Chapter 4

Advance Dimensioning and Base Feature Options

Learning Objectives

After completing this chapter you will be able to:

- *Dimension the sketch using the autodimension sketch tool.*
- *Dimension the sketch using the ordinate dimensioning.*
- *Dimension the true length of arc.*
- *Measure Distances and View Section Properties*
- *Create solid base extruded features.*
- *Create thin base extruded features.*
- *Create solid base revolved features.*
- *Create thin base revolved features.*
- *Dynamically rotate the view to display the model from all directions.*
- *Modify the orientation of the view.*
- *Change the display modes of the solid model.*

ADVANCE DIMENSIONING TECHNIQUES

In this chapter, you will learn about some of the advance dimensioning techniques used in dimensioning the sketches in SolidWorks. With this release of SolidWorks, you are able to apply all the possible dimensions to a sketch using a single option. This option is known as **Autodimension Sketch**. The other dimensions options discussed in this chapter are horizontal dimensioning, vertical dimensioning, ordinate dimensioning, and dimensioning true length of an arc. The advanced dimensioning techniques are discussed next.

Autodimension the Sketches

Toolbar:	Sketch Relations > Autodimension Sketch	(*Customize to Add*)
Menu:	Tools > Dimensions > Autodimension Sketch	

The **Autodimension Sketch** option is used to automatically apply the dimensions to the sketch. You can apply the absolute dimension, incremental dimension, and ordinate dimension using this option. To apply autodimensions to a sketch, create the sketch using standard sketching tools and then apply the required relations to the sketch. Now, choose **Tools > Dimensions > Autodimension Sketch** from the menu bar. The **Autodimension sketch PropertyManager** is displayed as shown in Figure 4-1. The various options available in the **Autodimension Sketch PropertyManager** are discussed next.

Entities to Dimension

The **Entities to Dimension** rollout is used to specify the entities on which the dimension has to be applied. The **All entities in sketch** radio button is selected to apply the dimension to all the entities drawn in the current sketching environment. This radio button is selected by default if you do not select any entity before invoking the **Autodimension sketch PropertyManager**. The **Selected entities** radio button is selected if you have to dimension only the selected entities. When you select this radio button, the **Selected Entities to Dimension** display area is displayed in the **Entities to Dimension** rollout. Select the entities to dimension using the select cursor. The name of the selected entities is displayed in the **Selected Entities to Dimension** display area. If you select any entity or entities before invoking the **Autodimension sketch PropertyManager**, then the **Selected entities** radio button is selected by default and the name of the selected entities will be displayed in the **Selected Entities to Dimension** display area.

Horizontal Dimensions

The **Horizontal Dimensions** rollout is used to specify the type of horizontal dimension, reference for the horizontal dimension, and the dimension placement. The various options available in the **Horizontal Dimensions** rollout are discussed next.

Scheme

The **Scheme** area is used to specify the type of dimension to be applied to the sketch. The various types of dimensioning schemes available in the **Horizontal Dimensioning Scheme** drop-down list are discussed next.

Chain

The **Chain** option is used for the relative or incremental horizontal dimensioning of the sketch. When you invoke the **Autodimension sketch PropertyManager** and select
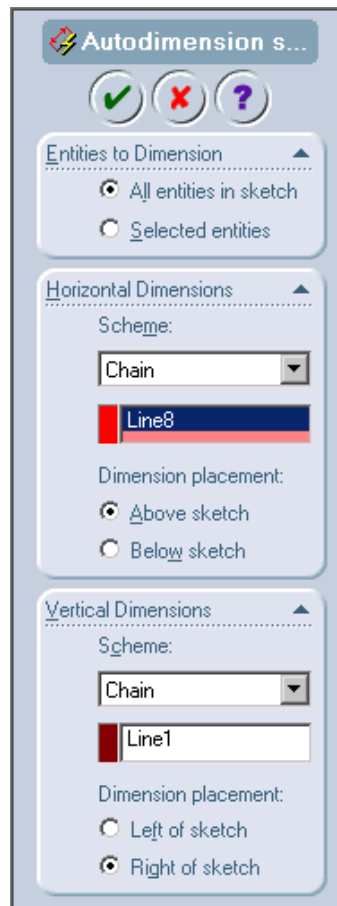
*Figure 4-1 The **Autodimension sketch PropertyManager***

this scheme, a point or a vertical line is selected as the reference entity. This reference entity is used as a datum for the generation of dimension. The name of selected reference entity is displayed in the **Point or Vertical Line on Baseline** display area and the reference entity is displayed in red color in the drawing area. You can also specify a user-defined reference entity.

Note

Chain dimensioning should be avoided if the tolerances relative to a common datum are required in the part.

Baseline

The **Baseline** option is used for absolute or datum vertical dimensioning of the sketch. In this dimensioning method, the dimensions are applied to the sketch with respect to the common datum. When you invoke the **Autodimension sketch PropertyManager** and select this option, a point or a vertical line is selected as the reference entity, which is used as a datum for the generation of dimension. The name of selected reference entity is displayed in the **Point or Vertical Line on Baseline** display area and the

reference entity is displayed in red color in the drawing area. You can also specify a user defined reference entity.

Ordinate
The **Ordinate** option is used for the ordinate dimensioning of the sketch. When you invoke the **Autodimension Sketch PropertyManager** and select this option, a point or a vertical line is selected as the reference entity, which is used as a datum for the generation of dimension. The name of selected reference entity is displayed in the **Point or Vertical Line on Ordinate Datum** display area and the reference entity is displayed in red color in the drawing area. You can also specify a user defined reference entity.

Dimension placement
The **Dimension Placement** area of the **Horizontal Dimensions** rollout is used to define the position where the generated dimensions will be placed. Two radio buttons are available in this area. The first radio button is the **Above sketch** radio button and is selected by default. If you use this option, the horizontal dimensions generated using the **Autodimension Sketch** tool will be placed above the sketch. If you select the **Below sketch** radio button, the generated dimensions will be placed below the sketch.

Vertical Dimensions
The **Vertical Dimensions** rollout is used to specify the type of vertical dimension, reference for the vertical dimension, and the dimension placement. The various options available in the **Scheme** area are similar to those discussed under the **Horizontal Dimensions** rollout. The remaining options are discussed next.

Dimension placement
The **Dimension Placement** area of the **Vertical Dimensions** rollout is used to define the position where the generated dimensions will be placed. The **Left of the Sketch** radio button is selected to place the dimensions on the left of the sketch. The **Right of the Sketch** radio button is selected to place the dimensions on the right of the sketch. The **Right of the Sketch** radio button is selected by default.

After specifying all the parameters in the **Autodimension Sketch PropertyManager**, choose the **OK** button or choose the **OK** icon from the **Confirmation Corner**. The dimension will be created with the selected dimension scheme. Figure 4-2 shows the autodimension created using the **Chain** scheme. Figure 4-3 shows the autodimension created using the **Baseline** scheme. Figure 4-4 shows the autodimension created using the **Ordinate** scheme.

