



Chapter 3

Editing, Extruding, and Revolving Sketches

Learning Objectives

After completing this chapter, you will be able to:

- *Edit sketches using various editing tools.*
- *Create rectangular and circular patterns.*
- *Write text in the sketching environment and convert it into feature.*
- *Insert external images, Word documents, and Excel spreadsheets in the sketching environment.*
- *Convert sketches into base features using the Extrude tool.*
- *Convert sketches into base features using the Revolve tool.*
- *Dynamically rotate the view of a model in 3D space and use the existing common views to view the model from various directions.*

EDITING SKETCHED ENTITIES

Autodesk Inventor provides you with a number of tools that can be used to edit the sketched entities. These tools are discussed next.

Extending Sketched Entities

Toolbar: 2D Sketch Panel > Extend
Panel Bar: 2D Sketch Panel > Extend



This tool is used to extend or lengthen the selected sketched entity up to a specified boundary. Therefore, to use this tool, you should have at least two entities such that when extended, they meet at a point. Taking the reference of one of the entities, the other will be extended. The entities that can be extended are lines and arcs. On invoking this tool, you will be prompted to select the curve to be extended. As you move the cursor close to the curve to be extended, the original curve will be displayed in red and the portion that will be extended will be displayed in black. While extending the arcs, the point where you select the arc will determine the side that will be extended. Figures 3-1 and 3-2 show the curves before and after extending.

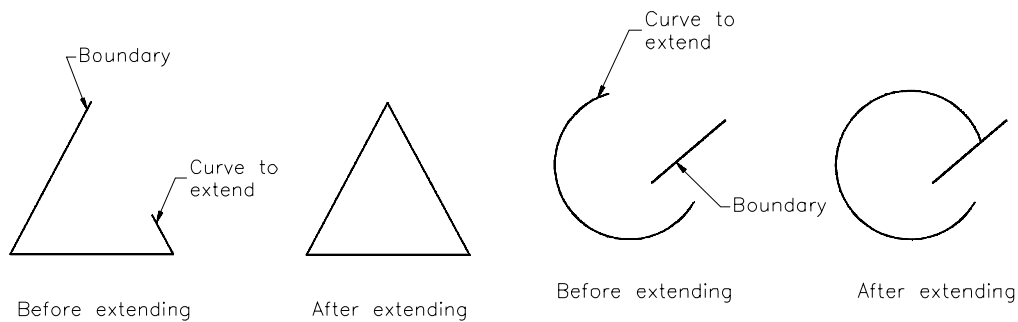


Figure 3-1 Line before and after extending

Figure 3-2 Arc before and after extending

Trimming Sketched Entities

Toolbar: 2D Sketch Panel > Trim
Panel Bar: 2D Sketch Panel > Trim



This tool can be considered as the opposite of the **Extend** tool and is used to chop the selected sketched entity by using an edge (also called the knife edge). The knife edge, in its current form, may or may not actually intersect the entity to be trimmed. However, when extended, the knife edge must intersect the entity to be trimmed. On invoking this tool, you will be prompted to select the portion of the curve to be trimmed. As you take the cursor close to the curve, the portion to be trimmed will be displayed as dashed red lines and the remaining curve will be displayed as a red continuous line. You can select the side of the curve to be trimmed by selecting that side of the curve. Figure 3-3 shows the curves selected for trimming and Figure 3-4 shows the sketch after trimming the edges.

If you use this tool on an isolated entity, it will work as the **Delete** tool and will delete the isolated entity.

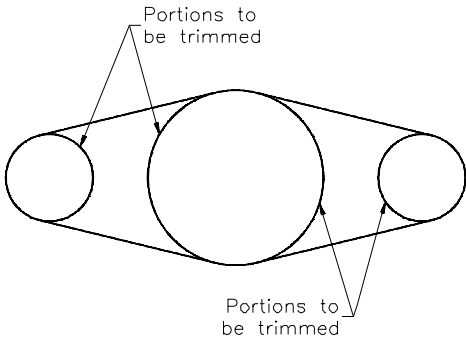


Figure 3-3 Selecting the edges for trimming

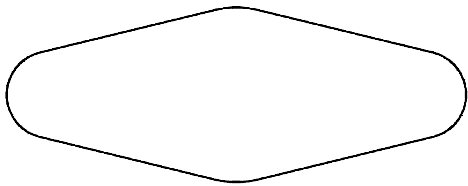


Figure 3-4 Sketch after trimming the edges



Tip. If you are performing editing using the **Trim** or the **Extend** tools, you can toggle between them using the shortcut menu displayed on right-clicking. The active operation has a check mark on its left. You can choose the other operation to switch to that command.

If you are performing the **Extend** or the **Trim** editing operation, you can also temporarily switch to the other one by just pressing the **SHIFT** key. For example, if the active editing operation is **Trim** and you press and hold the **SHIFT** key down, it will act as the **Extend** operation. When you release the **SHIFT** key, the original operation will resume its function of trimming.

Offsetting Sketched Entities

Toolbar: 2D Sketch Panel > Offset
Panel Bar: 2D Sketch Panel > Offset



Offsetting is one of the easiest methods of drawing parallel lines, or concentric arcs and circles. You can select the entire loop as a single entity or select the individual entities to be offset. When you invoke the **Offset** tool, you will be prompted to select the curve to be offset. If you right-click at this point, a shortcut menu will be displayed, as shown in Figure 3-5.

The **Loop Select** option is chosen by default, see Figure 3-5. This option allows you to select the entire loop as a single entity. However, if this option is cleared, the entire loop will be considered as a combination of individual segments and you will be allowed to select individual entities. The **Constrain Offset** option applies the constraints automatically when the loop or the individual entity is offset.

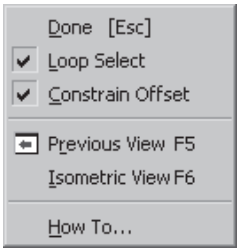


Figure 3-5 The offset shortcut menu

If you select the **Loop Select** option from the shortcut menu, you will be prompted to specify the offset position for the new loop immediately after selecting the original loop. If you specify the location inside the original loop, the new loop will be smaller than the original loop. If you specify the location outside the original loop, the new loop will be bigger.

In case of individual entities, once you have selected the entity to be offset, right-click to display the shortcut menu and choose **Continue**, or press the ENTER key to continue. You will be prompted to specify the location for the new entity. If the selected entity is a line segment, its length will remain the same and if it is an arc or a circle, the size of the new entity will depend on the location of the new point. Figure 3-6 shows offsetting of a loop and Figure 3-7 shows offsetting of an individual entity.

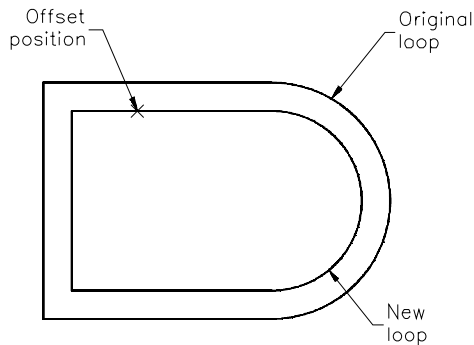


Figure 3-6 Creating a new loop by offsetting the original loop

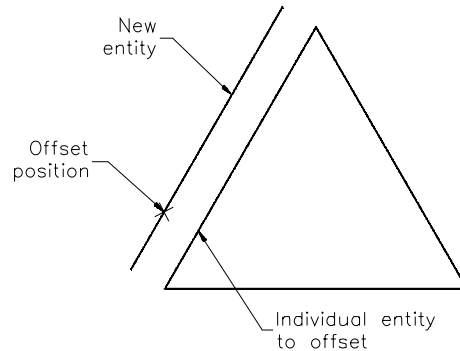


Figure 3-7 Offsetting an individual line segment to a new location

Mirroring Sketched Entities

Toolbar: 2D Sketch Panel > Mirror
Panel Bar: 2D Sketch Panel > Mirror



The **Mirror** tool is used to create a mirror image of the selected entities. The entities are mirrored about a straight line segment. This tool is used to draw sketches that are symmetrical about a line or sketches that have some portion symmetrical about a line. When you invoke this tool, the **Mirror** dialog box will be displayed, see Figure 3-8. The **Select** button will be chosen by default and you will be prompted to select the geometries to be mirrored. You can select multiple entities to be mirrored. Once you have selected the entities to be mirrored, choose the **Mirror line** button; you will be prompted to select the line about which the entities should be mirrored. After selecting the mirror line, choose the **Apply** button; the selected entities will be mirrored about the mirror line. If the mirror line is at an angle, the resultant entities that will be created upon mirroring will also be at an angle. After mirroring the entities, choose the **Done** button to exit this dialog box.



Figure 3-8 The **Mirror** dialog box

Figure 3-9 shows various sketched entities selected for mirroring and the mirror line that will be used to mirror the entities. Figure 3-10 shows the sketch after mirroring the entities. Figure 3-11 shows the entities selected to be mirrored about an inclined mirror line and Figure 3-12 shows the sketch after mirroring the entities.

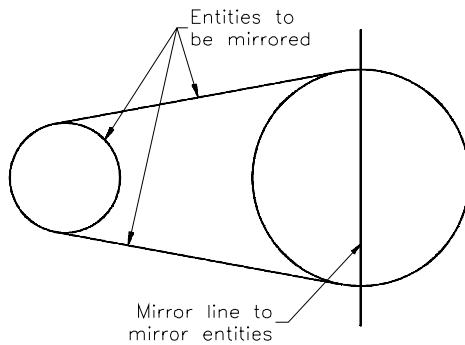


Figure 3-9 Selecting the geometries to be mirrored about the mirror line

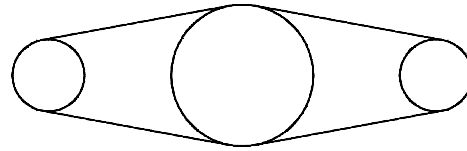


Figure 3-10 Sketch after mirroring the geometries and deleting the mirror line

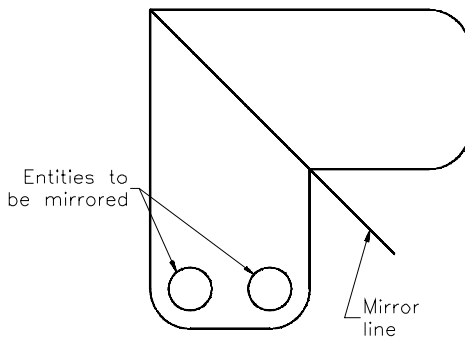


Figure 3-11 Selecting the geometries to be mirrored about the mirror line

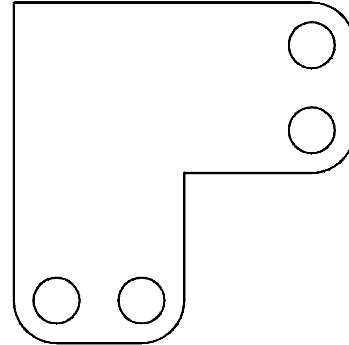


Figure 3-12 Sketch after mirroring the geometries and deleting the mirror line

Moving Sketched Entities

Toolbar: 2D Sketch Panel > Move
Panel Bar: 2D Sketch Panel > Move



The **Move** tool is used to move one or more selected sketched entities from one specified point to the other. The points that can be used to move the entities include the sketched points/hole centers, endpoints of lines, arcs, splines, and the center points of arcs, circles, and ellipses. When you invoke this command, the **Move** dialog box will be displayed, as shown in Figure 3-13. You can also use this dialog box to create copies of the selected entities.

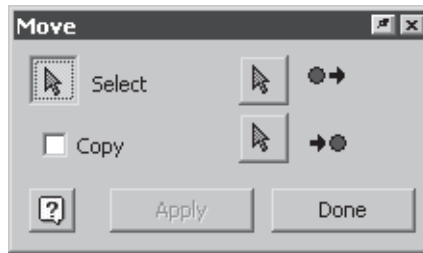


Figure 3-13 The Move dialog box



Note

Remember that the behavior of the selected entities while moving is also governed by the constraints that are applied to them. If the selected entities are constrained with some other entities, the constrained entities will also move. However, if the other entities have a **Fix** constraint applied to them, because of which they cannot move from their location, the original entities will also not be able to move.

Move Dialog Box Options

The options provided in the **Move** dialog box are discussed next.

Select

This button is chosen to select the entities to be moved. When you invoke the **Move** tool, this button is automatically chosen. You can select more than one object using the Window or Crossing options or by selecting them one by one using the left mouse button.

Copy

This check box is selected to create a copy of the selected entities as they are moved. If this check box is selected, a copy of the selected entities will be created and placed at the destination point, keeping the original entities intact at their original location.

From Point

This button is chosen to specify the point that will act as the base point for moving the selected entities. Once you have selected all the entities to be moved, choose this button to select the point from where the movement will start.

To Point

This button is chosen to specify the destination point where the selected entities will be moved. As soon as you specify the from point for moving the entities, this button is automatically chosen in the dialog box and you will be prompted to specify the point where the selected entities will move.

Apply

This button is chosen to apply the move operation to the selected entities such that they are moved from the specified from point to the specified to point. Unless this button is chosen, the objects will not be moved from their original location. After you have applied

the move operation by choosing this button, the **Select** button is again chosen so that you can select other entities for moving.

Done

This button is chosen to exit the **Move** dialog box, thus exiting the **Move** tool.

Figures 3-14 through 3-17 show moving and copying of various sketched entities from one specified point to the other specified point.

