer to Figure 20-32. The options in these areas are discussed next.

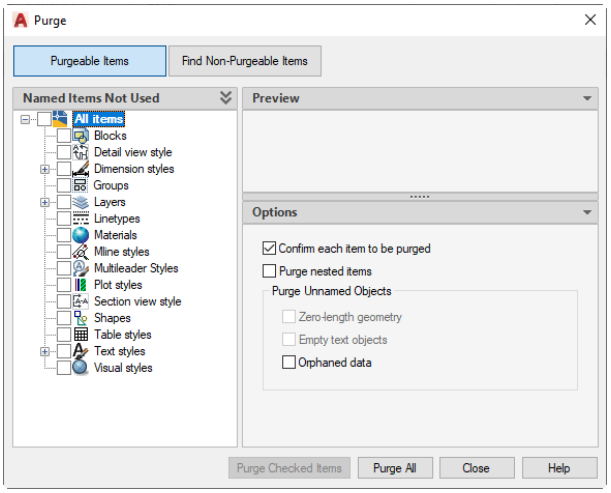

Figure 20-32 *The* ***Purge*** *dialog box with purgeable items*

Named Item Not Used

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 the items to be purged. You can select more than one item by holding down the SHIFT or the CTRL key.

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** button is chosen. You can confirm or skip the items to be purged in this dialog box.

Purge nested items

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

Purge zero - length geometry and empty text objects

While drawing a line, arc, or polyline if the same point is selected as the start and the end point by mistake, then a zero length geometry will be created. Also, if there is only empty space created by mistake while creating a text, then the **Purge zero-length geometry and empty text objects** check box will be available. You can purge the zero length geometry or the empty spaces by selecting this check box.

Find Non-Purgeable Items Tab

In this tab, AutoCAD displays the tree view of all the named items that are in the current drawing and cannot be purged, refer to Figure 20-33. On choosing this tab, the items in the **Named Items in Use** area are displayed. When you select an item from this area, the information about why the selected item cannot be purged is displayed in the **Possible Reasons** area.

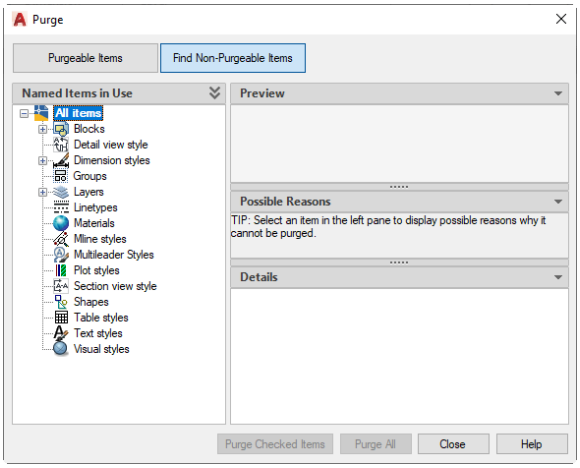

Figure 20-33 *The **Purge** dialog box with non-purgeable items*



Note

*1. The **Entire Drawing** option of the **Write Block** dialog box, which is invoked on using the **WBLOCK** command or the **-WBLOCK** option, has the same effect as the **PURGE All** command. The only difference is that in this case the unused named objects are removed automatically without any verification, though this method is much faster as compared to the previous one.*

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

SETTING SELECTION MODES USING THE OPTIONS DIALOG BOX

Application Menu: Options **Command:** OPTIONS/OP

When you select a number of objects, the selected objects form a selection set. You can specify the objects to be selected in the **Options** dialog box which is invoked by choosing the **Options** button from the **Application Menu**, as shown in Figure 20-34. Five 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.