Note
*As evident in Figures 4-2 through 4-4, the dimensions added using the **Autodim** option are not properly arranged. You need to manually place them at their proper location.*
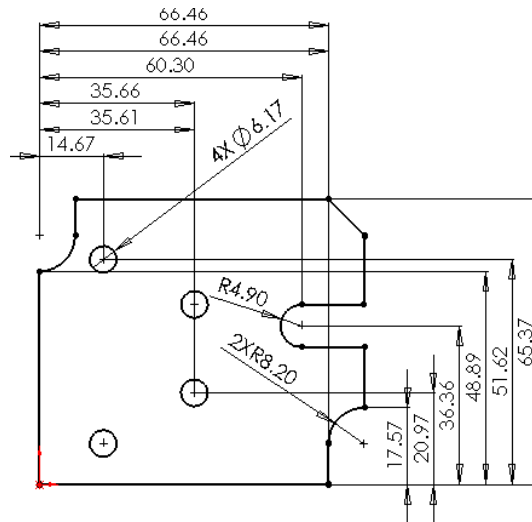
Ordinate Dimensioning of Sketches

Menu: Tools > Dimensions > Ordinate

The **Ordinate** option of dimensioning is extensively used in industry for the dimensioning of shop floor drawings. This is because this type of drawing interprets the drawing in the coordinate
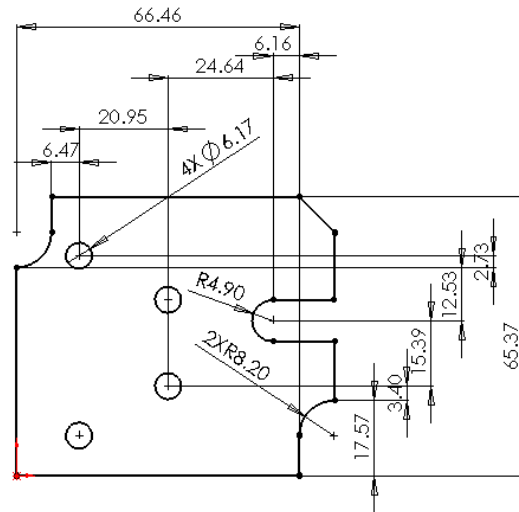
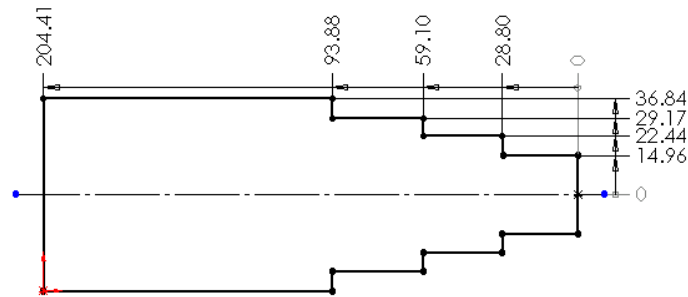*Figure 4-2* *Chain dimension created using autodimension option*

Figure 4-3 *Baseline dimension created using autodimension option*

form and the coordinates are required as input for the NC and CNC machines. In the ordinate dimensioning you have to define a zero (datum); and all the dimensions will be created with respect to that zero.

Note

If you want ordinate dimensions in the drawing views then you have to create the ordinate dimensions in the sketch itself, because the dimensions created in the sketches and in part mode are generated in drawing mode when you opt for generative dimensioning.

Figure 4-4 *Ordinate dimension created using autodimension option*

Tip. *Choose **Tools > Dimensions > Parallel** from the menu bar to create the parallel dimension. Using this option you can create the horizontal as well as the vertical dimensions. But you cannot create a aligned dimension using the parallel option.*

*Choose **Tools > Dimensions > Horizontal** from the menu bar to create the horizontal dimension. Using this option you can create only the horizontal dimensions. Generally, this option is used to create a horizontal dimension of an aligned line or a horizontal dimension between two points.*

*Choose **Tools > Dimensions > Vertical** from the menu bar to create the vertical dimension. Using this option you can create only the vertical dimensions. Generally, this option is used to create a vertical dimension of an aligned line or a vertical dimension between two points.*

After creating the sketch and applying the required relations to the sketch, choose **Tools > Dimensions > Ordinate** from the menu bar. The select cursor is replaced by the ordinate dimension cursor. For creating the horizontal ordinate dimension, select a vertical line or a point where you have to define the zero or datum. As soon as you select the line or a point, a dimension is attached to the cursor. Now, move the cursor and place the dimension. This datum dimension is a reference dimension; therefore, you cannot change the value of this dimension. Now, select the line or point using the ordinate dimension cursor; the dimension is automatically placed. Continue selecting points or lines to place ordinate dimensions. After creating all the horizontal ordinate dimensions, choose the **Select** button. Now, again choose **Tools > Dimensions > Ordinate** from the menu bar to create the vertical ordinate dimensions. Select a point or a horizontal line to define the zero and then apply the ordinate dimensions to the sketch. Figure 4-5 shows a sketch with ordinate dimensions.
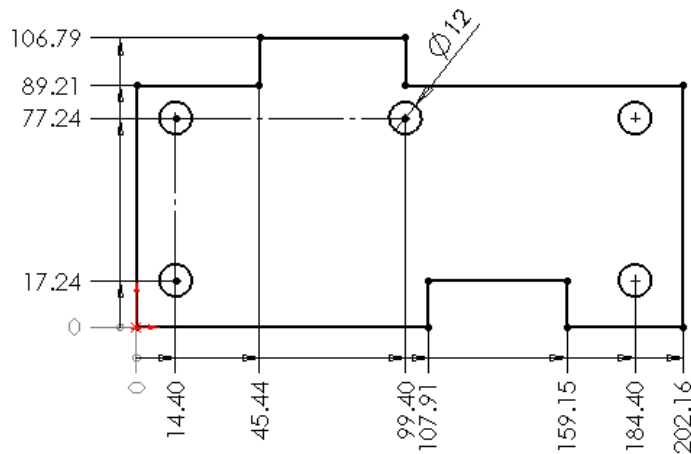
Figure 4-5 *Ordinate dimension of a sketch*

> **Tip**. *Using **Tools** > **Dimensions** > **Ordinate** you can create either horizontal or vertical ordinate dimensions. If you want to create only horizontal ordinate dimensions then choose **Tools** > **Dimensions** > **Horizontal Ordinate**. To create vertical ordinate dimensions choose **Tools** > **Dimensions** > **Vertical Ordinate**.*
>
> *When you are in the selection mode, right-click in the drawing area to invoke the shortcut menu and choose the **Dimension** option from the shortcut menu to invoke the dimension tool. Now, again right-click to invoke the shortcut menu and choose the **Ordinate Dimension** option from the shortcut menu to invoke the ordinate dimension tool. You can choose the **Horizontal Ordinate**, **Vertical Ordinate**, **Horizontal Dimension**, and **Vertical Dimension** tools from the same shortcut*

Dimensioning of the True Length of an Arc

In SolidWorks you can also create the dimension of the true length of an arc, which is one of the advantages of the sketching environment of SolidWorks. To create the dimension of the true length, invoke the dimension tools and select the arc using the dimension cursor. A radial dimension is attached to the cursor. Move the cursor to any of the endpoints of the arc. When the cursor snaps the endpoint, use the left mouse to specify the first endpoint of the arc. A linear dimension is attached to the cursor; move the cursor to the second endpoint of the arc and when the cursor snaps the endpoint, select it. A dimension is attached to the cursor. Move the cursor to an appropriate place to place the dimension. The dimension of the true length of the arc is shown in Figure 4-6.