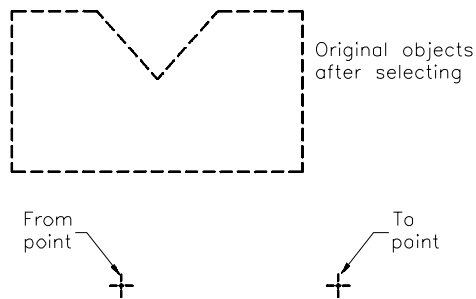


Figure 3-14 Moving the entities using the sketch points

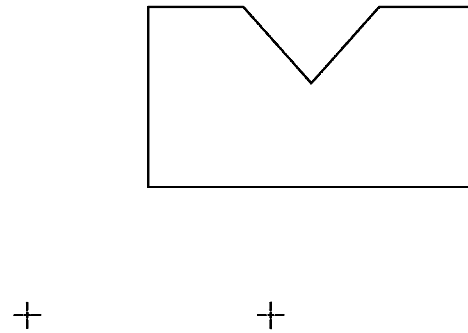


Figure 3-15 Objects after moving

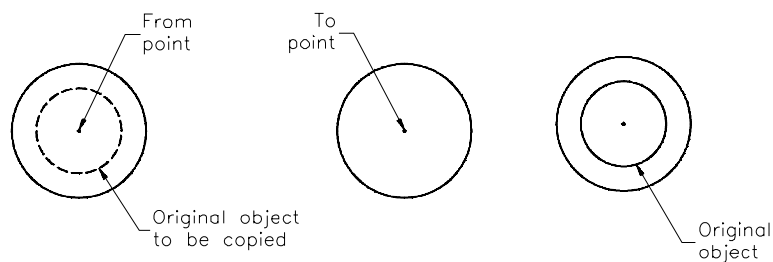


Figure 3-16 Moving and copying the entities using the center points of circles

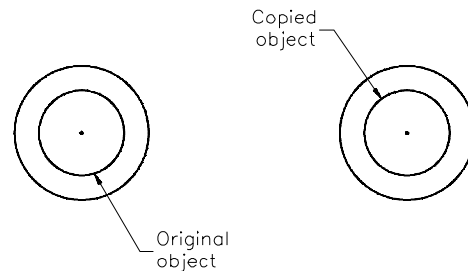


Figure 3-17 Objects after moving and copying

Rotating Sketched Entities

Toolbar: 2D Sketch Panel > Rotate
Panel Bar: 2D Sketch Panel > Rotate



The **Rotate** tool is used to rotate the selected sketched entities about a specified center point. You can also use this tool to create a copy of the selected entities while rotating them. When you invoke this tool, the **Rotate** dialog box will be displayed, as shown in Figure 3-18.

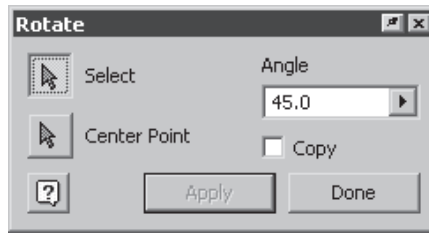


Figure 3-18 The **Rotate** dialog box

Rotate Dialog Box Options

The options provided in the **Rotate** dialog box are discussed next.

Select

This button is chosen to select the entities to be rotated. When you invoke the **Rotate** tool, the **Rotate** dialog box will be displayed and the **Select** button will be automatically chosen. You can use any object selection technique to select one or more objects.

Center Point

This button is chosen to define the base point around which the selected entities will be rotated. This point can also be referred to as the base point of rotation.

Angle

This edit box is used to define the value of the angle through which the selected entities will be rotated. You can enter the value in this edit box or choose the arrow on the right of this edit box to specify the predefined angle values. Remember that a positive angle will rotate the selected entities in the counterclockwise direction and a negative angle will rotate the selected entities in the clockwise direction.

Copy

This check box is selected to create a copy of the selected entities as they are rotated. If this check box is selected, a copy of the selected entities will be created and placed at the angle that you have specified in the **Angle** edit box. The original entities will be intact at their original location.

Apply

This button is chosen to apply the rotate operation to the selected entities. Until this button is chosen, the selected entities will not be rotated. After you apply the rotate operation by choosing this button, the **Select** button is again chosen so that you can select other entities for rotating.

Done

This button is chosen to exit the **Rotate** dialog box.

Figure 3-19 shows the rotation of the selected entities at various angles.

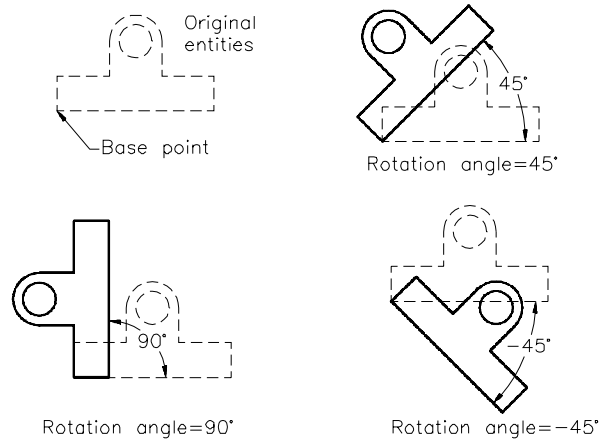


Figure 3-19 Rotating the entities at various angles



Tip. You can also create a copy of the sketched entities using the shortcut menu. Select the sketched entities and right-click to display the shortcut menu. In this menu, choose **Copy**. Again, right-click to display the shortcut menu and choose **Paste** to paste the selected entities, thus creating a copy of the selected entities.

CREATING PATTERNS

Generally, in the mechanical industry, you come across various designs that consist of multiple copies of a sketched feature arranged in a particular fashion. For example, it can be multiple grooves around an imaginary circle. It can also be along the edges of an imaginary rectangle, such as the grooves in the pedestal bearing. Drawing the sketches for such features again and again is a very tedious and time-consuming process. To avoid this lengthy process, Autodesk Inventor provides you with an option of creating patterns of the sketched entities during the sketching stage itself. The patterns are defined as the sequential arrangement of the copies of selected entities. You can create the patterns in a rectangular fashion or a circular fashion. Both these types of patterns are discussed next.

Creating Rectangular Patterns

Toolbar: 2D Sketch Panel > Rectangular Pattern

Panel Bar: 2D Sketch Panel > Rectangular Pattern



Rectangular patterns are the patterns that arrange the copies of the selected entities in rows and columns. When you invoke this tool, the **Rectangular Pattern** dialog box will be displayed, as shown in Figure 3-20. The options provided under this dialog box are discussed next.

Geometry

This button is chosen to select the entities to be patterned. When you invoke the **Rectangular**

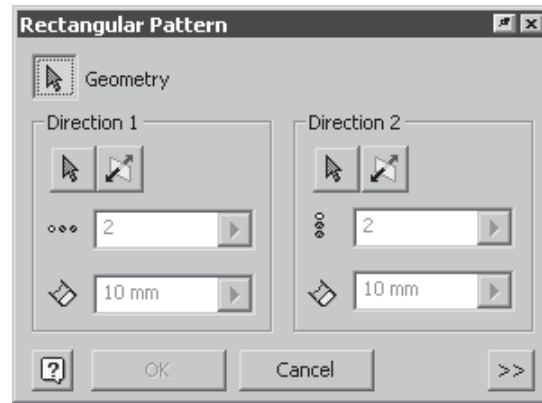


Figure 3-20 The *Rectangular Pattern* dialog box

Pattern tool, the **Geometry** button is automatically chosen. You can select one or more entities to be patterned using any object selecting technique.

Direction 1 Area

This area provides the option of defining the first direction of pattern creation, the number of copies to be created in this direction, and the spacing between the entities. These options are discussed next.

Direction

This is the button with an arrow and is chosen to select the first direction for arranging the items in a rectangular pattern. The other options in **Direction 1** area will be available only after you define the first direction of the pattern creation. The direction can be defined by selecting a line segment, which can be at any angle. The resultant pattern will also be created at an angle, if the line selected to specify the direction is at an angle. As you define the first direction, you can preview the pattern, created using the current values, in the drawing window. The pattern in the preview will be modified dynamically on changing the values in this dialog box.

Flip

This is the button available on the right of the **Direction** button and is chosen to reverse the first direction of the pattern creation. When you define the first direction using the **Direction** button, an arrow appears on the sketch. This arrow displays the direction in which the items of the pattern will be created. If you choose this button, the direction will be reversed and the arrow will point in the opposite direction.

Count

This edit box is used to specify the number of items in the pattern along the first direction. Remember that this value includes the original selected item. On increasing the value in this edit box, you can dynamically preview the increased items in the pattern in the drawing window. You can also select a predefined number of items by choosing the arrow on the right of this edit box. However, if you are using this tool for the first time in the current session of Autodesk Inventor, this arrow will not provide any value.

Spacing

This edit box is used to define the distance between the individual items of the pattern in the first direction. You can enter a value in this edit box or choose the arrow on the right of this edit box to use the **Measure** or **Show Dimension** options to define this value. The **Measure** option allows you to select a line segment, the length of which will specify the distance between the individual items. The **Show Dimension** option allows you to use an existing dimension to specify the distance between the individual items of the pattern. The selected dimension will automatically appear in the edit box. You need to delete the existing value in this edit box to use the measured value.

**Note**

You will learn more about parameters in later chapters.

Direction 2 Area

This area provides the option of defining the second direction of the pattern creation, the number of copies to be created in this direction, and the spacing between the entities. All these options are discussed next.

Direction

This button is chosen to select the second direction for arranging the items of the rectangular pattern.

Flip

This button is available on the right of the **Direction** button and is chosen to reverse the second direction of pattern creation.

Count

This edit box is used to specify the number of items in the pattern along the second direction.

Spacing

This edit box is used to define the distance between the individual items of the pattern in the second direction. Similar to the **Spacing** edit box in the **Direction 1** area, you can directly enter a value in this edit box or use the **Measure** or the **Show Dimension** options to define this value.

Figure 3-21 shows various parameters involved in creating a rectangular pattern with three items along direction 1 and four items along direction 2.

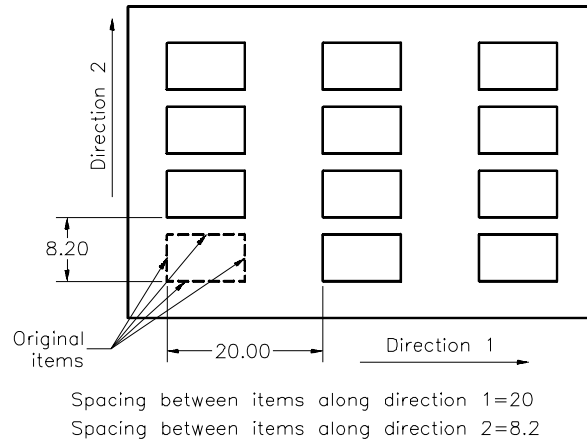


Figure 3-21 Creating a rectangular pattern

More



This button is provided on the lower right corner of the **Rectangular Pattern** dialog box. When you choose this button, this dialog box expands, providing you with more options for creating the pattern, see Figure 3-22. These options are discussed next.

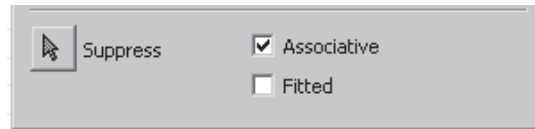


Figure 3-22 More options of the **Rectangular Pattern** dialog box

Suppress

This button is chosen to suppress the selected item from the pattern. When you select any item of the pattern using this button, it will change into dashed lines. The items that are suppressed will be displayed on the screen, but will not participate in the feature creation when you finish the sketch. You can unsuppress these items later, if required.



Note

Editing the sketches of features will be discussed in later chapters.

Associative

This check box is selected so that all the items of the pattern are associated with each other. All the items of the associative pattern are automatically updated, if any one of the entity is modified. For example, if you modify the dimension of any of the items of the pattern, the dimensions of all the other items are also modified. However, if you clear

this check box before creating the pattern, all the items will be individual entities and can be modified individually.

Fitted

This option works in combination with the **Spacing** option in the **Direction 1** and **Direction 2** areas. If you select this option, the specified number of items will be created in the distances specified in the **Spacing** edit boxes in the **Direction 1** and **Direction 2** areas. Figure 3-23 shows the pattern created by clearing this check box (spacing is incremental) and Figure 3-24 shows the pattern created by selecting this check box (included spacing between all the items).

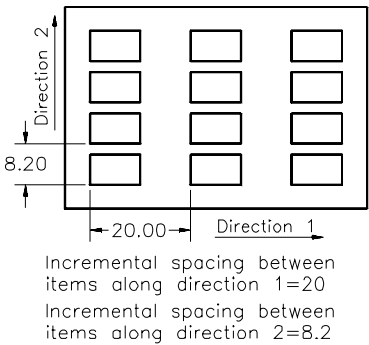


Figure 3-23 Pattern created with **Fitted** check box cleared

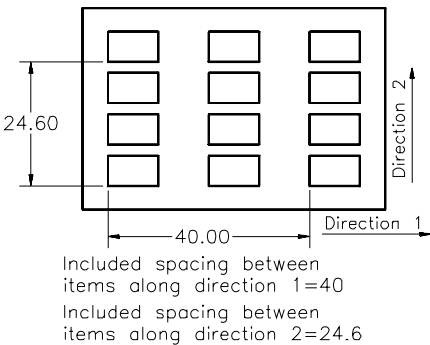


Figure 3-24 Pattern created with **Fitted** check box selected

Creating Circular Patterns

- Toolbar:** 2D Sketch Panel > Circular Pattern
- Panel Bar:** 2D Sketch Panel > Circular Pattern



Circular patterns are the patterns created around the circumference of an imaginary circle. To create the circular pattern, you will have to define the center of that imaginary circle. When you invoke this tool, the **Circular Pattern** dialog box will be displayed, see Figure 3-25. The options provided under this dialog box are discussed next.

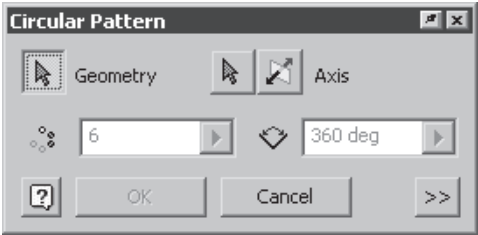


Figure 3-25 The **Circular Pattern** dialog box

Geometry

This button is chosen to select the entities to be patterned. As you select the individual entities, they turn blue, indicating that they are selected.

Axis

This is the button with an arrow and is provided on the right of the **Geometry** button. This button is chosen to select the center of the imaginary circle around which the circular pattern will be created. The points that can be used to define the center of the pattern creation are the endpoints of lines, splines, and arcs, center points of arcs, circles, and ellipses, and the points/hole centers. Most of the options in the **Circular Pattern** dialog box are enabled only after you select the axis of rotation. You can dynamically preview the pattern created using the current values. If you modify the other values in this dialog box, the preview of the pattern will also be modified.



Tip. If you select an arc or a circle to define the axis of the circular pattern, its center will be automatically selected as the center of the circular pattern. However, this is not possible in case of an ellipse. You cannot select an ellipse to define the center of the circular pattern. You will have to select the center of ellipse.

Flip

This is the button provided on the right of the **Axis** button and is chosen to reverse the direction of pattern creation. By default, the circular pattern will be created in the counterclockwise direction. If you choose this button, the circular pattern will be created in the clockwise direction.



Tip. If the circular pattern is created through 360-degree, you cannot notice the difference in changing the direction of the pattern creation from counterclockwise to clockwise. However, if the pattern is created through an angle less than 360-degree, you will notice the difference of changing the direction of pattern.

Count

This edit box is used to specify the number of items in the circular pattern. You can enter a value in this edit box or choose the arrow provided on the right of this dialog box for using the predefined values, or for using the **Measure** or **Show Dimension** options. These options are the same as those discussed in the rectangular pattern.