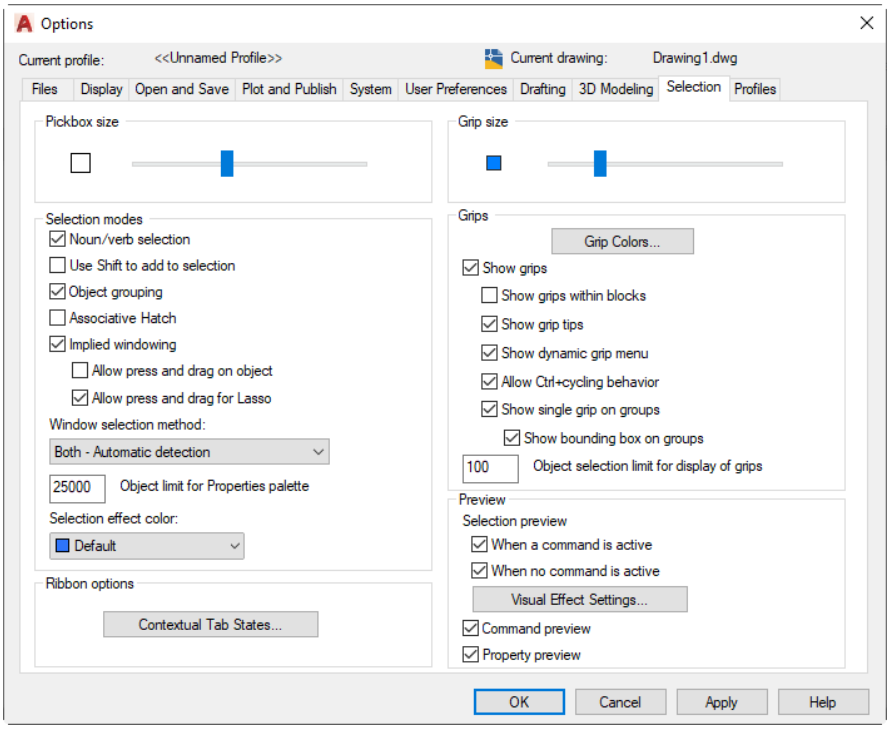

Figure 20-34 *The* ***Selection*** *tab of the* ***Options*** *dialog box*

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 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.

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.

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 prompts 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; and 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).

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.

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. 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 50. The default size is 3. You can also use the **PICKBOX** system variable.

Window selection method

This drop-down list can be used to control the selection method. During the window selection, you can control the window behaviour in the following two ways:

Click and Click. Using this method, you can define the selection window by clicking in the screen to specify the start point and endpoint of the window.

Press and drag. Using this method, you can define the selection window by clicking on any position in the drawing screen and dragging the cursor according to requirement.

You can also use both options by selecting **Both - Automatic detection** from the **Window selection method** drop-down list.

Self-Evaluation Test

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

1. 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.

2. If the **Press and Drag** mode is not active, you have to select _____ to specify the two diagonal points of a selection window.

3. The _____ option in the prompt of the **Edit Polyline** tool can be used to change the starting and ending widths of a polyline separately to a desired value.

4. The _____ option of the **Edit Polyline** tool is used to select more than one polyline.

5. You can move the first and last vertices in a closed polyline by using either the _____ option or the _____ option.

6. While using the _____ tool, if you select a line or an arc, AutoCAD provides you with the option of converting them into a polyline first.

7. Even if a group is defined as selectable and the **PICKSTYLE** system variable is set to 0, you cannot select the entire group by selecting one of its members. (T/F)

8. The **PROPERTIES** palette is used to change the properties associated with an object. (T/F)

9. The **PROPERTIES** palette can be used to modify the geometry of objects apart from their general properties. (T/F)

10. The **RENAME** command can be used to change the name of a drawing file created by using the **Write Block** tool. (T/F)

Review Questions

Answer the following questions:

1. In which of the following notations will the name of a group be displayed in the **Group Name** list box if you make a copy of the group without naming it?

   (a) $A1 (b) @A1
   (c) %A1 (d) *A1

2. Which of the following options is used to delete a selected group from a drawing?

   (a) **Remove** (b) **Reorder**
   (c) **Explode** (d) **Rename**

3. Which of the following system variables controls the smoothness of a curve?

   (a) **SPLINETYPE** (b) **SPLINESEGS**
   (c) **PLINEGEN** (d) **PICKFIRST**

4. Which of the following options of the **Edit Polyline** tool can straighten a curve drawn with the help of the **Polyline** tool?

   (a) **Join** (b) **Close**
   (c) **Decurve** (d) None of these

5. Which of the following properties cannot be changed using the **CHANGE** command?

   (a) **Lineweight** (b) **Color**
   (c) **Plotstyle** (d) **Thickness**

6. The _____ command can be used to change the name of a block.

7. If the _____ option of the **UNDO** command is set to off, any group of commands grouped together using the **Begin** and **End** options of the **UNDO** command are undone together.

8. Circles drawn using the **Circle** tool can be changed to _____.

9. The _____ option in the **Group** tool is used to change the order of the objects in the selected group.

10. You can select a group by entering _____ at the command prompt.

11. The _____ option of the **UNDO** command disables the **UNDO** and **U** commands entirely.

12. When you use the **Group** tool to form object groups, AutoCAD allows you to sequentially highlight the groups of which the selected object is a member. (T/F)

13. The standard objects created by AutoCAD (such as layer 0, STANDARD text style, and linetype CONTINUOUS) cannot be removed by using the **PURGE** tool. (T/F)

14. If you explode an object, it remains identical to its original geometry; only the color and linetype may change because of floating layers, colors, or linetypes. (T/F)

15. The **PURGE** tool has the same effect as that of the **WBLOCK Entire drawing** or the **-WBLOCK** method. The only difference is that with the **PURGE** tool, deletion takes place automatically. (T/F)

Exercise 4 — Properties and Edit Polyline

Draw part (a) in Figure 20-35 and then using the **Properties** and relevant **Edit Polyline** tool options, convert it into parts (b), (c), and (d). The linetype used in part (d) is HIDDEN.