MEASURING DISTANCES AND VIEWING SECTION PROPERTIES

SolidWorks allows you to measure the distance of the entities and also view the section properties. These tools are discussed next.
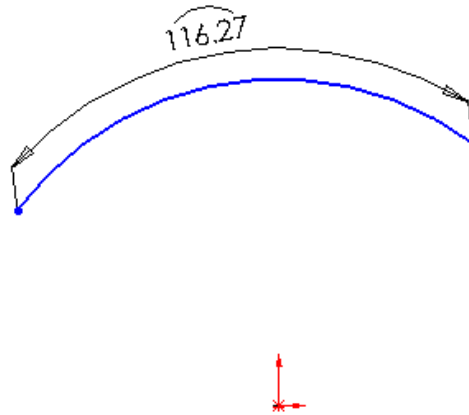
Figure 4-6 *Dimensioning the true length of an arc*

Measuring Distances

Toolbar:	Tools > Measure
Menu:	Tools > Measure

The measure tools is used to measure perimeter, angle, radius, and distance between lines, points, surfaces, and planes in sketches, 3D models, assemblies, or drawings. To use the measure tools, you have to invoke the **Measure** dialog box by choosing **Tools > Measure** from the menu bar or by choosing the **Measure** button from the **Tools** toolbar. When you choose the **Measure** button, the **Measure** dialog box is displayed. The name of the document in which you are working is displayed at the top of the **Measure** dialog box. The **Measure** dialog box is displayed in Figure 4-7. The current cursor is replaced by the measure cursor. Using the measure cursor select the entity or entities to measure. The various options in the **Measure** dialog box are discussed next.

Output coordinate system drop-down list

In the **Measure** dialog box you are provided with an **Output coordinate system** drop-down list. Using this drop-down list you can specify in reference to which coordinate system you want to measure the selected entities. By default, the **default** option is selected in the drop-down list. If you create a coordinate system, then you can also select that coordinate system from this drop-down list. Creation of the coordinate system is discussed in the later chapters.

Select items Area

The **Select items** area consists of a display box that displays the entities that are selected using the measure cursor. When you select the entities using the measure cursor, the name of the selected entity is displayed in the **Select items** area. You can remove all the entities from the selection set, and select one of the entity from the **Select items** display area. Select the **Clear Selections** option from the shortcut menu. If you select the **Delete** option from the shortcut menu then only the selected entity is removed from the selection set.
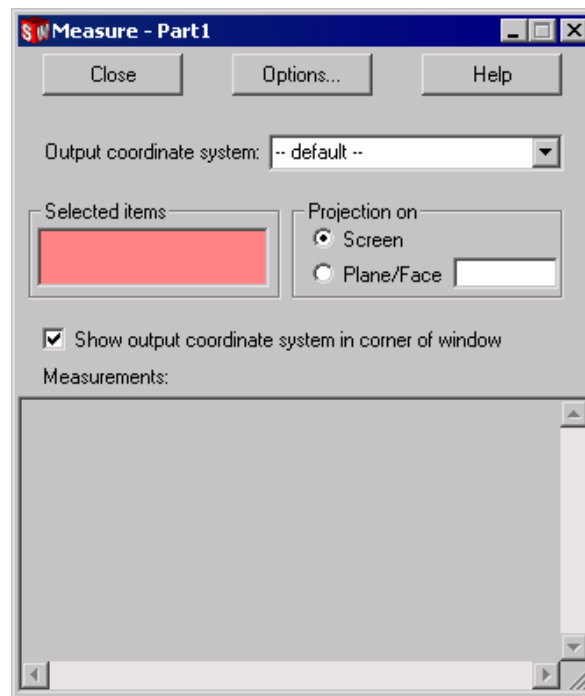
*Figure 4-7 The **Measure** dialog box*

Projection on

The **Projection on** area is used to specify where the selected entity should be projected. You can project the selected entity either on the screen or on the specific plane. Then the system will calculate the measurement of the true projection. This area is provided with two radio buttons; the first radio button is the **Screen** radio button and the second radio button is the **Plane/Face** radio button. You will learn more about planes in the later chapters.

Show output coordinate system in corner of window

The **Show output coordinate system in corner of window** check box is selected to display the coordinate system at the corner of the screen. This check box is selected by default; if you clear this check box then you will notice that the coordinate system is placed at the origin in the drawing area.

Measurements Area

After selecting the entity or entities to be measured using the measure cursor, you are provided with the appropriate values of the length, angle, perimeter, and so on displayed in the **Measurements** area. New measurements update dynamically when you change the selection of entities. If the combination of selected entities is not appropriate for the measure function, then a message is displayed in the **Measurements** area. This message prompts you that the selection is invalid. If two selected entities are intersecting lines then you are provided the value of the angle between the intersecting entities and the message prompts you that the selected items intersect.

Options

The **Options** button is used to invoke the **Measurement Options** dialog box. In this dialog box you can specify the units for linear measurement, units for angular measurement, material properties, and the accuracy level. The **Measurement Options** dialog box is shown in Figure 4-8. The options in the **Measurement Options** dialog box are discussed next.
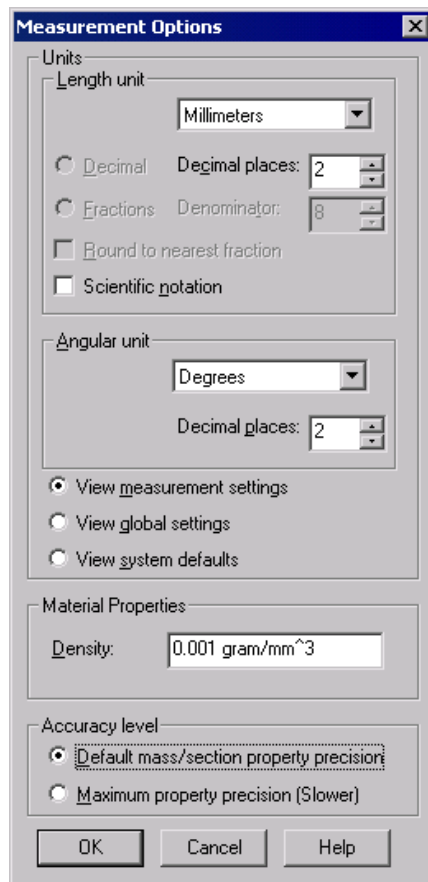

*Figure 4-8 The **Measurement Options** dialog box*

Units Area

The **Units** area of the **Measurement Options** dialog box is used to set the linear and the angular units for measurement. By default, the linear units are selected as millimeters with decimal upto two places, and the angular units are selected as with decimal upto two decimal. The various options in the **Units** area are discussed next.

Length unit Area

The **Length unit** area is used to set the units and options of linear dimensions for measuring the entities. The **Unit** drop-down list is provided at the top right corner of the **Length unit** area. In this drop-down list you can select any type of unit such as **Angstroms, Nanometers, Microns, Millimeters**, **Centimeters**, **Meters**, **Microinches, Miles, Inches**, **Feet**, and **Feet and Inches**. By default, the units selected in the

drop-down list are **Millimeters** because during installation you had selected millimeters as the default units. A **Decimal places** spin box is provided to control the decimal places. The other options in the **Length unit** area are discussed next.