Angle

This edit box is used to define the angle for creating the circular pattern. You can directly enter an angle in this edit box or use the predefined values by choosing the arrow on the right of this edit box. You can also use the **Measure** or the **Show Dimension** options to define the angle. Figures 3-26 and 3-27 show the circular patterns created using various angles.

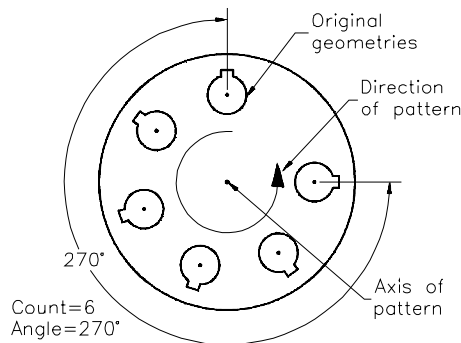


Figure 3-26 Circular pattern with 6 items and a 270-degree angle

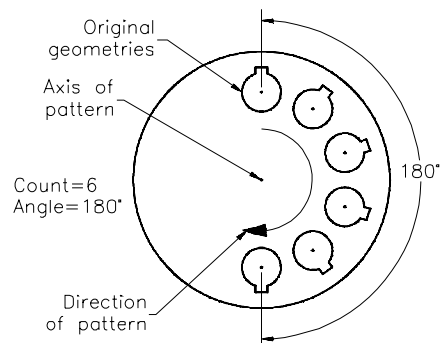


Figure 3-27 Circular pattern with 6 items and a 180-degree angle

More

This is the button with two arrows and is provided on the lower right corner of the **Circular Pattern** dialog box. When you choose this button, the **Circular Pattern** dialog box expands, providing more options, see Figure 3-28. These options are discussed next.

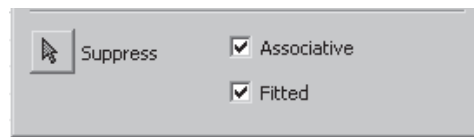


Figure 3-28 More options of the **Circular Pattern** dialog box

Suppress

This button is chosen to suppress the selected item from the pattern. Similar to the rectangular pattern, when you select any item of the circular pattern, it will change into dashed lines. Although the items that are suppressed will be displayed in the drawing window, they will not participate in the feature creation when you finish the sketch. However, you can unsuppress these items later, if you need them.

Associative

This check box is selected so that all the items of the pattern are associated with each other. All the items of the associative pattern are automatically updated if any one of the entity is modified. If you clear this check box before creating the pattern, all items will become individual entities and can be modified individually.

Fitted

This option works in combination with the **Angle** edit box. If you select this option, the specified number of items will be created such that the angle specified in the **Angle** edit box defines the included angle between all the items. This check box is selected by default in the **Circular Pattern** dialog box. If you clear this check box, the angle that you specify in the **Angle** edit box will be considered as the incremental angle between each item. Figure 3-29 shows the pattern created by selecting this check box (included angle between all the items) and Figure 3-30 shows the pattern created by clearing this check box (angle is incremental).

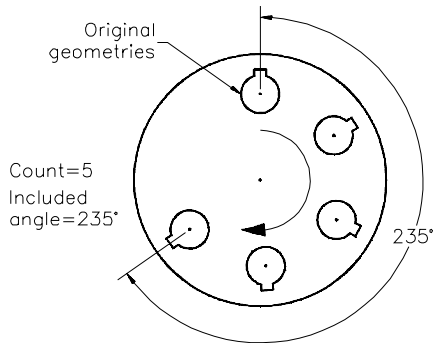


Figure 3-29 Pattern created with the **Fitted** check box selected

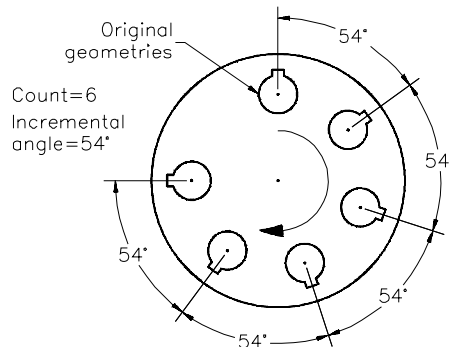


Figure 3-30 Pattern created with the **Fitted** check box cleared



Tip. If you create a circular pattern through an angle of 360-degree and clear the **Fitted** check box, you will see only one item in the drawing window. This is because the incremental angle between the individual items is 360-degree and all the items will be arranged on top of each other, displaying only one copy.

WRITING TEXT IN THE SKETCHING ENVIRONMENT

Toolbar: 2D Sketch Panel > Create Text
Panel Bar: 2D Sketch Panel > Create Text



Autodesk Inventor allows you to write text in the sketching environment. The text behaves like other sketched entities and can be converted into features using the modeling tools of Autodesk Inventor. To write the text, choose the **Create Text** button from the **2D Sketch Panel** panel bar; you will be prompted to select the location of the text. You can also drag a window to define the text box. Specify a point in the drawing window to start the text or press and hold the left mouse button down and drag the mouse to define a window; the **Format Text** dialog box will be displayed, as shown in Figure 3-31.

Format Text Dialog Box Options

The options in this dialog box are discussed next.

Text Justification

You can select the justification for writing the text by choosing the buttons in the upper left portion of this dialog box. The justification of a text is defined using a combination of two buttons. By default, the **Left Justification** and **Top Justification** buttons are chosen. As a result, the justification for the text is top left. You can select other justifications by choosing their respective buttons. The **Baseline Spacing** button is available only when you choose the **Single Line Text** button on the left of the **Spacing** drop-down list.

Text Box

If this button is chosen, a construction line box is placed around the text.

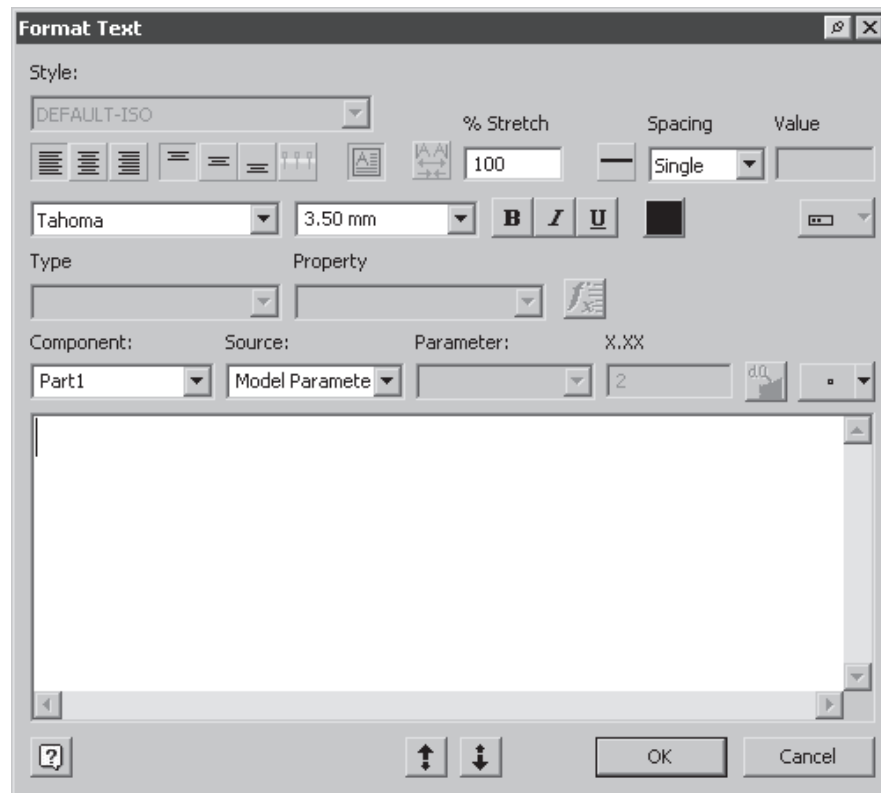


Figure 3-31 The *Format Text* dialog box

Fit Text

This button is available only when you choose the **Single Line Text** button and is used to fit the text in a single line inside the window that you defined by dragging the mouse.

Stretch

You can define the percentage of text stretching in the **% Stretch** edit box. The default value in this edit box is 100. As a result, there is no stretching of the text. If you enter a value more than 100, the text width will be increased. If you enter a value less than 100, the width of the text will be reduced.

Spacing

You can select the option to define the spacing between the text lines using the **Line Spacing** drop-down list. If you select the **Multiply** option from this drop-down list, the **Value** edit box will be enabled and you can enter the multiplication factor for the line spacing in this edit box.

Text Font

The **Text Font** drop-down list is located below the justification buttons. You can select the font for the text using this drop-down list.

Text Height

The **Text Height** edit box is used to specify the height of the text. You can enter the height in this edit box or select the standard values using the down arrow on the right of this edit box.

Text Style

You can define the text style by choosing the **Bold**, **Italics**, and **Underline** buttons on the right of the **Text Height** drop-down list.

Color

The default color of the text is black. You can change the color of the text by choosing the **Color** button on the right of the **Underline** button. When you choose this button, the **Color** dialog box is displayed. You can specify the color of the text using this dialog box.

Rotation Angle

The **Rotation Angle** flyout is available only when the **Text Box** button is not chosen. This flyout is used to specify the rotation angle for the text.

Inserting Symbols in the Text

Autodesk Inventor provides you with some standard symbols that you can insert in the text. To insert the symbols, choose the down arrow on the right of the **Insert symbol** button. The standard symbols in Autodesk Inventor are displayed, as shown in Figure 3-32.



Figure 3-32 Default symbols

Text Window

You can enter the in the **Text Window**. This is the white area provided in the **Format Text** dialog box. You can also paste the text copied from some other source. The text written in this window will appear on the screen.

Zoom In/Zoom Out



You can zoom in and out of the text in the **Text Window** by choosing the **Zoom In** and **Zoom Out** buttons provided at the bottom of this dialog box.



Note

The remaining options in the **Format Text** dialog box are used in the **Drawing** module and so are not discussed here.

INSERTING IMAGES AND DOCUMENTS IN THE SKETCHES

Toolbar: 2D Sketch Panel > Insert Image
Panel Bar: 2D Sketch Panel > Insert Image



The **Insert Image** tool allows you to insert the external images in the sketch. The images that you can insert include JPG, BMP, PCX, TIFF, TGA, and so on. You can also insert Word documents or Excel Spreadsheets using this tool. To insert an image, invoke this tool; the **Open** dialog box will be displayed, as shown in Figure 3-33.

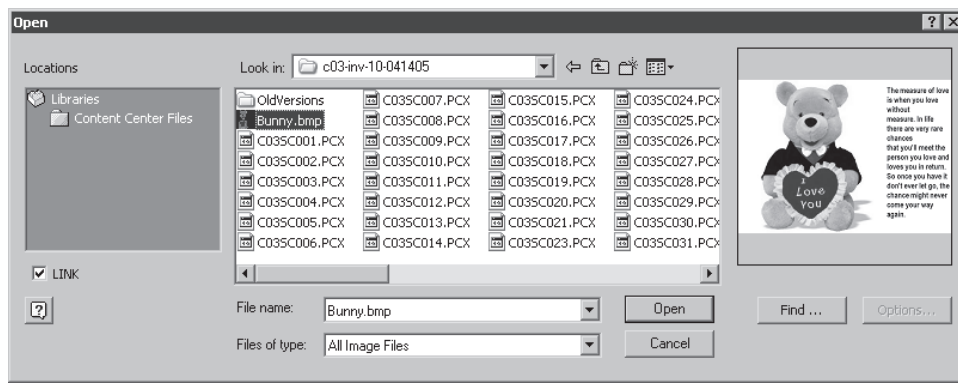


Figure 3-33 The **Open** dialog box

Select the image or document using this dialog box and then choose the **Open** button; the dialog box will be closed and you will be prompted to select the sketch point. The point you specify on the screen will be taken as the insertion point for the image. After inserting the image, right-click and choose **Done** from the shortcut menu to exit this tool.



Note

You may need to modify the drawing display area using the **Zoom All** tool to display the image on the screen.



Tip. To modify the size of the image inserted in the sketch, drag it by holding one of its four edges. Depending on the direction in which you drag it, the size will increase or decrease. To rotate the image, hold it from one of the corners and drag. The image will be rotated in the direction in which you drag the cursor. To move the image, press and hold the left mouse button anywhere on the image and drag the cursor.

EDITING SKETCHED ENTITIES BY DRAGGING

You can also edit the sketched entities by dragging them. Depending on the type of entity selected and the point of selection, the object will be moved or stretched. For example, if you select a circle at its center and drag, it will be moved. However, if you select the same at a point on its circumference, it will be stretched to a new size. Similarly, if you select a line at its

endpoints, it will be stretched and if you select a line at a point other than its endpoints, it will be moved. Therefore, editing the sketched entities by dragging is entirely based on the selection points. The following table gives you the details of the operation that will be performed when you drag various objects.

Object	Selection point	Operation
Circle	On circumference	Stretch
	Center point	Move
Arc	On circumference	Stretch
	Center point	Move
Polygon	Any of the edges	Move
	Endpoints	Move
Single line	Any point other than the endpoints	Move
	Endpoints	Stretch
Rectangle	All lines selected together	Move
	Any one line or any endpoint	Stretch

TOLERANCES

In simple terms, tolerance is defined as the permissible variation from the actual value. Because it is an allowed variation, you can vary the dimension of the component through the specified value while manufacturing.

Adding Tolerances to the Dimensions in the Sketching Environment

In Autodesk Inventor, the tolerances are added after creating the dimensions. To add tolerance to a dimension, invoke the **Select** tool to make sure that no other drawing or dimensioning tool is active. Next, right-click on the dimension and choose **Dimension Properties** from the shortcut menu; the **Dimension Properties** dialog box will be displayed. You can use the options in the **Dimension Settings** tab of this dialog box to add tolerances, see Figure 3-34.