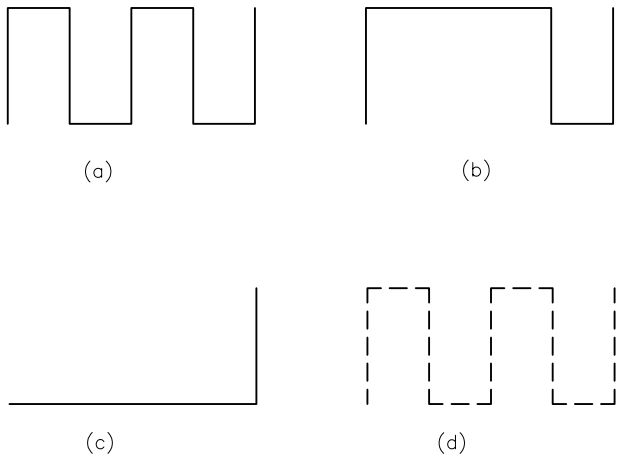

Figure 20-35 *Drawing for Exercise 4*

Exercise 5 — Edit Polyline

Draw the sketch shown in Figure 20-36 using the **Line** tool. Change the object to a polyline with a width of 0.05.

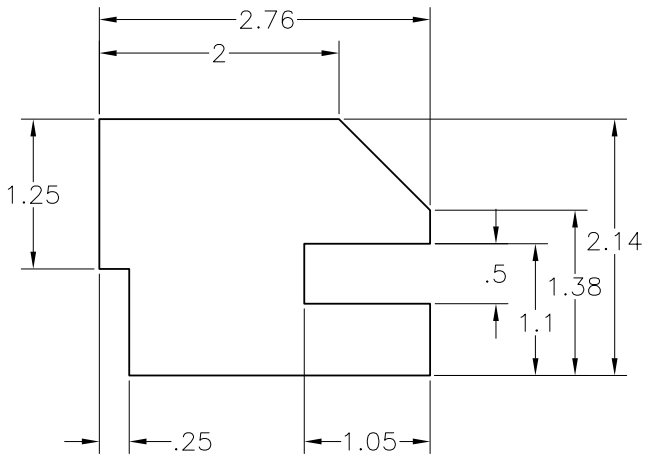

Figure 20-36 *Drawing for Exercise 5*

Exercise 6 *Edit Polyline*

Draw part (a) of Figure 20-37 and then using the relevant **Edit Polyline** tool options, convert it into drawing (b).

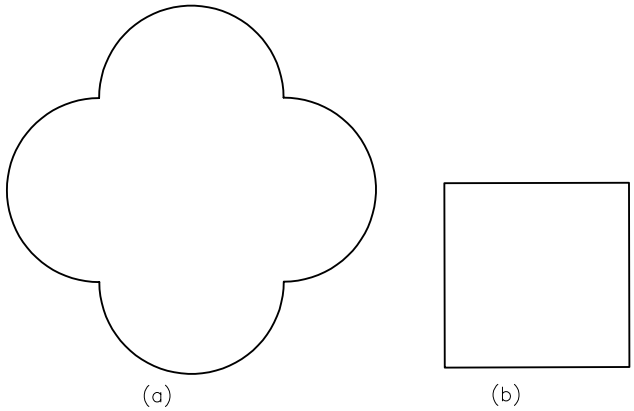

Figure 20-37 *Drawing for Exercise 6*

Exercise 7 *Edit Polyline*

Draw part (a) of Figure 20-38 and then using the relevant **Edit Polyline** tool options, convert it into drawings (b), (c), and (d). Identify the types of curves.

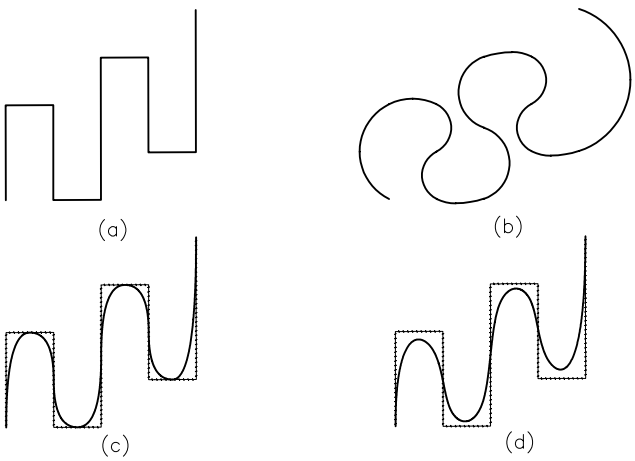

Figure 20-38 *Drawing for Exercise 7*

Exercise 8 *Edit Polyline*

Draw part (a) of Figure 20-39 and then using the relevant **Edit Polyline** tool options, convert it into drawing (b).