Decimal. The **Decimal** radio button is available only when you choose the units as **Microinches, Miles, Inches**, or **Feets and Inches** from the **Unit** drop-down list. This radio button is selected to display the dimension in the decimal form. You can also specify the decimal places using the **Decimal places** spin box provided at the right of the **Decimal** radio button.

Fractions. The **Fractions** radio button is available only when you choose the units as **Microinches, Miles, Inches**, or **Feets and Inches** from the **Units** drop-down list. This radio button is selected to display the dimension in the fraction form. You can also set the value of the denominator using the **Denominator** spin box provided at the right of the **Fractions** radio button.

Round to nearest fraction. The **Round to nearest fraction** check box is selected to display the value in fractions by rounding the value to the nearest fraction.

Scientific notation. The **Scientific notation** check box is selected to display the value in the scientific notation units.

Angular unit Area
The **Angular unit** area is used to set the angular units for measurement. The **Angular unit** area is provided with a drop-down list to specify the angular measurement unit such as **Degrees**, **Deg/Min**, **Deg/Min/Sec**, and **Radians**. A **Decimal places** spin box is also provided under the drop-down list to specify the decimal places.

Mass Properties

View measurement settings
The **View measurement settings** radio button is selected to activate a temporary system of units. After selecting this radio button if you change the units, it will not affect the units specified for the document, or the default units of the system for new documents.

View global settings
The **View global settings** radio button is selected to display the units specified for the active document. If you change the unit setting in the **Length unit** and **Angular unit** areas and choose the **View global settings** radio button then the units will be reset to the default units of the active document.

View system defaults
The **View system defaults** radio button is selected to displays the default units of the system for new documents. If you change the unit setting in the **Length unit** and **Angular unit** areas and choose the **View system defaults** radio button then the units will be reset to the default units of the system and when you create a new document the default units will be displayed.

Material Properties area

In the **Material Properties** area you can change the density of the material for the current document. You can enter the density value using any units. For example, if the units of the part are grams and millimeters, you can enter a density value using pounds and inches. The SolidWorks converts the value to the document's units when you choose **OK** from the **Measurement Options** dialog box.

Accuracy level

The **Accuracy level** area of the **Measurement Options** dialog box is used to specify the accuracy level provided by SolidWorks for the measurement. The **Default mass/section property precision** radio button is selected by default; this option calculates faster but the precision is not very high. The **Maximum property precision (Slower)** radio button is selected to increase the accuracy and precision for measuring. This option when used slows the working of the computer but gives a very precise output result.

Section Properties

The **Section Properties** tool enables you to calculate the section properties of the sketch in the sketching environment or of selected planar face in the part mode and assembly mode. The section properties include the area, centroid relative to sketch origin, centroid relative to part origin, moment of inertia, polar moment of inertia, angle between the principle axes and sketch axes, and principle moment of inertia. To calculate the section properties, create the sketch. The sketch must be a closed loop. Choose **Tools** > **Section Properties**. The **Section Properties** dialog box is displayed as shown in Figure 4-9.
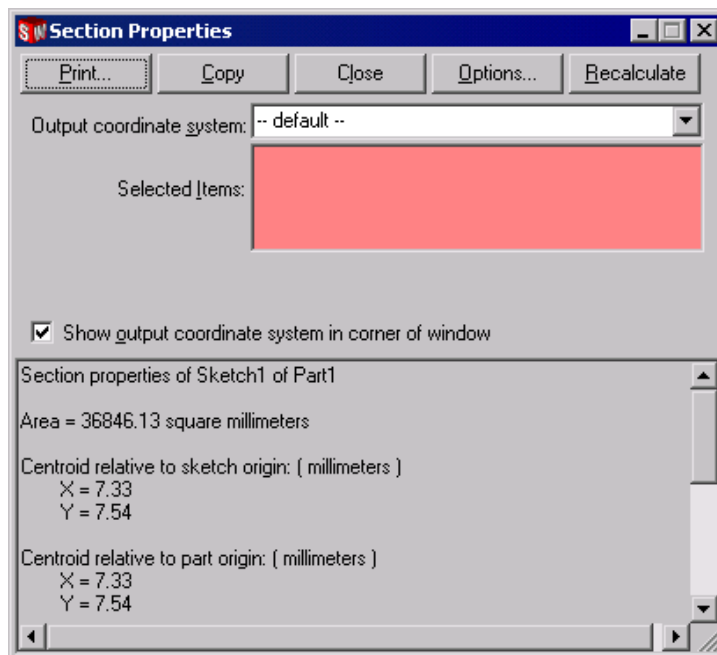
Figure 4-9 *The* **Section Properties** *dialog box*

When you invoke the **Section Properties** dialog box a red 3D triad is placed at the centroid of the sketch. The mass properties of the sketch are displayed in the **Section Properties** dialog box. The **Output coordinate system** drop-down list is used to select the coordinate system along which you want to calculate the section properties. By default, the calculation is done with respect to the default coordinate system. The **Selected Items** display box is used to display the name of the selected planar face whose section properties are to be calculated. When you are in the part mode select the face to calculate the section properties, and choose the **Recalculate** button to display the properties. If you want to calculate the section properties of some other face, clear the previously selected face from the selection set and select the new face and choose the **Recalculate** button. The **Print** button available in the **Section Properties** dialog box is used to print the section properties. Using the **Options** button you can invoke the **Measurement Options** dialog box. This dialog box is used to set the units, density, and accuracy level. This dialog box was discussed previously. The **Show output coordinate system in the corner of window** check box is selected to display the coordinate system at the corner of the drawing area; if you clear this check box then the coordinate system will shift to the origin.

CREATING BASE FEATURES BY EXTRUDING THE SKETCHES

|	
Toolbar:	Features > Extruded Boss/Base
Menu:	Insert > Base > Extrude

The sketches that you have drawn until now can be converted into base features by extruding using the **Extruded Boss/Base** tool. This tool is available in the **Features** toolbar. After drawing the sketch, as you choose this tool, you will notice that the sketching environment is closed and the part modeling environment is invoked and the confirmation corner is displayed. Based on the options and the sketch selected for extruding, the resultant feature can be a solid feature or a thin feature. If the sketch is closed, it can be converted into a solid feature or a thin feature. However, if the sketch is open, it can be converted into a thin feature only. The solid and thin features are discussed next.

Creating Solid Extruded Features

After you have completed drawing and dimensioning the closed sketch and converted it into a fully defined sketch, choose the **Extrude Boss/Base** button from the **Features** toolbar. You will notice that the view is automatically changed to a 3D view, the confirmation corner appears on the top right of the drawing area, and the **Extrude PropertyManager** is displayed as shown in Figure 4-10.