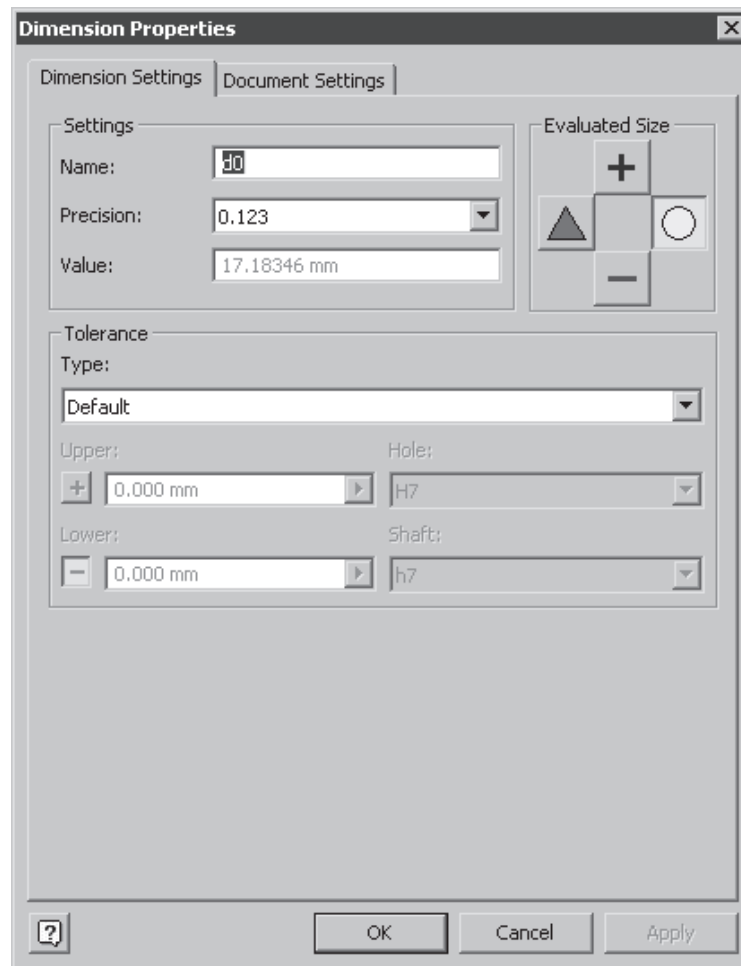


Figure 3-34 The *Dimension Settings* tab of the *Dimension Properties* dialog box

By default, the **Default** option is selected in the **Type** drop-down list of the **Tolerance** area. As a result, no tolerance is added to the dimension. To add tolerance, select the required tolerance type from this drop-down list. Next, specify the value of the upper and lower limits in the **Upper** and **Lower** edit boxes. If you select the fit type, you can select the values of the hole fit and the shaft fit from the **Hole** and **Shaft** drop-down lists.

Once you have defined all the tolerance values, choose the **Apply** button in the **Dimension Properties** dialog box. You will notice that the tolerance is applied to the selected dimension. Now, choose **OK** to exit this dialog box. Figure 3-35 shows a sketch with the tolerance applied to the dimensions.

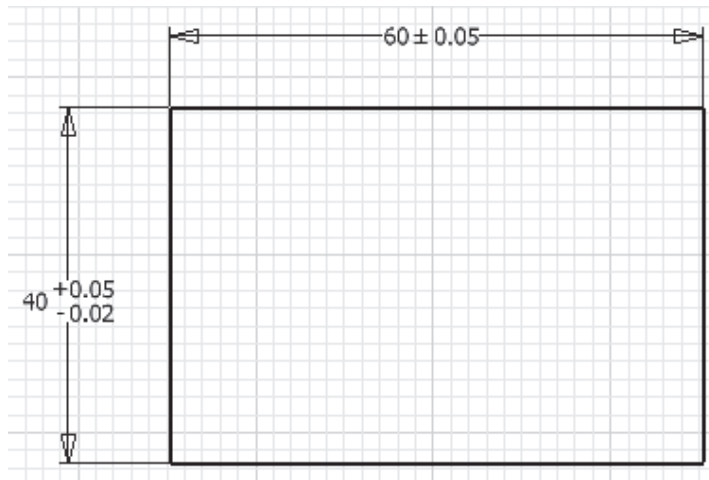


Figure 3-35 Sketch with tolerance applied to the dimensions

In this figure, the vertical dimension is applied the deviation tolerance type. The upper deviation value for this tolerance is 0.05 mm and the lower deviation value is 0.02 mm. The horizontal dimension in the same figure is applied the symmetric tolerance with a value of 0.05 mm.

CONVERTING THE BASE SKETCH INTO A BASE FEATURE

As mentioned earlier, any 3D design is a combination of various sketched, placed, and work features. The first feature, generally, is a sketched feature. In this chapter and in previous chapters, you have learned to draw the sketches for these base features and to dimension them. After you have finished drawing and dimensioning the sketch, choose the **Return** button on the **Inventor Standard** toolbar. On choosing this button, you will exit the sketching environment and enter the **Part** module. You will also notice that the **2D Sketch Panel** panel bar is replaced by the **Part Features** panel bar. Autodesk Inventor provides you with a number of tools such as **Extrude**, **Revolve**, **Loft**, **Sweep**, and so on to convert these base sketches into base features. However, in this chapter, you will learn the use of the **Extrude** and **Revolve** tools for converting the base sketch into a base feature. The remaining tools will be discussed in later chapters.



Tip. You can also proceed to the **Part** module from the sketching environment by using the shortcut menu. Right-click in the graphics window and choose **Finish Sketch**.



Note

It is recommended that when you switch to the part modeling environment, you should change the current view to the isometric view. This is because in the isometric view, you can dynamically preview the result of the **Extrude** or the **Revolve** tools while you are defining the values in their respective dialog boxes. To change the current view to the isometric view, right-click in the graphics window and choose **Isometric View** from the shortcut menu.

EXTRUDING THE BASE SKETCH

Toolbar: Part Features > Extrude
Panel Bar: Part Features > Extrude



The **Extrude** tool is one of the most extensively used tools for creating a design. Extrusion is a process of adding or removing material defined by the sketch, along the Z axis of the current sketching plane. If you create the first feature, you will be given the option of just adding the material and not removing it. This is because there is no existing feature from which you can remove the material. When you invoke this tool, the **Extrude** dialog box will be displayed. The options provided in the **Extrude** dialog box are discussed next.

Shape Tab

The options provided in this tab (Figure 3-36) are discussed next.

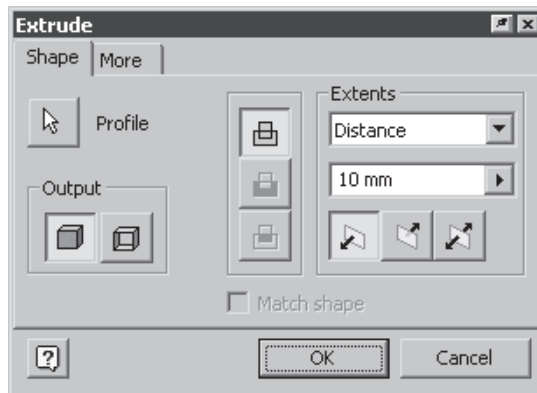


Figure 3-36 The **Shape** tab of the **Extrude** dialog box

Profile

This button is chosen to select the sketch to be extruded. If the sketch consists of a single loop, it will be automatically selected to be extruded when you invoke the **Extrude** tool. Also, because in this case the sketch is already selected, the **Profile** button will not be chosen. However, if the sketch consists of more than one loop, this button will be chosen and you will be prompted to select the profile you want to extrude. As you move the cursor close to one of the loops, it will be highlighted. After you have selected the sketch to be extruded, choose this button again; the preview of the resultant solid will be displayed in the drawing window. Note that if you select any of the inner loops, only that single loop will be extruded. Also, after the extrusion of one of the inner loops, the remaining loops will no more be displayed on the screen. But, if you select the profile by specifying a point inside the outer loop but outside the inner loops, the sketch will be extruded such that the resultant solid will have the inner loops subtracted from the outer loop, see Figures 3-37 and 3-38.

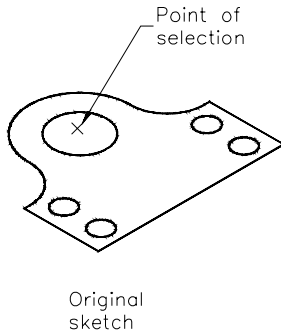


Figure 3-37 Specifying the selection point inside the inner loop

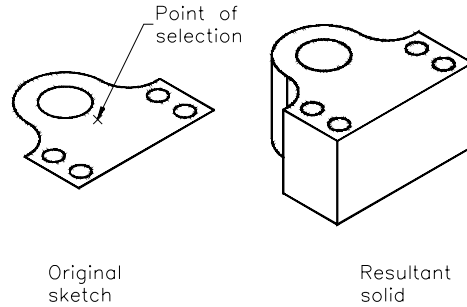


Figure 3-38 Specifying the selection point between the inner and outer loops



Tip. To remove the closed loop that has been selected using the **Profile** button, hold down the **SHIFT** key and then select the closed loop to be removed. You will notice that the selected closed loop is no more highlighted, suggesting that it has been removed from the selection set.

Output Area

The buttons in the **Output** area are used to specify the type of resulting feature that will be created. If you select a closed sketch to extrude, the **Solid** button is chosen automatically. As a result, a solid feature will be created. If you choose the **Surface** button, the resultant feature will be a surface and not a solid. Remember that the sketches for surface models do not need to be closed loops. If you select an open sketch to extrude, this button is chosen automatically when you invoke the **Extrude** dialog box.

Operation Area

This area is provided on the left of the **Extents** area and has three buttons. Because you are creating the first feature, only the **Join** button will be available in this area. This button is discussed next. The remaining buttons are discussed in Chapter 4.

Join

This is the first button provided in the **Operation** area. This button is chosen to create a feature by adding the material defined by the sketch.

Extents Area

You can select the method of terminating the extruded feature using the options in the drop-down list available in the **Extents** area. These options are discussed next.

Distance

By default, the **Distance** option is selected in this drop-down list. This option is used to define the extrusion depth by specifying its numeric value. The value of extrusion can be specified in the **Depth** edit box available below this drop-down list. When this



Tip. You can dynamically modify the depth of extrusion by moving the cursor over one of the edges in the preview. The outline of the preview turns red and a double-sided arrow is displayed below the cursor. Press and hold the left mouse button down and drag the cursor to change the extrusion depth.

termination option is selected from the drop-down list, there will be three buttons provided below the **Depth** edit box. These buttons are used to specify the direction of extrusion. The current direction will be displayed by the first button. You can reverse the direction of feature creation by choosing the second button. The third button extrudes the feature equally in both directions from the current sketch plane. This button is also called the **Mid-plane** button. For example, if the specified extrusion depth is 20 mm, the resultant feature will be created such that it is extruded 10 mm above the current sketch plane and 10 mm below the current sketch plane.



Note

You can also select a predefined distance value or the **Measure** and the **Show Dimension** options by choosing the arrow on the right of the **Depth** edit box to specify the depth of extrusion.

To

This is the second option in the **Distance** drop-down list and is used to define the termination of the extruded feature using a work plane, planar face, or extended face. When you select this option, all other options in the **Extents** area are removed and only the **Select surface to end the feature creation** button is displayed. If you select a plane or a face that does not intersect the extruding feature to terminate the extrusion, the **Check to terminate feature on the extended face** check box is displayed. This check box is selected to terminate the feature on the plane or the planar face as if it was extended.

From To

This is the third option in the **Distance** drop-down list. This option uses two planes to define a feature. The first plane defines the plane from which the feature will start and the second plane defines the plane for terminating the feature. When you select this option, all the remaining options in the **Extents** area are replaced by two buttons. The upper button is the **Select surface to start the feature creation** button and is used to define the plane where the feature starts. The lower button is the **Select the surface to end the feature creation** button and is used to define the plane where the feature terminates.



Note

The methods of creating work planes will be discussed in later chapters. The **Distance** drop-down list will display more options once you have created the base feature. These options will be discussed in Chapters 4.

More Tab

The **Taper** option in the **More** tab (Figure 3-39) is discussed next. The remaining options will be discussed in later chapters.

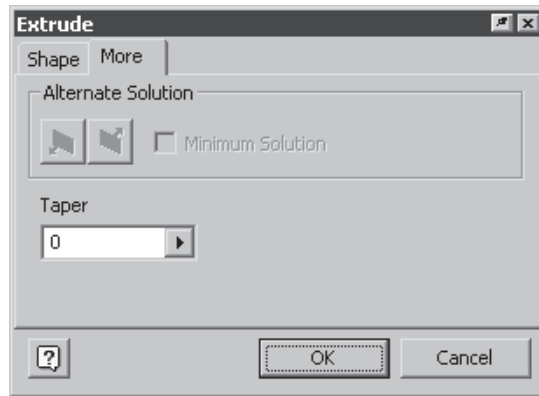
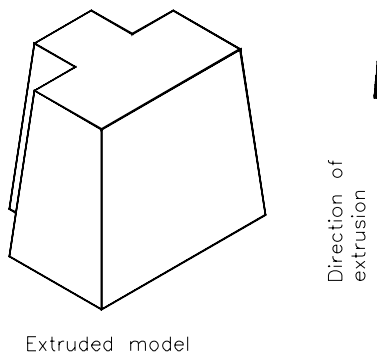


Figure 3-39 More tab of the **Extrude** dialog box

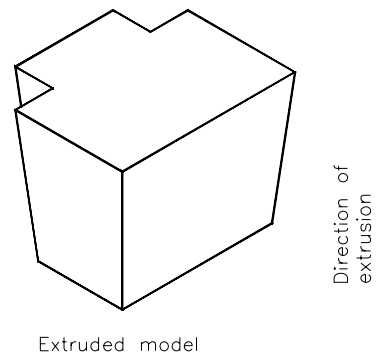
Taper

This edit box is used to define the taper angle for the resulting solid model. Taper angles are generally provided to solid models for their easy withdrawal from the molds. A negative taper angle will force the solid to taper inwards, thus creating a negative taper. A positive taper angle will force the resultant solid to taper outwards, thus creating a positive taper. When you define a taper angle, an arrow is displayed in the preview of the solid model in the drawing window. Depending on the positive or negative value of the taper angle, this arrow will point inwards or outwards from the sketch. Figures 3-40 and 3-41 show the model created using the negative and positive taper angles.



Extruded model

Figure 3-40 Extruding the model with a negative taper angle



Extruded model

Figure 3-41 Extruding the model with a positive taper angle

Figure 3-42 shows a model extruded with a positive taper angle using the **Mid-plane** option.



Tip. You will notice that as soon as a feature is created, it is assigned a material. The default material is **As Material**. You can also change the material of the feature. To assign a new material, select it from the drop-down list in the **Inventor Standard** toolbar. The selected material is applied to the model.