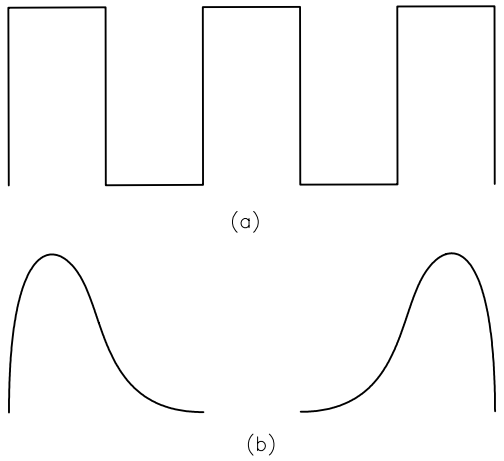

Figure 20-39 *Drawing for Exercise 8*

Exercise 9 *Edit Polyline*

Draw the part (a) of Figure 20-40 and change it into the figure shown in part (b) by using the appropriate commands. You can also use grips to obtain this illustration.

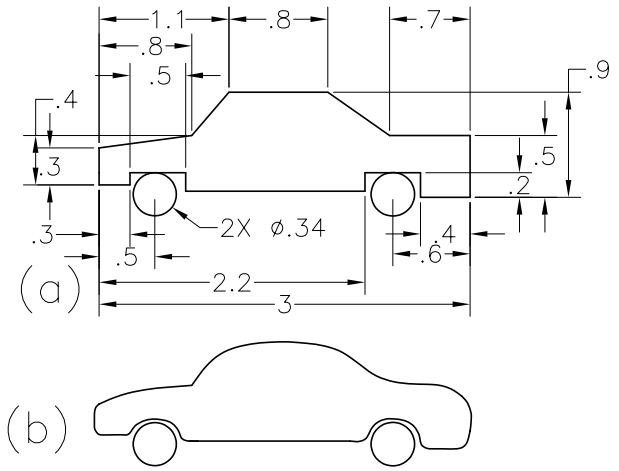

Figure 20-40 *Drawing for Exercise 9*

Exercise 10 *Edit Polyline*

Create the drawing shown in Figure 20-41. Use the splined polylines to draw the break lines. Dimension the drawing as shown.

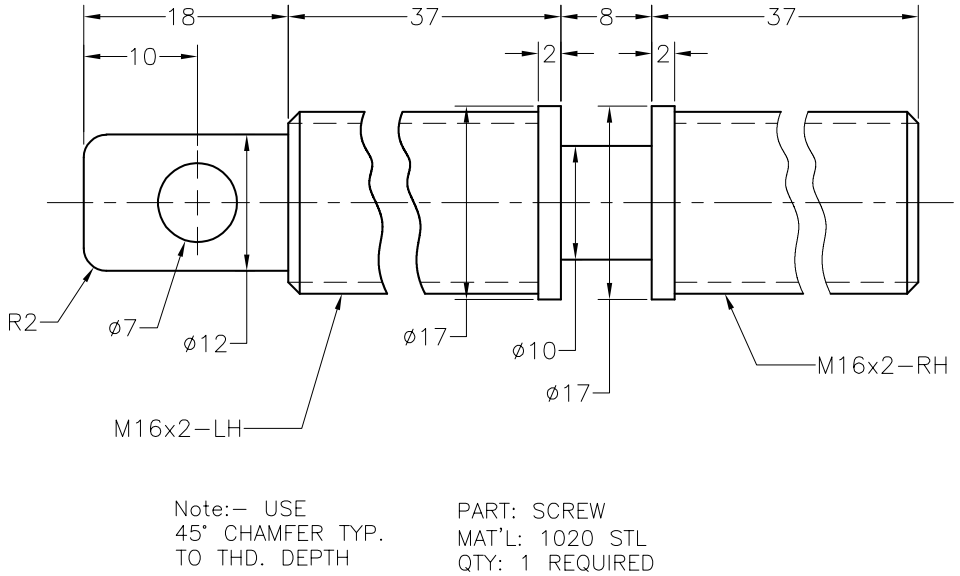

Figure 20-41 *Drawing for Exercise 10*

Answers to Self-Evaluation Test

1. Begin, **End**, **2.** Twice, **3.** Width, **4. Multiple**, **5. Next**, **Previous**, **6. Edit Polyline**, **7.** T, **8.** T, **9.** T, **10.** F