You will notice that the preview of the base feature will be displayed in temporary graphics and an arrow will appear on the sketch. The arrow appears on the front of the sketch and is transparent. Figure 4-11 shows the preview of the sketch being extruded. Note that if the sketch consists of some closed loops inside the outer loop, they will be automatically subtracted from the outer loop while extruding. The various options available in the **Base-Extrude PropertyManager** are discussed next.
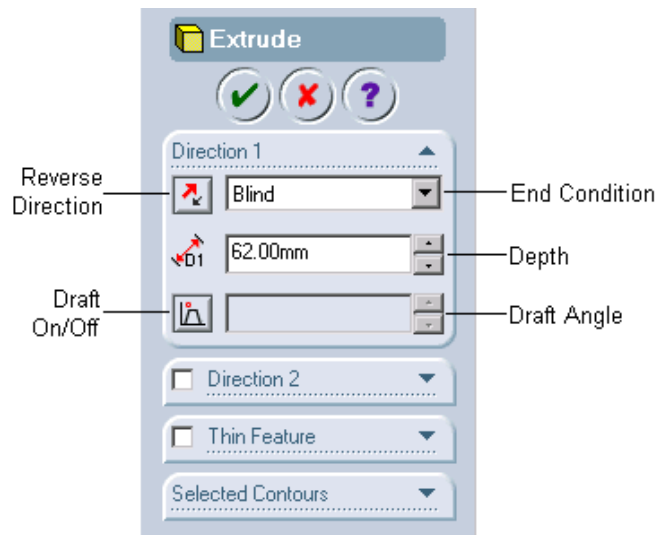
Figure 4-10 Extrude PropertyManager
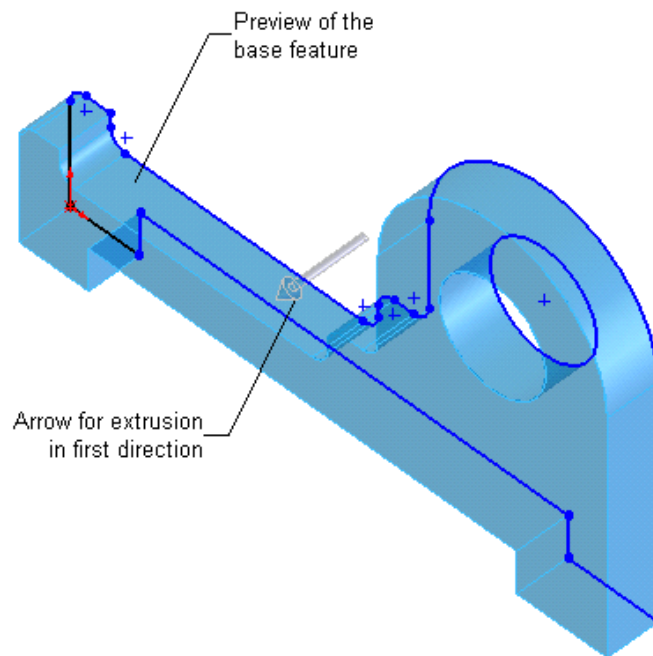
*Figure 4-11 Preview of the feature being extruded using the **Extrude-Base PropertyManager***

Direction 1

The **Direction 1** rollout is used to specify the end condition for extruding the sketch in one direction from the sketch plane. The various options available in the **Direction 1** drop-down list are discussed next.

End Condition

The **End Condition** drop-down list provides the options to define the termination of the extruded feature. Note that since this is the first feature, some of the options available in this drop-down list will not be used at this stage. Also, some additional options will be available later in this drop-down list. The options that are available to define the termination of the base feature are discussed next.

Blind

The **Blind** option is selected by default and this option is used to define the termination of the extruded base feature by specifying the depth of extrusion. The depth of extrusion can be specified in the **Depth** spinner that is displayed below this drop-down list when you select the **Blind** option. Since the **Blind** option is selected by default, the **Depth** spinner is also displayed by default. You can reverse the extrusion by selecting the **Reverse Direction** button provided on the left of this drop-down list. Figure 4-12 shows the preview of the feature being created by extruding the sketch using the **Blind** option.
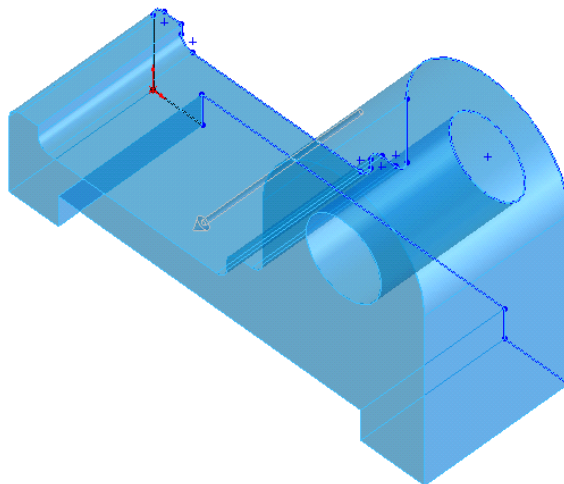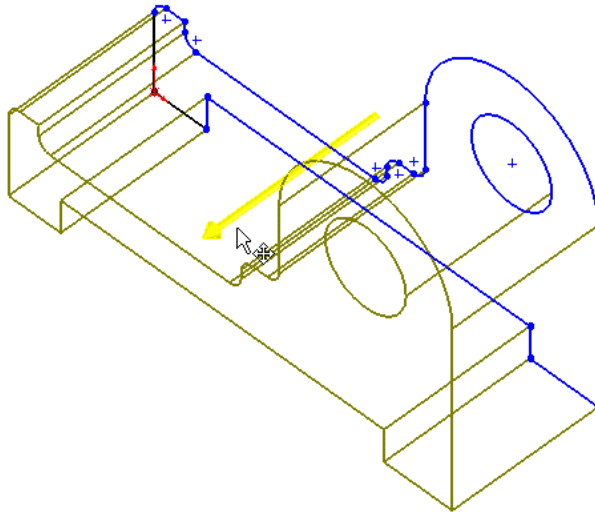

Figure 4-12 Preview of the feature being extruded

You can also extrude a sketch to a blind depth by dynamically dragging the feature using the mouse. Invoke the **Extrude-Base PropertyManager** and move the mouse to the transparent arrow and when the color of the arrow changes to red use the left mouse button. Move the cursor to specify the depth of extrusion and when you specify the depth of extrusion use the left mouse button to specify the termination of extruded feature. The preview of the sketch being dragged is shown in Figure 4-13. The select cursor will be replaced by the mouse cursor. Use the right mouse button to complete the feature creation or choose the **OK** button from the **Base-Extrude PropertyManager**.

Mid Plane

The **Mid Plane** option is used to create the base feature by extruding the base sketch equally in both the directions of the plane on which the sketch is drawn. For example,

Figure 4-13 *Preview of the feature being extruded by dynamically dragging*

Tip. *You can also extrude an under defined or an over defined sketch. However, if you extrude an under defined sketch, a - sign is displayed on the left of the sketch in the **Feature Manager**. Similarly, if you extrude an over defined sketch, you will find a + sign on the left of the sketch in the **Feature Manager**. To check these signs, click the + sign available on the left of **Base-Extrude** in the **Feature Manager**. The sketch will be displayed and you can see the + or the - signs.*

if the total depth of the extruded feature is 30mm, it will be extruded 15mm toward the front of the plane and 15mm toward the back. The depth of the feature can be defined in the **Depth** spinner that is displayed below this drop-down list.