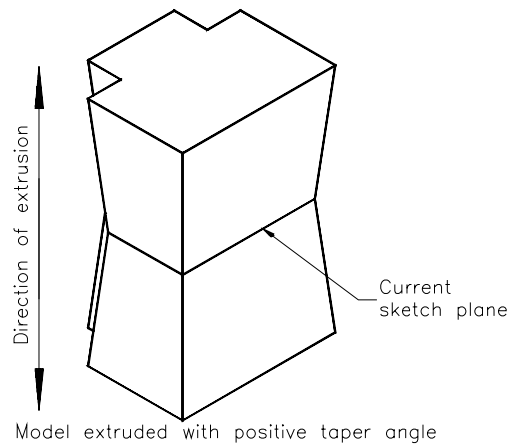


Figure 3-42 Model extruded with a positive taper angle and using the *Mid-plane* button


REVOLVING THE BASE SKETCH

Toolbar:

Part Features > Revolve

Panel Bar:

Part Features > Revolve

 The **Revolve** tool is used to create circular features like shafts, couplings, pulleys, and so on. You can also use this tool for creating cut features that are circular in shape. A revolved feature is created by revolving the sketch about an axis. You can use a normal line segment, a center line, or a construction line of a sketch as the axis for revolving the sketch. When you invoke this tool, the **Revolve** dialog box will be displayed, as shown in Figure 3-43.

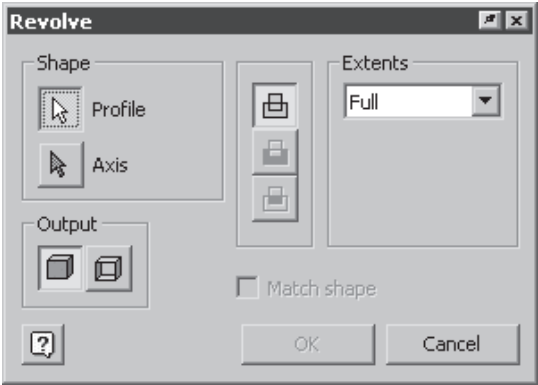


Figure 3-43 The *Revolve* dialog box

Shape Area

The buttons in this area are used to select the sketch to be revolved and the axis of revolution. These buttons are discussed next.

Profile

This button is chosen to select the sketch to be revolved. If there is only one loop in the drawing window, it will be automatically selected and the **Profile** button will not be chosen. However, if there are more than one loops, this button will be chosen and you will be prompted to select the profile to be revolved.

Axis

This button is chosen to select the axis for revolving the sketch. As mentioned earlier, you can select a line segment in the sketch as the axis for creating the revolved feature. When you select the axis, the preview of the feature that will be created using the current values will be displayed in the drawing window.

Output Area

The buttons in the **Output** area are used to specify the type of output for the revolved feature. If you choose the **Solid** button, the resulting feature will be a solid. However, if you choose the **Surface** button, the resulting feature will be a surface. The sketch for the surface may or may not be closed. But for a solid feature, the sketch needs to be closed.

Operation Area

This is the area with three buttons located on the right of the **Shape** area. While creating the base features, only one button will be enabled in this area. The remaining buttons will be enabled after you have created at least one feature. The button that is enabled is discussed next and the remaining buttons will be discussed in Chapter 4.

Join

This is the first button in the **Operation** area and is chosen to create a revolved feature by adding the material defined by the sketch. This is the option you will be using for creating the base feature.

Extents Area

The drop-down list provided under this area is used to specify the methods of termination of a revolved feature. These options are discussed next.

Full

This option is chosen to create a feature by revolving the sketch through 360-degree. This is the default option.

Angle

This option is used to terminate the revolved feature at an angle less than 360-degree. The angle of revolution can be specified in the edit box displayed below this drop-down list when you select the **Angle** option. You can use a predefined value by choosing the arrow provided on the right side of this edit box. You can also use the **Measure** and the **Show Dimensions** options to define the angle of revolution.

When you select the **Angle** option, three buttons will be displayed in this area. These buttons are used to define the direction of rotation. You can also revolve the sketch equally in both directions by choosing the **Mid-plane** button. Figures 3-44 and 3-45 show the features created by revolving the sketches through various angles.

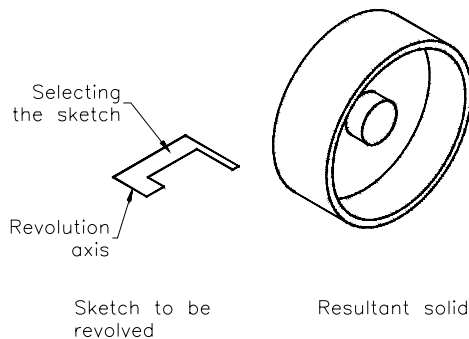


Figure 3-44 Revolving the sketch through 360-degree

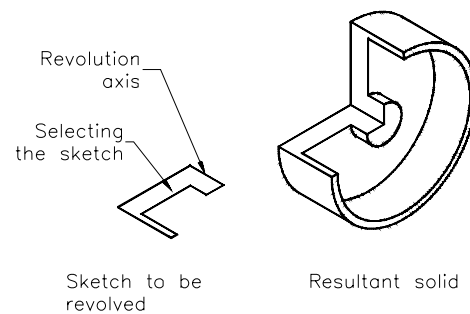


Figure 3-45 Revolving the sketch through 270-degree

ROTATING THE VIEW OF A MODEL IN 3D SPACE



Autodesk Inventor provides you with an option of rotating the view of a solid model freely in 3D space. This allows you to visually maneuver around the solid model and view it from any direction. To invoke this option, choose the **Rotate** button from the **Inventor Standard** toolbar. When you choose this button, a circle will be displayed with small lines at all four quadrant points and a cross at the center of the circle. The circle is called the **rim**, the small lines at four quadrants are called **handles**, and the cross at the center is called the **center point**. Also, when you invoke this tool, the shape of the cursor changes and the new shape will depend on its current position. For example, if the cursor is inside the rim, it will show two elliptical arrows, suggesting that the model can be freely rotated in any direction. If you move the cursor close to the horizontal handles, the cursor will be changed to a horizontal elliptical arrow. The methods to rotate the view of a model are discussed next.

Rotating the View of a Model Freely in 3D Space



To freely rotate the view of a model, move the cursor inside the rim; the cursor will be replaced by two elliptical arrows. Select a point inside the rim and then drag it anywhere in the drawing window. The model will dynamically rotate as you drag the cursor around the drawing window.

Rotating the View of a Model Around the Horizontal Axis



To rotate the view of a model around the horizontal axis, move the cursor close to one of the horizontal handles; the cursor will be replaced by a horizontal elliptical arrow. Now, select a point and drag the cursor to rotate the model along the horizontal axis.

Rotating the View of a Model Around the Vertical Axis



To rotate the view of a model around the vertical axis, move the cursor close to one of the vertical handles; the cursor will be replaced by a vertical elliptical arrow. Now, select a point and drag the cursor to rotate the model along the vertical axis.

Rotating the View of a Model Around the Center Point



To rotate the view of a model around the center point of the current view, which is also normal to it, move the cursor close to the rim; the cursor will be replaced by a circular arrow. Now, select a point and drag the cursor. The model will be rotated around the center point.

Figure 3-46 shows the view of a model being rotated freely in 3D space.

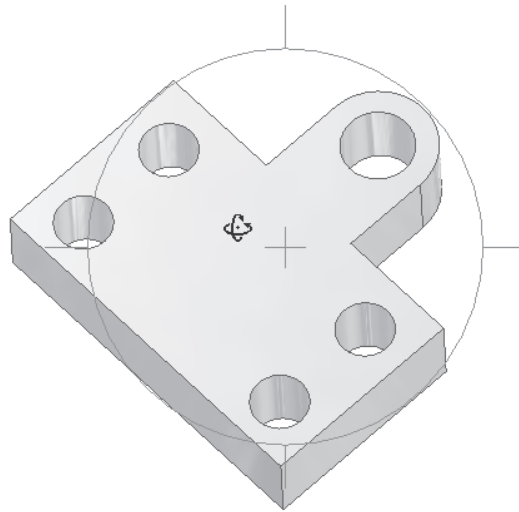


Figure 3-46 Rotating the view of a model freely in 3D space

To exit this tool, right-click to display the shortcut menu and choose **Done**. This shortcut menu also displays the other options that are discussed next.

Common View

If you choose this option, a cube is displayed at the center of the drawing window. All eight vertices and six faces of this cube have green arrows. You can choose any of these arrows and the model will be displayed as viewed from that direction. Figure 3-47 shows the common views that can be used to reorient the model. Once you exit the **Rotate** tool in the **Common View** mode, the next time you invoke this tool, the **Common View** mode will be invoked instead of the **Free Rotation** mode. The **Free Rotation** option replaces the **Common View** option in the shortcut menu. If you choose the **Free Rotation** option, you will again switch to the 3D rotation mode. You can also toggle between these two options using the SPACEBAR key.

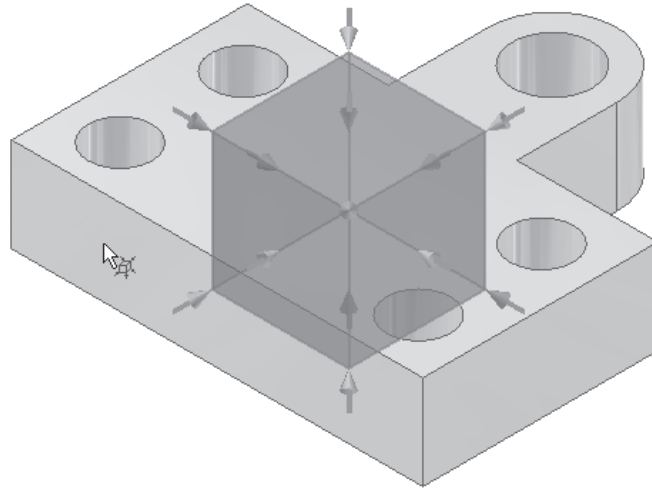


Figure 3-47 Reorienting the model using the common views

Redefine Isometric

This option is used in combination with the **Common View** option. When you choose the **Isometric View** option from the shortcut menu, the model is displayed as SE isometric view, where the direction of +X axis is considered as East. However, if you want to set the isometric view to any other view, you can use this option. Modify the viewing direction by using the **Common View** option and then choose this option from the shortcut menu. The current view will be displayed whenever you invoke the isometric view.

Previous View

When you choose this option, the view showing the previous orientation of the model will be displayed again. The **Next View** option is added to the shortcut menu when you choose this option. The **Next View** option is used to activate the view that was current before you chose the **Previous View** option. You can also invoke the previous view by pressing the F5 key and the next view by holding down the SHIFT key and then pressing the F5 key.

Isometric View

This option is chosen to reorient the model such that it is displayed in the isometric view. You can also invoke the isometric view by pressing the F6 key on the keyboard.



Note

The **Pan** and the **Zoom** options are the same as those discussed in Chapter 1, *Drawing Sketches for the Solid Models*.

CONTROLLING THE DISPLAY OF MODELS

Autodesk Inventor allows you to control the display of the models by setting various display modes and setting the camera type for displaying them. You can also control the display of their shadows. These options of controlling the display of the models are discussed next.

Setting Display Modes

You can set the display modes for the solid models using the buttons provided in the **Shaded Display** flyout in the **Inventor Standard** toolbar. Various display modes that you can set for the solid models are discussed next.

Shaded Display



This is the default display mode set for the models. Because the display is set to shaded, the model will be assigned a material and will behave as a solid model with an opaque material assigned to it.

Hidden Edge Display



The **Hidden Edge Display** mode displays the model with a material applied to it. Also, the hidden edges of the model are displayed in light color. To change the display type, choose the arrow on the right of the **Shaded Display** button in the **Inventor Standard** toolbar; the **Shaded Display** flyout will be displayed. From this flyout, choose **Hidden Edge Display** button.

Wireframe Display



The **Wireframe Display** displays the model in a wireframe mode and you can see through the model. To set this display mode, choose the **Wireframe Display** button from the **Shaded Display** flyout.

Setting the Camera Type

By default, the models are displayed using the orthographic camera type. You can change the camera type from the default orthographic to the perspective camera. This is done by choosing the arrow on the right of the **Orthographic Camera** button in the **Inventor Standard** toolbar. From the flyout, choose the **Perspective Camera** button to display the model using a perspective camera. Figure 3-48 shows a model using the perspective camera.

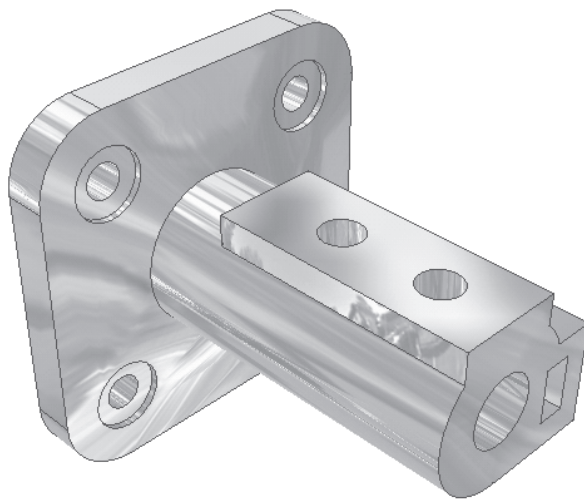


Figure 3-48 Displaying the model using the perspective camera

Setting the Shadow Options

Autodesk Inventor allows you to make your design realistic by casting their shadows. By default, the shadow option is turned off. You can cast two type of shadows using the **No Ground Shadow** flyout in the **Inventor Standard** toolbar. These two types of shadows are discussed next.

Ground Shadow



A ground shadow is a flat shadow below the model. To cast a flat shadow, choose the down arrow on the right of the **No Ground Shadow** button in the **Inventor Standard** toolbar and then choose the **Ground Shadow** button from the flyout. The flat shadow of the model appears below it, as shown in Figure 3-49.

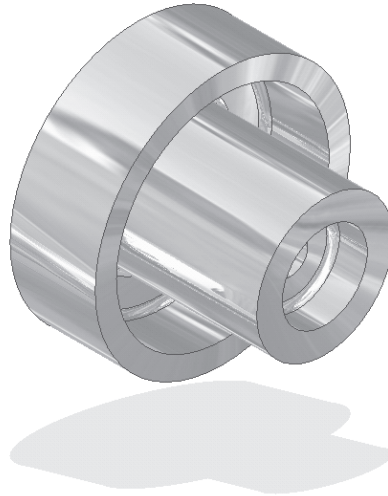


Figure 3-49 Model with the ground shadow

X-Ray Ground Shadow



The X-ray ground shadow shows the features of the model. To cast this type of shadow, choose the down arrow on the right of the **No Ground Shadow** button in the **Inventor Standard** toolbar and then choose the **X-Ray Ground Shadow** button from the flyout. The flat shadow of the model appears below it. Figure 3-50 shows a model with this kind of shadow. Notice the details of the features in the shadow.

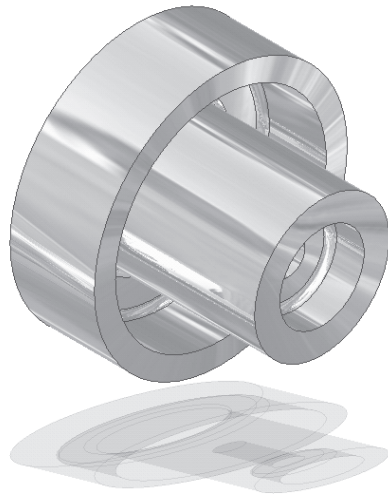


Figure 3-50 Model with the X-ray ground shadow



Note

You can cast shadows only below the model and not on any other plane. Also, the distance of the shadow from the model is modified as you zoom the model.

TUTORIALS

Tutorial 1

In this tutorial, you will open the sketch drawn in Tutorial 1 of Chapter 2. You will then convert this sketch into a solid model by extruding it to a distance of 20 mm. After creating the solid model, you will change its material to **Metal-Steel (Polished)** and rotate its view in 3D space using the **Rotate** tool. **(Expected time: 30 min)**

Before you start creating the model, it is recommended that you outline the steps that will be required to complete the tutorial. The following steps are required to complete this tutorial:

- Save the sketch from the *c02* folder to the *c03* folder with another name.
- Open the sketch from the *c03* folder and extrude it to a distance of 20 mm using the **Extrude** tool, refer to Figure 3-53.
- Change the material of the model by using the drop-down list in the **Inventor Standard** toolbar.
- Rotate the model in 3D space using the **Rotate** tool, refer to Figure 3-54.

Opening the Sketch Drawn in Chapter 2

1. Start Autodesk Inventor by double-clicking on its shortcut icon on the desktop of your computer or by using the **Start** menu. The **Open** dialog box will be displayed after Autodesk Inventor is started.
2. Choose **Open** to display the **Open File - Select a file to open** option.
3. Open the `\PersonalProject\c02` folder and then open the *Tutorial1.ipt* file.

When you open an existing part drawing, you are by default in the **Part** module even if the drawing is just a sketch. Also, the sketch is displayed in the isometric view, see Figure 3-51.

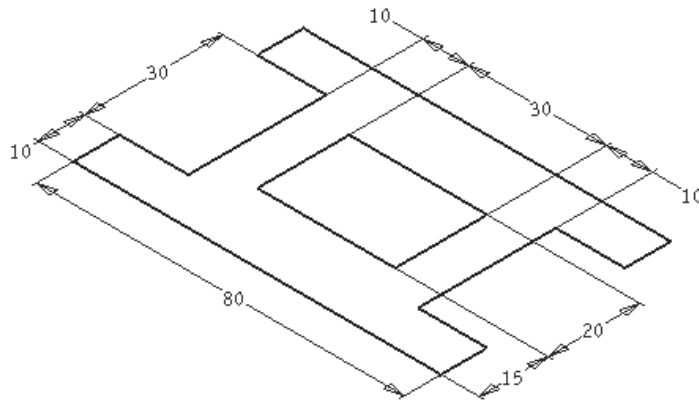


Figure 3-51 The sketch displayed in Isometric view when opened

Saving the Sketch with Another Name

After opening the sketch, you need to first save it with another name so that the original sketch drawn in Chapter 2 is not modified. Remember that when you save the file with another name in Autodesk Inventor, a new file is created and saved, but the original file is still open. You need to close the original file and open the new file using the **Open** dialog box.

1. Choose **File > Save Copy As** from the menu bar; the **Save Copy As** dialog box is displayed.
2. Select the `\PersonalProject\c03` folder from the **Save in** drop-down list.



Tip. If you have not created the `c03` folder, you can create it using the **Save Copy As** dialog box also. Select the `PersonalProject` folder from the **Save in** drop-down list. It will now be displayed in the **Save in** drop-down list. Choose the **Create New Folder** button and specify `c03` as the name of the folder.

3. Save the sketch with the name *Tutorial1.ipt*.
4. Now, choose **File > Close** from the menu bar to close this file. If you are prompted to specify whether or not you want to save the changes in this file, choose **No**.

- Choose the **Open** button from the **Inventor Standard** toolbar to display the **Open** dialog box. Open the file *\\PersonalProject\\c03\\Tutorial1.ipt*.

Extruding the Sketch

As mentioned earlier, when you open an existing part file, you are by default in the part modeling environment and the sketch is displayed in the Isometric view. Sometimes when you open an existing sketch, the dimensions of the sketch are not displayed in the current view. You can use the **Zoom** tool to increase the drawing display area so that all the dimensions are displayed in the current view. Now, because you are in the part modeling environment, all the tools of this module are available in the **Part Features** panel bar.

- Choose the **Extrude** button from the **Part Features** panel bar to invoke the **Extrude** dialog box.



Because the sketch consists of two loops, the sketch is not automatically selected. The **Profile** button in the **Shape** tab is chosen and you are prompted to select the profile to be extruded.

- Select the profile to be extruded by defining a point outside the inner loop but inside the outer loop; the profile will be highlighted, as shown in Figure 3-52.

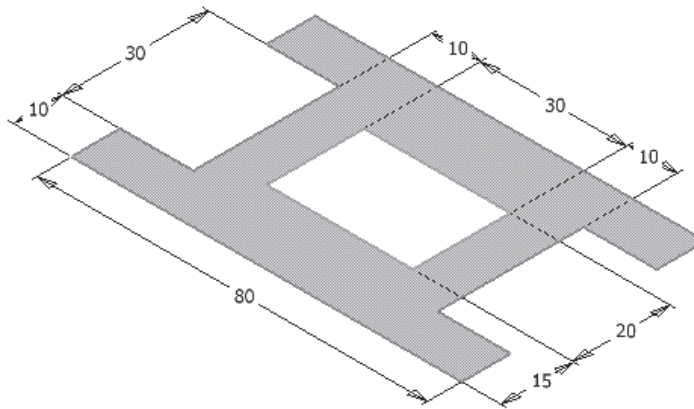


Figure 3-52 Selecting the profile to be extruded

As evident in Figure 3-51, the inner loop is not selected. This suggests that the selected profile when extruded will have a cavity in the center. This cavity is defined by the inner loop.

- Choose the **Profile** button again; the preview of the extruded model is displayed in the drawing window.
- Enter **20** in the **Depth** edit box available in the **Extents** area.

You will notice that the depth of the model in the preview will be increased, as the original depth was 10 mm.

5. Choose the **OK** button to create the model and exit the **Extrude** tool. The extruded model is shown in Figure 3-53.

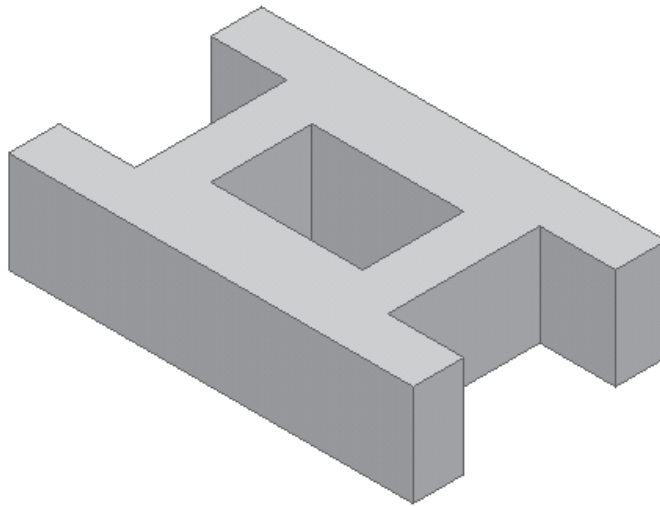



Figure 3-53 Resulting model created by extruding the profile

Changing the Material of the Model

When a model is created, it is applied a default material. However, as mentioned earlier, you can change this material and use one of the materials that are provided in Autodesk Inventor. Note that you do not need to select the model and then select the new material. You can directly select the required material from the drop-down list in the **Inventor Standard** toolbar and the material will be applied to the model.

1. Select **Metal-Steel (Polished)** from the drop-down list in the **Inventor Standard** toolbar; the color of the model will change to the new selected color.

Rotating the View of the Model in 3D Space

1. Choose the **Rotate** button from the **Inventor Standard** toolbar; the rim with four handles is displayed. 
2. Move the cursor inside the rim and then drag it to rotate the view of the model freely in 3D space, see Figure 3-54.
3. Now, move the cursor outside the rim and drag it to rotate the model. Notice the difference between rotating the view by dragging inside the rim and dragging outside the rim.

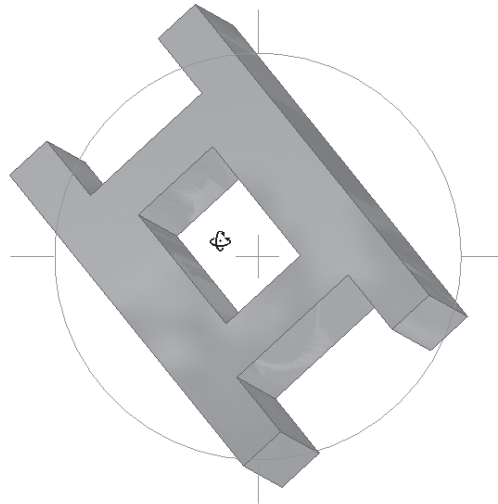


Figure 3-54 Rotating the view of the model freely in 3D space

4. Right-click to display the shortcut menu and choose **Done** to exit this tool.
5. Again, right-click and choose **Isometric View** from the shortcut menu to change the current view to isometric view.

Saving the Model

1. Choose the **Save** button in the **Inventor Standard** toolbar to save the model. Because the file is already saved once, the **Save As** dialog box is not displayed.
2. Choose **File > Close** from the menu bar to close this file.

Tutorial 2

In this tutorial, you will open the sketch drawn in Tutorial 2 of Chapter 2 and then convert it into a fully revolved model. Next, you will change the camera type to perspective and then view the model. **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

- a. Open the sketch from the *c02* folder and save it in the *c03* folder.
- b. Open the sketch from the *c03* folder and revolve it through an angle of 360-degree by using the **Revolve** tool.
- c. Change the camera type from the **Inventor Standard** toolbar and view the model.

Opening the Sketch Drawn in Chapter 2

1. Choose the **Open** button from the **Inventor Standard** toolbar to display the **Open** dialog box.

2. Using this dialog box, open the file `\PersonalProject\c02\Tutorial2.ipt`; the sketch will be displayed in the **Part** module in the isometric view.

Saving the Sketch with Another Name

1. Choose **File > Save Copy As** from the menu bar. The **Save Copy As** dialog box is displayed.
2. Browse to the `\PersonalProject\c03` folder and save the sketch with the name *Tutorial2.ipt*.

Because the current file is the original Chapter 2 file, you need to close it and open the Chapter 3 file.

3. Choose **File > Close** from the menu bar to close this file. Now, choose the **Open** button to display the **Open** dialog box and open the file `\PersonalProject\c03\Tutorial2.ipt`.

The dimensions of the sketch may not be displayed completely inside the current drawing display area. You may have to increase the drawing display area using the **Zoom** and **Pan** tools to fit the dimensions in the current view, see Figure 3-55.

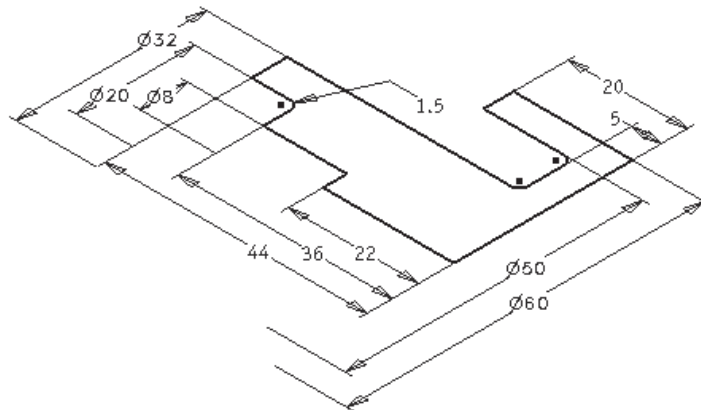


Figure 3-55 Sketch displayed in the Isometric view

Revolving the Sketch

As mentioned earlier, you need an axis of revolution to revolve a profile. This axis can be a sketched line segment. In this case, the revolution axis will be the line that measures 22 mm and the sketch will be revolved about it.

1. Choose the **Revolve** button from the **Part Features** panel bar to display the **Revolve** dialog box.



Because the sketch has just one loop, it is automatically selected and highlighted. Also, the **Profile** button in the **Shape** area is not chosen. Instead, the **Axis** button is chosen.

2. Select the bottom horizontal line that measures 22 mm as the axis of revolution.

When you move the cursor close to this line, it will be highlighted and will turn red. As soon as you select this line, the preview of the revolved model will be displayed in the drawing window.

3. Accept the default values and choose the **OK** button to complete the process of creating the revolved model.

Changing the View of the Model

The current view, in which the model is displayed, does not show the model properly. Therefore, you will have to change the view of the model.

1. Choose the **Rotate** button from the **Inventor Standard** toolbar; the rim is displayed in the drawing window.



You will use the shortcut menu of this tool to invoke the common views for displaying the model.

2. Right-click to display the shortcut menu. Choose **Common View** from this shortcut menu to display a cube with arrows in various directions.
3. Move the cursor close to the arrow on the upper left vertex on the left face of the cube, see Figure 3-56; the arrow turns red in color. Select this arrow.

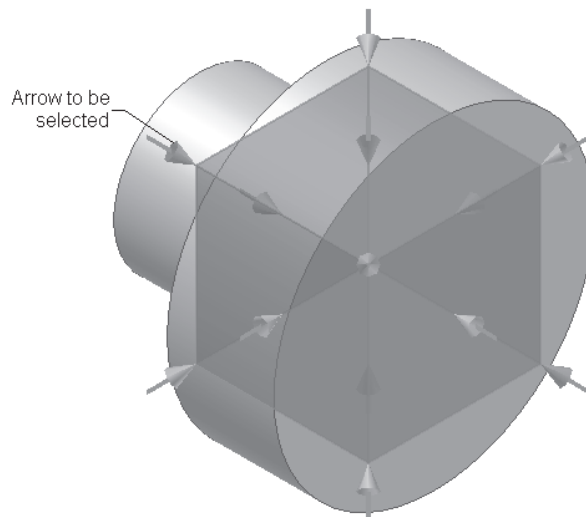


Figure 3-56 Selecting the **Common View** options to reorient the model

4. When you select this arrow, the revolved model will be reoriented and you will get a better view of the model. Right-click and choose **Done** to exit this tool.
5. Choose the **Zoom All** button from the **Inventor Standard** toolbar to modify the drawing display.