Figure 4-14 shows the preview of the feature being created by extruding the sketch using the **Mid Plane** option.

Tip. *Right-click in the drawing area to display the shortcut menu. Using the left mouse button choose the **Mid Plane** option from the shortcut menu. Move the mouse and use the left mouse button to specify the depth of extrusion.*

Draft On/Off
The **Draft On/Off** button is used to specify a draft angle while extruding the sketch. Applying the draft angle will taper the resultant feature. This button is not chosen by default. Therefore, the resultant base feature will not have any taper. However, if you want to add a draft angle to the feature, choose this button. The **Draft Angle** spinner and the **Draft outward** check box will be available. You can enter the draft angle for the feature in the **Draft Angle** spinner. By default, the feature will be tapered inward as shown in Figure 4-15. If you want to taper the feature outward, select the **Draft outward** check box that is displayed below the **Draft Angle** spinner. The feature created with outward taper is shown in Figure 4-16.
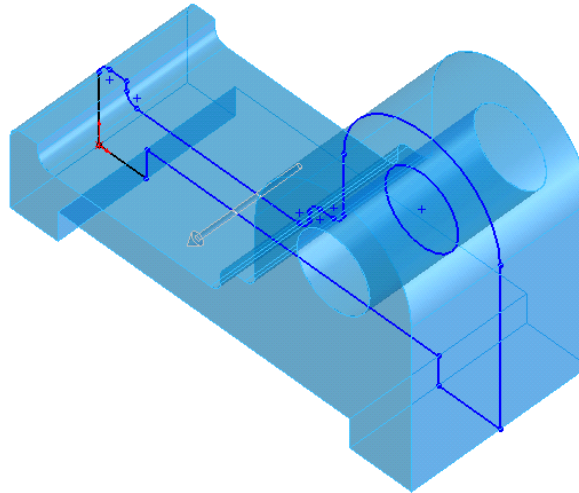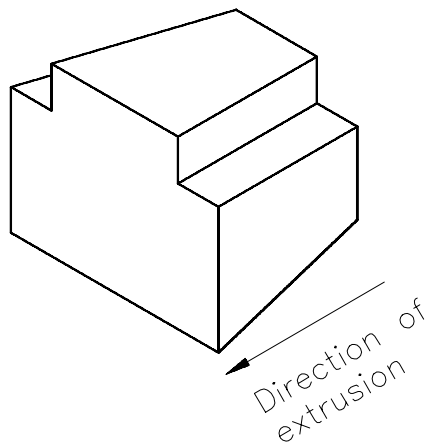
*Figure 4-14* *Preview of the feature being extruded using the **Mid Plane** option*

Note
*The **Selection Contours** rollout is discussed in the later chapters of this book.*


Figure 4-15 *Feature created with outward draft*

Direction 2

The **Direction 2** check box is selected to invoke the **Direction 2** rollout and this rollout is used to extrude the sketch with different values in the second direction of the sketching plane. This check box will not be available if you select the **Mid Plane** termination type. Unlike the **Mid Plane** termination option, the depth of extrusion and other parameters in both the directions can be different. For example, you can extrude the sketch to a blind depth of 10mm and an inward draft of 35° in front of the sketching plane and to a blind depth of 15mm and an outward
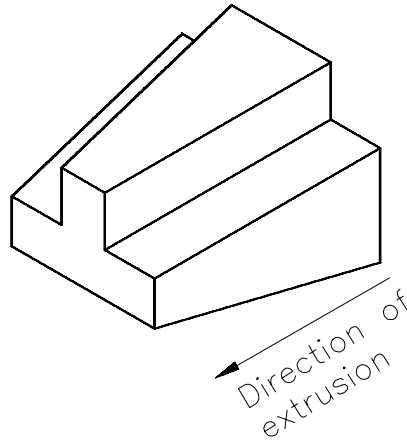
Figure 4-16 *Feature created with inward draft*

draft of 0° behind the sketching plane as shown in Figure 4-17. When you select the **Direction 2** check box the **Direction 2** rollout is activated. Using the various options available in the **Direction 2** rollout you can specify the termination condition for extrusion in the second direction. When you invoke this rollout the preview of direction 2 is displayed on the screen with the default values. After setting the values for both the directions, choose the **OK** button or choose the **OK** icon from the confirmation corner. The feature will be created with different values in both the directions.
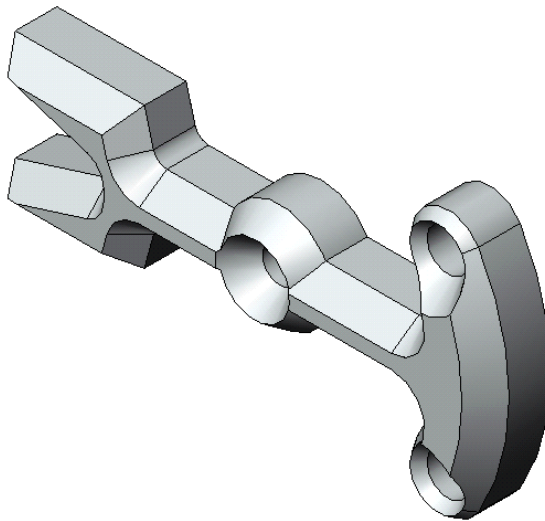
Figure 4-17 *Feature created in both directions with different values*
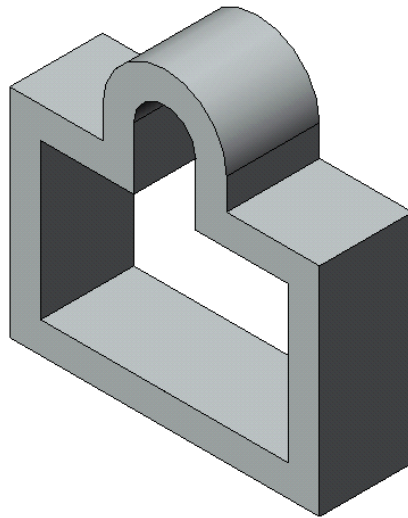
Note

The draft will not be displayed in the preview of the feature that is displayed when you invoke the
Base-Extrude PropertyManager *unless you extrude the sketch by dynamically dragging.*

Tip. *Select the direction 2 arrow from the drawing area and move the cursor to specify the depth of extrusion in the second direction and use the left mouse button to complete the feature creation.*

Creating Thin Extruded Features

The thin extruded features can be created using a closed or an open sketch. If the sketch is closed, it will be offsetted inside or outside to create a cavity inside the feature as shown in Figure 4-18.

Figure 4-18 *Thin feature created using a closed loop*

If the sketch is open, as shown in Figure 4-19, the resultant feature will be as shown in Figure 4-20. Note that you can also apply fillets at all the sharp corners of the open loop while creating thin features.

To convert a closed sketch into a thin feature, choose the **Thin Feature** checkbox to invoke the **Thin Feature** rollout. The **Thin Feature** rollout, shown in Figure 4-21, is used to create a thin feature. However, if the sketch to be extruded is open, then the **Thin Feature** rollout will be displayed when you invoke **Base-Extrude PropertyManager**.

The options under the **Thin Feature** rollout of the **Extrude PropertyManager** are discussed next.

Type

The options provided in the **Type** drop-down list are used to select the method to define the thickness of the thin feature. These options are discussed next.

One-Direction

The **One-Direction** option is used to add the thickness on one side of the sketch. The thickness can be specified in the **Thickness** spinner provided below this drop-down list.
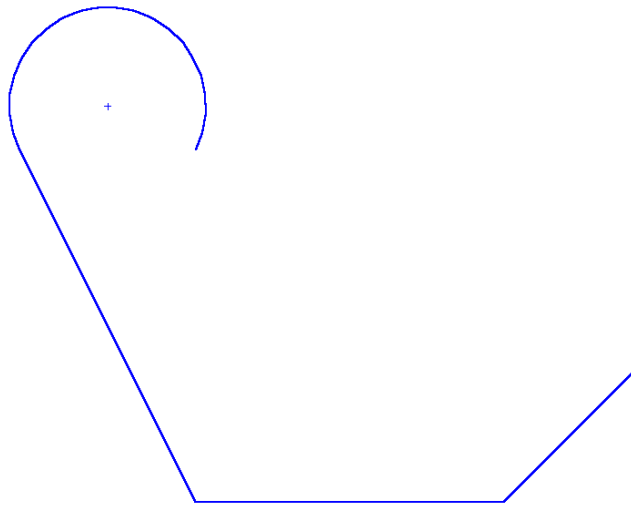
Figure 4-19 Open loop to be converted into thin feature
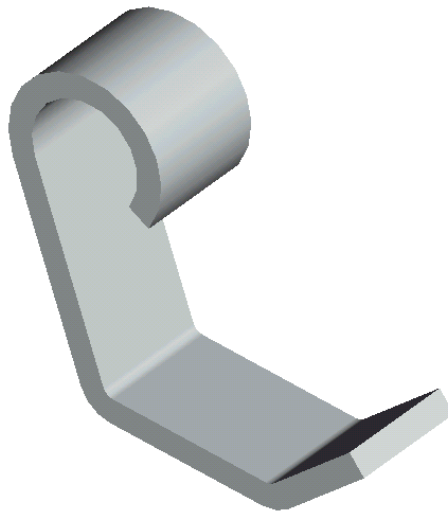
Figure 4-20 Resultant thin feature created with fillets at sharp corners

*Figure 4-21 The **Thin Feature** rollout*

For the closed sketches, the direction can be inside or outside the sketch. Similarly, for open sketches, the direction can be below or above the sketch. You can reverse the direction of thickness using the **Reverse** check button available on the right of this drop-down list. This check box will be available only when you select the **One-Direction** option from this drop-down list.

Mid-Plane

The **Mid-Plane** option is used to add the thickness equally on both the sides of the sketch. The value of the thickness of the thin feature can be specified in the **Thickness** spinner provided below this drop-down list.

Two-Direction

The **Two-Direction** option is used to create a thin feature by adding different thickness on both the sides of the sketch. The thickness values in direction 1 and direction 2 can be specified in the **Direction 1 Thickness** spinner and the **Direction 2 Thickness** spinner respectively. These spinners are displayed below the **Type** drop-down list automatically when you select the **Two-Direction** option from this drop-down list.

Cap Ends

The **Cap Ends** check box is displayed only when the sketch selected to convert into a thin feature is closed. This check box is selected to cap the two ends of the thin extruded feature. Both the ends will be capped with a face of the thickness you specify. When you select this check box, the **Cap Thickness** spinner is displayed on the right of this check box. The thickness of the end caps can be specified using this spinner.

Auto-fillet corners

The **Auto-fillet corners** check box is displayed only when you select an open sketch to convert into a thin feature. If you select this check box, all the sharp vertices in the sketch will be automatically filleted while converting into a thin feature. The radius of the fillet can be specified in the **Fillet Radius** spinner that is displayed below the **Auto-fillet corners** check box when you select this check box.

Figure 4-22 shows the thin feature created by extruding an open sketch in both the directions. Notice that a draft angle is applied to the feature while extruding in the front direction and the **Auto Fillet** option is selected while creating this thin feature.

Note
Only the corners of the thin features that can accommodate the given radius will be filleted; other corners that do not accommodate the given radius will not be filleted.

CREATING BASE FEATURES BY REVOLVING THE SKETCHES

The sketches that you have drawn until now can also be converted into base features by revolving using the **Revolved Boss/Base** tool. This tool is available in the **Features** toolbar. However, note that a sketch can be revolved only if you draw a centerline in the sketch around which the sketch will be revolved. Also, the sketch must be drawn on one side of the centerline. The **Revolved**

Figure 4-22 *Thin feature created in both the directions*

Boss/Base tool will be available only after you draw the centerline in the sketch. Note that the centerline around which you want to revolve the sketch should not cross the sketch.

After drawing the sketch, as you choose this tool, you will notice that the sketching environment is closed and the part modeling environment is invoked. Similar to extruding the sketches, the resultant feature can be a solid feature or a thin feature based on the sketch and the options selected to revolve. If the sketch is closed, it can be converted into a solid feature or a thin feature. However, if the sketch is open, it can be converted into a thin feature only. The solid and thin features are discussed next.

Creating Solid Revolved Features

Toolbar:	Features > Revolved Boss/Base
Menu:	Insert > Base > Revolve

After you have completed drawing and dimensioning the closed sketch and converted it into a fully defined sketch, choose the **Revolve Boss/Base** button from the **Features** toolbar. You will notice that the view is automatically changed to a 3D view, the **Revolve PropertyManager** will be displayed as shown in Figure 4-23, and the confirmation corner will also be displayed. Also, the preview of the base feature, as it will be created using the default options, will be displayed in temporary shaded graphics. The direction arrow will also be displayed in gray color. The various options available in the **Revolve Parameters** rollout of the **Revolve PropertyManager** are discussed next.

Revolve Type

The **Revolve Type** drop-down list provides the options to define the termination of the revolved feature. The options that are available in this drop-down list to terminate the revolved feature are discussed next.
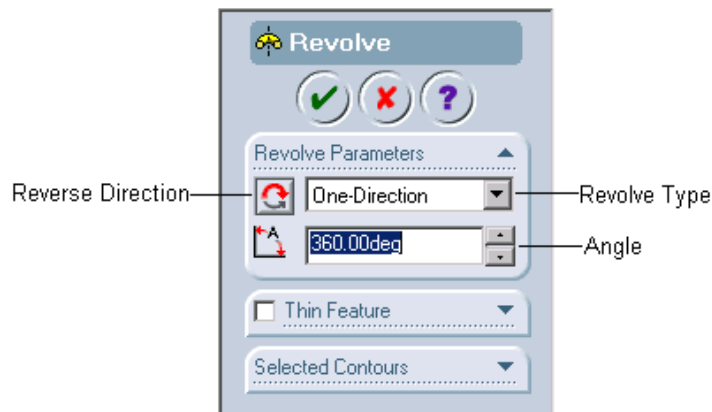
*Figure 4-23 The **Revolve PropertyManager***

> **Tip**. *If the preview of the revolved feature is not complete in the current display of the screen, choose the **Zoom to Fit** button from the **View** toolbar or choose the **F** key from the keyboard. The display will be modified such that the preview is displayed in the current view.*

One-Direction

The **One-Direction** option is used to revolve the sketch on one side of the plane on which it is sketched. The angle of revolution can be specified in the **Angle** spinner displayed below this drop-down list. The default value of the **Angle** spinner is **360deg**. Therefore, if you revolve the sketch using this value, a complete round feature will be created. You can also reverse the direction of revolution of the sketch by choosing the **Reverse Direction** button that is displayed when you select this option. Figure 4-24 shows the sketch and the centerline used to revolve the sketch and Figure 4-25 shows the piston created by revolving the sketch through an angle of 360°.