Changing the Camera Type

By default, the orthographic camera type is used to display the model. In this type of viewing, the parallel lines in the model do not meet at any point. This is the reason it is also termed as parallel viewing. The second type of camera that is available for displaying the model is perspective. In this type of viewing, the model in the drawing window is displayed exactly as it would be displayed in real 3D space. In this type of viewing, the lines in the model, when extended, meet at three points. Therefore, this type of viewing is also called the three-point perspective viewing.

1. Choose the down arrow on the right of the **Orthographic Camera** button in the **Inventor Standard** toolbar. From the flyout, choose the **Perspective Camera** button; the model will now be displayed using the perspective camera, see Figure 3-57.

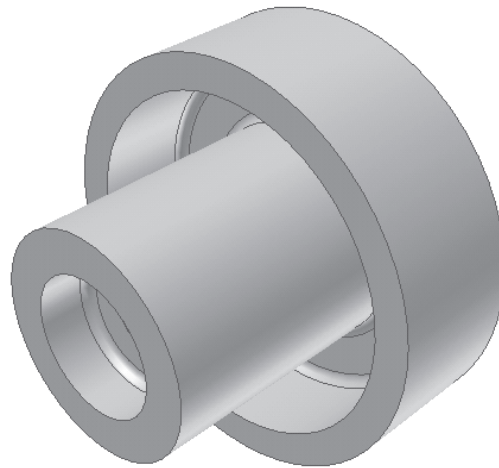


Figure 3-57 Model displayed using the perspective camera

Saving the Model

1. Choose the **Save** button in the **Inventor Standard** toolbar to save the model.
2. Now, choose **File > Close** from the menu bar to close this file.

Tutorial 3

In this tutorial, you will create the model shown in Figure 3-58. Its dimensions are given in Figure 3-59. The extrusion height for the model is 10 mm. After extruding it, you will set the option of casting the X-ray ground shadow. **(Expected time: 45 min)**

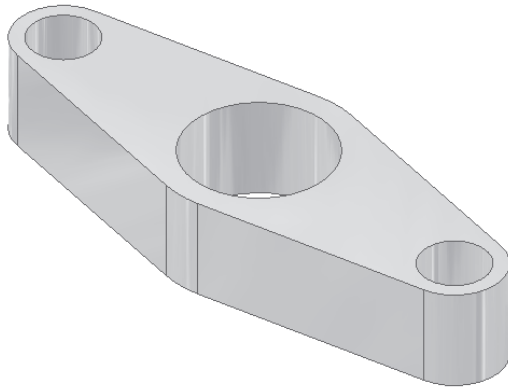


Figure 3-58 Model for Tutorial 3

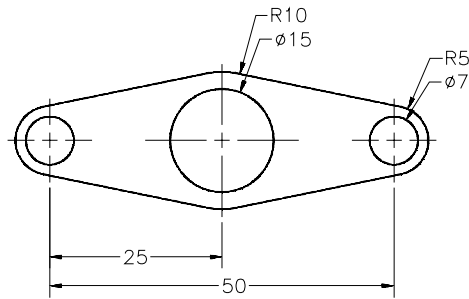


Figure 3-59 Dimensions for the model

The following steps are required to complete this tutorial:

- Start a new metric standard part file. Draw the sketch of the outer loop and add the constraints.
- Draw the inner circles and add the required constraints. Dimension the complete sketch, refer to Figure 3-62.
- Extrude the sketch to a distance of 10 mm using the **Extrude** tool, refer to Figure 3-63.
- Choose the **X-Ray Ground Shadow** button to cast the X-ray ground shadow, refer to Figure 3-64.

Starting a New Part File

If you have installed Autodesk Inventor with millimeter as the unit of measurement, you can directly start a new metric standard part file, thus avoiding the use of the **Open** dialog box for opening a new part file. This is done using the down arrow on right of the **New** button in the **Inventor Standard** toolbar. Note that this method is not applicable for starting a metric template if you have not installed Autodesk Inventor with millimeter as the unit of measurement.

- Choose the down arrow on the right of the **New** button in the **Inventor Standard** toolbar; a flyout is displayed with the buttons for part, assembly, drawing, and presentation files.
- Choose the **Part** button to start a new metric part file. If Autodesk Inventor was not installed with millimeter as the measurement unit, you need to use the **Metric** tab of the **Open** dialog box to start the new metric standard part file.

Creating the Sketch of the Model

As shown in Figure 3-58, the sketch is a combination of an outer loop and three circles. First, you will create the outer loop. This outer loop will be created by drawing three circles, two at the ends and one at the center. Next, you will draw tangent lines that will join the left circle with the middle circle and the middle circle with the right circle. Finally, you will trim the unwanted portions of the circles.

1. Draw the sketch, which is a combination of three circles and tangent lines. Add the **Tangent** constraint to the lines wherever it is missing. Also, add the **Equal** constraint to all the four lines and to the circles on the left and the right. The sketch after drawing and adding the constraints should look similar to the one shown in Figure 3-60.

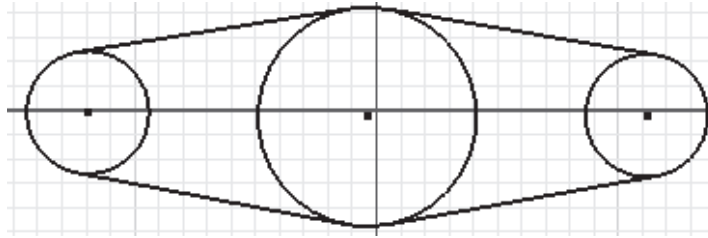


Figure 3-60 Sketch after adding all the constraints

Next, you need to remove the unwanted portions of the circles using the **Trim** tool.

2. Choose the **Trim** button from the **2D Sketch Panel** panel bar; you are prompted to select the portion of the curves to be trimmed.



3. Move the cursor close to the right half of the circle on the left side of the sketch.

As you move the cursor close to the circle, the circle will turn red in color and the right portion of the circle will be changed into dashed lines. This suggests that if you select the circle at this point, the portion displayed as dashed lines will be trimmed. In this case, the upper and lower left tangent lines will be taken as the cutting edges.

4. Specify a point on the right half of the circle that is on the left side of the sketch. The portion on the right of this circle will be trimmed. Similarly, select portions of the other circles also to trim, as shown in Figure 3-61.
5. Now, taking the center point of the trimmed arcs, draw the three circles. Add the **Equal**

constraint to the smaller circles. The sketch, after drawing the circles and applying the **Equal** constraint, should look similar to the one shown in Figure 3-62.

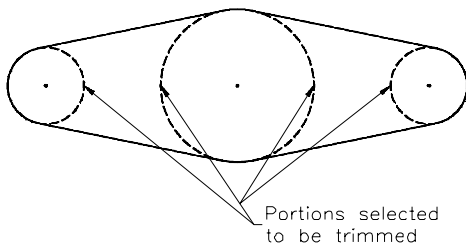


Figure 3-61 Selecting the portions to be trimmed

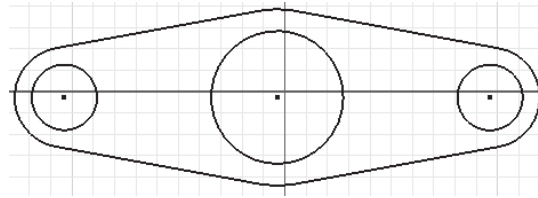


Figure 3-62 Sketch after creating the outer loop and the three circles

Dimensioning the Sketch

1. Add the required dimensions to the sketch. The sketch, after adding all the dimensions, should look similar to the one shown in Figure 3-63. Note that you may also have to add additional constraints, such as the **Horizontal** constraint, between the center points of the inner circles.

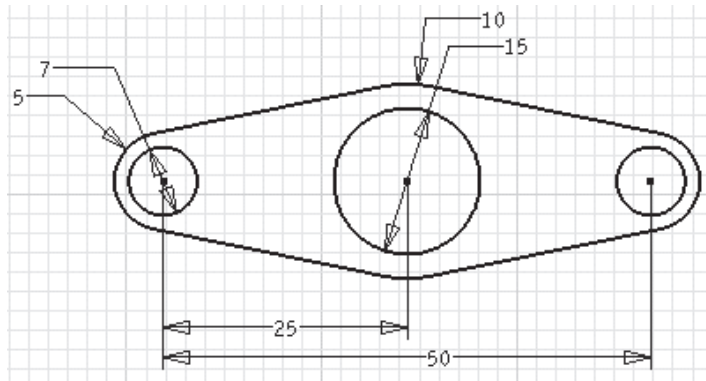


Figure 3-63 Sketch after adding dimensions


Extruding the Sketch

The sketch consists of four loops: the outer loop and the three circles. When you extrude this sketch, the three circles will be automatically subtracted from the outer loop. As a result, you will get the required model. However, this is possible only if you specify the point for selecting the profile inside the outer loop but outside all the three circles.

1. Choose **Return** on the **Inventor Standard** toolbar to exit the sketching environment.

As mentioned earlier, before proceeding to convert the sketch into a model, it is better to

change the current view to the isometric view. This is because you can preview the depth of the model only if you view the model from the isometric view.

2. Right-click in the drawing window to display the shortcut menu. Choose **Isometric View** to view the sketch from the isometric view.
3. Choose the **Extrude** button from the **Part Features** panel bar to invoke the **Extrude** dialog box. 

Because the sketch consists of more than one loop, the **Profile** button in the **Shape** area is chosen and you are prompted to select the profile to be extruded.

4. Select the profile to be extruded by specifying the selection point anywhere inside the outer loop but outside all the three circles.

The selected profile is shaded. Notice that the area inside all the three circles is not shaded. This suggests that the area inside these circles will not be extruded. This is also one of the methods to cross check whether the profile selected is the one you need to extrude or not.

5. Choose the **Profile** button again to proceed with the process of extruding the model.

The preview of the model is displayed in the drawing window.

6. Accept the default values to extrude the profile through a depth of **10 mm**.

You may need to change the camera type to parallel from the **Inventor Standard** toolbar if you use the same session of Autodesk Inventor in which you changed the camera type to perspective in Tutorial 2. The final model is shown in Figure 3-64.

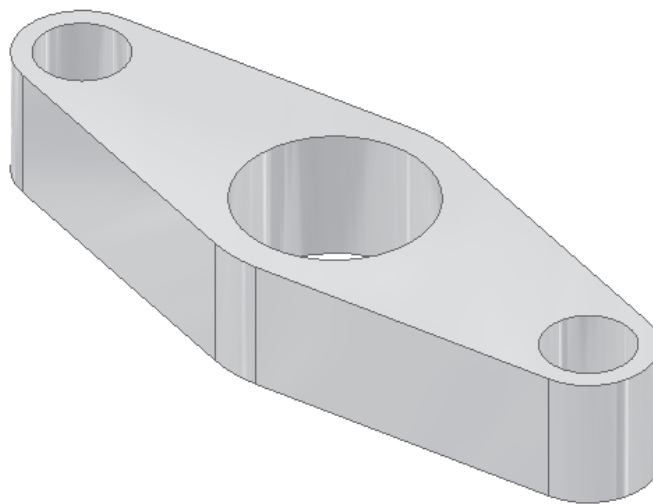


Figure 3-64 Final model for Tutorial 3

Casting the Shadow

Next, you need to cast the X-ray ground shadow of the model. You can cast the shadow using the tool in the **Inventor Standard** toolbar.

1. Choose the down arrow on the right of the **No Ground Shadow** button in the **Inventor Standard** toolbar; a flyout is displayed.
2. Choose the **X-Ray Ground Shadow** button from the flyout; the X-ray ground shadow will be displayed, as shown in Figure 3-65.

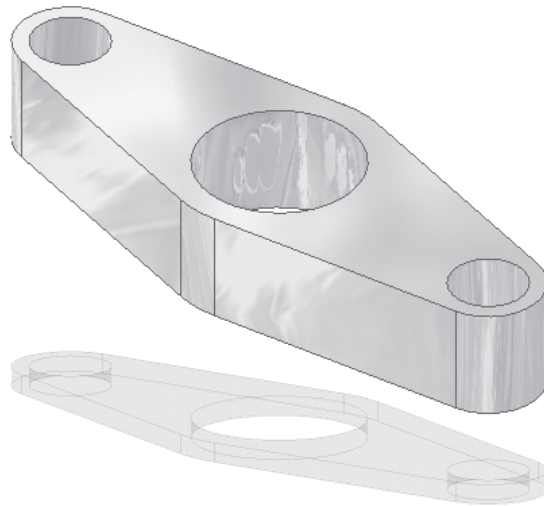


Figure 3-65 Model with the X-ray ground shadow

Saving the Model

1. Choose the **Save** button in the **Inventor Standard** toolbar.
2. Save the model with the name:
`\\PersonalProject\\c03\\Tutorial3.ipt`
3. Choose **File > Close** from the menu bar to close the file.

Tutorial 4

In this tutorial, you will write the text shown in Figure 3-66 and then cast the shadow of the text. The font size is 5 mm and the height of extrusion of the text is 2.5 mm. Change the material of the extruded text to Zinc Chromate. **(Expected time: 15 min)**

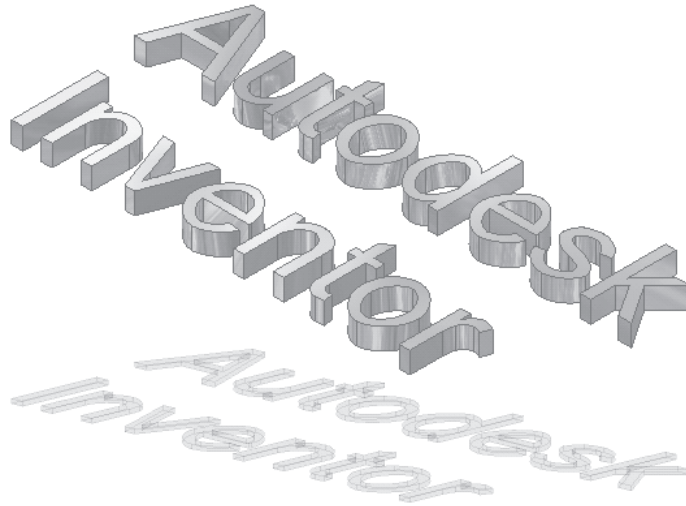


Figure 3-66 Extruded text with the shadow

The following steps are required to complete this tutorial:

- a. Start a new metric standard part file and write the text in the sketching environment.
- b. Extrude the text to a distance of 2.5 mm using the **Extrude** tool.
- c. Change the material of the feature to Zinc Chromate.
- d. Choose the **X-Ray Ground Shadow** button to cast the X-ray ground shadow.

Starting a New File and Writing the Text


1. Choose the down arrow on the right of the **New** button in the **Inventor Standard** toolbar. A flyout is displayed with the buttons for part, assembly, drawing, and presentation files.
2. Choose the **Part** option to start a new metric part file. If Autodesk Inventor was not installed with millimeter as the measurement unit, you need to use the **Metric** tab of the **Open** dialog box to start the new metric part file. Using the button and flyout in the **Inventor Standard** toolbar, make sure that the option to display the shadow is turn off.
3. Choose the **Create Text** button from the **2D Sketch Panel** panel bar; you are prompted to click on the location of the text. 
4. Specify a point somewhere in the drawing window; the **Format Text** dialog box is displayed.
5. Select the **Arial** font from the **Text Font** drop-down list and then set the font height to 5 mm using the **Text Height** edit box.
6. Enter the text **Autodesk Inventor** in the **Text Window** in two lines. Choose **OK** to exit the dialog box; the text is written and is displayed in the drawing window, as shown in Figure 3-67.



Figure 3-67 Text in the sketching environment

Extruding the Text

Next, you need to exit the sketching environment and extrude the text. But before you invoke the **Extrude** tool, you need to change the current view to isometric view.

1. Choose **Return** from the **Inventor Standard** toolbar and exit the sketching environment.
2. Right-click and choose **Isometric View** from the shortcut menu to change the current view to isometric view.
3. Choose the **Extrude** button from the **Part Features** panel bar to invoke the **Extrude** dialog box.



The profile button in the **Shape** tab is chosen and you are prompted to select the profile to extrude.

4. Move the cursor over the text and select it when it turns red.
5. Set the extrusion depth in the **Depth** edit box to **2.5** and choose **OK**. The text is extruded to a distance of 2.5 mm as shown in Figure 3-68.

Changing the Material and Casting the Shadow

As mentioned earlier, the material of the feature is changed using the drop-down list in the **Inventor Standard** toolbar.

1. Select **Zinc Chromate** from the drop-down list in the **Inventor Standard** toolbar. This is the last material in this list.

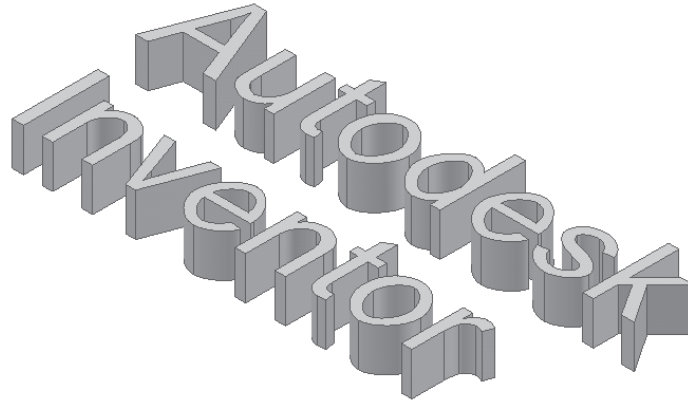


Figure 3-68 Feature created by extruding the text

The material of the text is changed to Zinc Chromate. Next, you need to cast the X-ray ground shadow.

2. Choose the down arrow beside the **No Ground Shadow** button in the **Inventor Standard** toolbar; a flyout is displayed.
3. Choose the **X-Ray Ground Shadow** button to cast this type of shadow below the extruded text. The final extruded text after casting the X-ray shadow is shown in Figure 3-69.

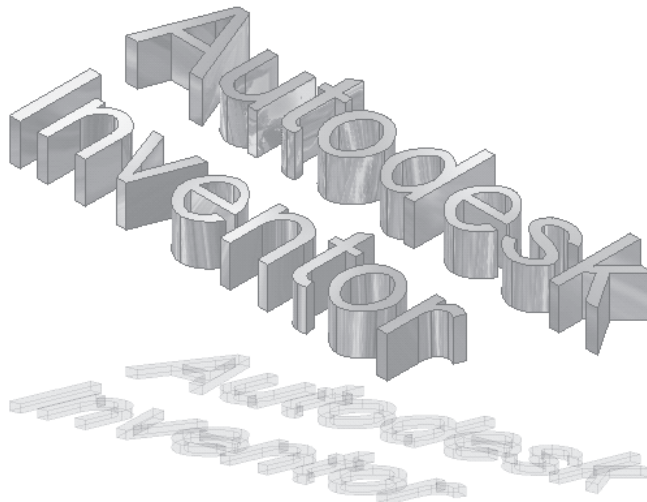


Figure 3-69 Text after changing the material and casting the shadow

**Note**

The distance of the feature from the shadow depends on the current zoom factor. As a result, the distance of shadow in your drawing may be different than that shown in Figure 3-68.

Saving the Sketch

1. Choose the **Save** button from the **Inventor Standard** tool bar; the **Save As** dialog box is displayed. Save the model with the name given below.

`\\PersonalProject\\c03\\Tutorial4.ipt`

2. Choose **File > Close** from the menu bar to close the file.

Self-Evaluation Test

Answer the following questions and then compare your answers with those given at the end of this chapter.

1. The line, once drawn in Autodesk Inventor, cannot be reduced in length. You will have to delete the line and then redraw a line of smaller length. (T/F)
2. The copies of the sketched entities can be arranged along the length and width of an imaginary rectangle using the **Circular Pattern** tool. (T/F)
3. When you select the profile by using a point that is inside the outer loop but outside the inner loops, the resultant solid will have the inner closed loops subtracted from the outer closed loop. (T/F)
4. To mirror the sketched entities, you necessarily require a mirror line. (T/F)
5. The _____ option should be cleared from the shortcut menu to select only one entity to offset from a closed loop.
6. To create a copy of an existing sketched entity by rotating, select the _____ check box in the **Rotate** dialog box.
7. Autodesk Inventor allows you to create two types of patterns. These are _____ patterns and _____ patterns.
8. The _____ option allows you to select a line segment, the length of which will define the distance between the individual items.
9. The _____ angles are generally provided to solid models for their easy withdrawal from the castings.
10. If the _____ button in the **Extrude** dialog box is chosen, the resulting feature will be a surface and not a solid.

Review Questions

Answer the following questions.

1. You can invoke the **Trim** tool from within the **Extend** tool by pressing the SHIFT key. (T/F)
2. Offsetting is one of the easiest methods of drawing parallel lines or concentric arcs and circles. (T/F)
3. If you select a circle from a point on its circumference and drag, it will be moved from its location. (T/F)
4. Selecting a line at its endpoint and dragging will stretch it. (T/F)
5. The preview of a model, as it will be created after extruding or revolving, is available in the drawing window even before you exit the **Extrude** or the **Revolve** dialog box. (T/F)
6. Which one of the following tools can be used to reposition the sketched entity from one place to the other using two points?
 - (a) **Move**
 - (b) **Rotate**
 - (c) **Mirror**
 - (d) **Extend**
7. Which one of the following tools can be used to arrange multiple copies of the sketched entities around an imaginary circle?
 - (a) **Move**
 - (b) **Rotate**
 - (c) **Rectangular Pattern**
 - (d) **Circular Pattern**
8. Which one of the following options allows you to use an existing dimension to define the distance between the individual items of a pattern.
 - (a) **Dimension**
 - (b) **Show Dimensions**
 - (c) **Measure**
 - (d) **None**
9. Which one of the following check boxes is selected to ensure that all items in the pattern are automatically updated, if any one of the entities is modified.
 - (a) **Associative**
 - (b) **Fitted**
 - (c) **Suppress**
 - (d) **None**
10. Which one of the following options is added to the shortcut menu after you choose the **Previous View** option from it?
 - (a) **Isometric View**
 - (b) **Common View**
 - (c) **Pan**
 - (d) **Next View**

Exercises

Exercise 1

In this exercise, you will extrude the sketch drawn in Exercise 1 of Chapter 2, see Figure 3-70. The extrusion depth for the model is 15 mm. **(Expected time: 30 min)**

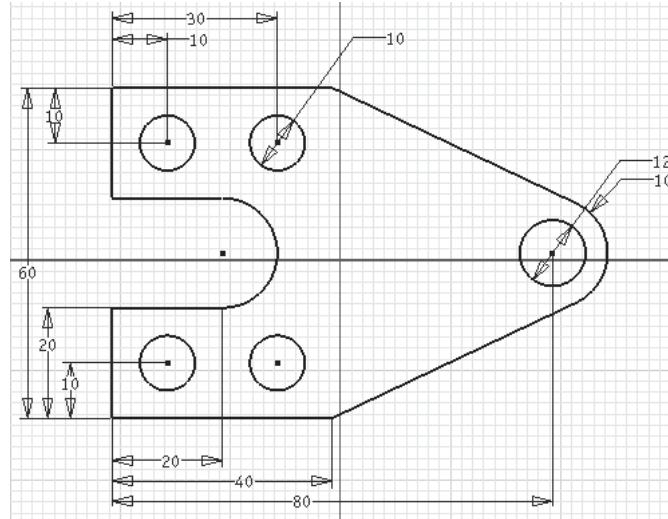


Figure 3-70 Sketch for Exercise 1

Exercise 2

In this exercise, you will extrude the sketch drawn in Exercise 2 of Chapter 2, see Figure 3-71. The extrusion depth for the model is 80 mm. **(Expected time: 30 min)**

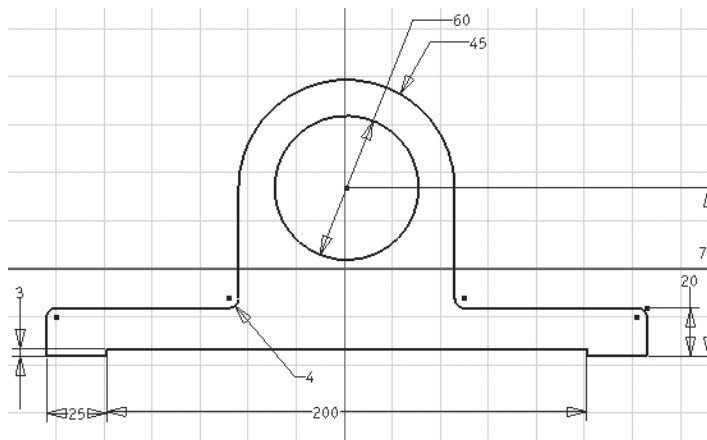


Figure 3-71 Sketch for Exercise 2

Exercise 3

In this exercise, you will extrude the sketch drawn in Exercise 3 of Chapter 2, see Figure 3-72. The extrusion depth for the model is 40 mm. (Expected time: 30 min)

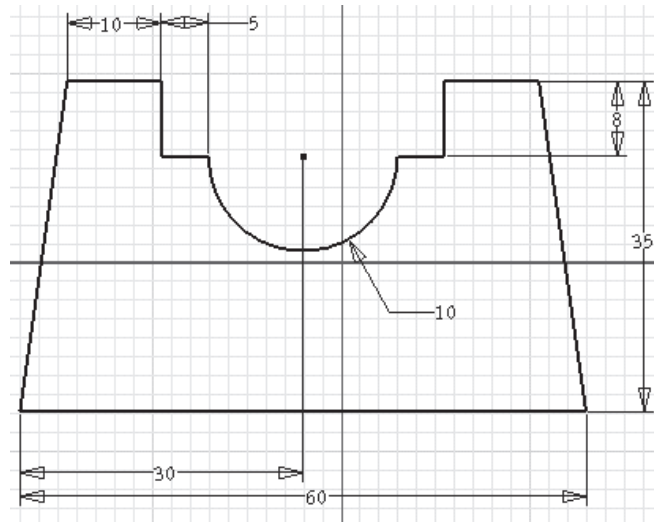


Figure 3-72 Sketch for Exercise 3

Exercise 4

In this exercise, you will extrude the sketch drawn in Exercise 4 of Chapter 2, see Figure 3-73. The extrusion depth for the model is 35 mm. (Expected time: 30 min)

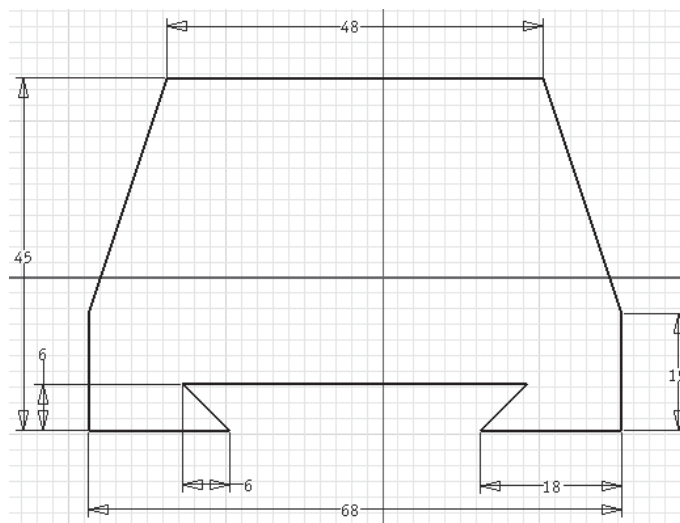


Figure 3-73 Sketch for Exercise 4

Exercise 5

In this exercise, you will extrude the sketch drawn in Exercise 5 of Chapter 2, see Figure 3-74. The extrusion depth for the model is 65 mm. **(Expected time: 30 min)**

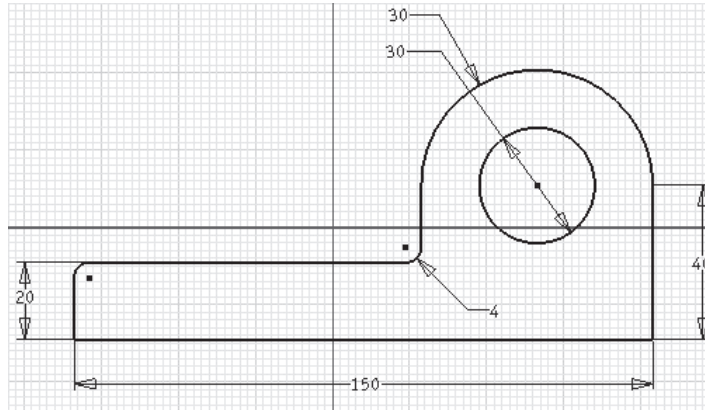


Figure 3-74 Sketch for Exercise 5



Note

When you extrude the sketches of Exercises 2, 3, 4, and 5 and change the view to isometric, the model will not be displayed as in the figures of exercises in Chapter 2. The reason is that these sketches are created on the default XY plane, which is horizontal. You will learn to draw sketches on other planes in the next chapter.

Answers to Self-Evaluation Test

1. F, 2. F, 3. T, 4. T, 5. Loop Select, 6. Copy, 7. Rectangular, Circular, 8. Measure, 9. taper, 10. Surface