Figure 4-26 shows a piston created by revolving the same sketch through an angle of 270°.

Mid-Plane

The **Mid-Plane** option is used to revolve the sketch equally on both the sides of the plane on which it is sketched. The angle of revolution can be specified in the **Angle** spinner. When you choose this option the **Reverse Direction** button will be unavailable.

Two-Direction

The **Two-Direction** option is used to create a revolve feature by revolving the sketch using different values on both the sides of the plane on which it is sketched. The angle values in direction 1 and direction 2 can be specified in the **Direction 1 Angle** spinner and the **Direction 2 Angle** spinner respectively. These spinners are displayed below the **Revolve Type** drop-down list automatically when you select the **Two-Direction** option from this drop-down list.
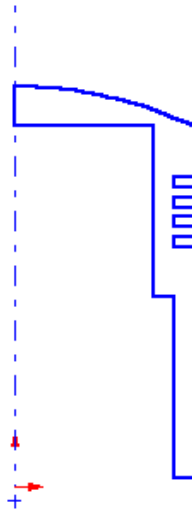
***Figure 4-24*** *Sketch to be revolved and the centerline around which the sketch will be revolved*

Figure 4-25 *Feature created by revolving the sketch through an angle of 360°*

Note

If you create the centerline of the sketch feature for creating a revolve feature from right to left, the sketch will be revolved in the clockwise direction when you create the revolve feature. If you create the centerline of the sketch feature from left to right, then the resultant revolve feature will revolve in counterclockwise direction.
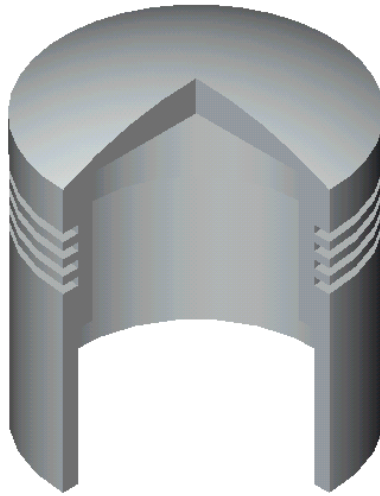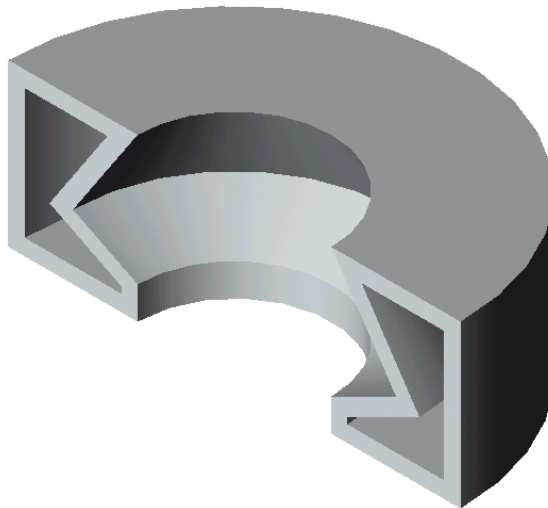
Figure 4-26 *Feature created by revolving the sketch through an angle of 270°*

Creating Thin Revolved Features

Similar to the thin extruded features, the thin revolved features can be created using a closed or an open sketch. If the sketch is closed, it will be offsetted inside or outside to create a cavity inside the feature as shown in Figure 4-27. In this figure, the sketch is revolved through an angle of 180°.
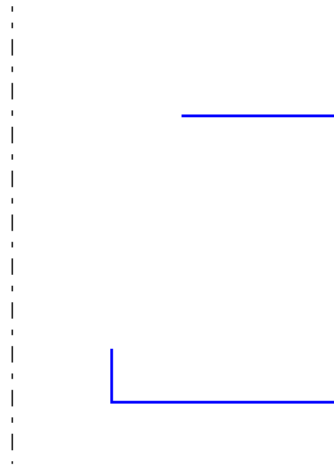

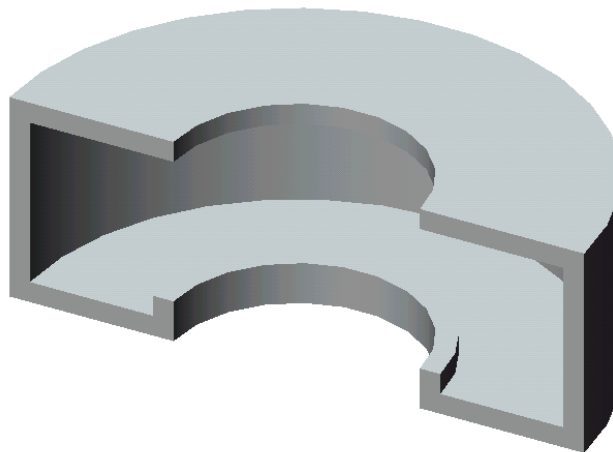*Figure 4-27* *Thin feature created by revolving the sketch through an angle of 180°*

> **Tip**. *You can dynamically specify the angle in a revolve feature by dragging the direction arrows. You can also use the right mouse button to display the shortcut menu; all the options available in the **PropertyManager** are also available in the shortcut menu.*

If the sketch is open, as shown in Figure 4-28, the resultant feature will be as shown in Figure 4-29.

Figure 4-28 *The open sketch to be revolved and the centerline to revolve the sketch*

Figure 4-29 *Thin feature created by revolving the open sketch through an angle of 180°*

To convert a closed sketch into a thin feature, select the **Thin Feature** check box from **Base-Revolve PropertyManager** to invoke the **Thin Feature** rollout. However, if the sketch to be revolved is open and you invoke the **Revolve Boss/Base** tool, then the **SolidWorks** information box will be displayed and you will be informed that the sketch is currently open and a nonthin

> **Tip**. *While defining the wall thickness of a thin revolved feature, remember that the wall thickness should be added such that the centerline does not intersect with the sketch. If the centerline intersects with the sketch, the sketch will not be revolved.*

revolve feature requires a closed sketch. You will be given an option of automatically closing the sketch. If you choose **Yes** from this dialog box, a line segment will be automatically drawn between the first and the last segment of the sketch and the **Base-Revolve PropertyManager** will be displayed. However, if you choose **No** from this dialog box, the **Base-Revolve PropertyManager** will be displayed and the **Thin Feature** rollout will be displayed automatically. The options under the **Thin Feature** rollout of the **Base-Revolve PropertyManager**, shown in Figure 4-30, are discussed next.

*Figure 4-30 The **Thin Feature** rollout*

Type

The options provided in the **Type** drop-down list are used to select the method to define the thickness of the thin feature. These options are discussed next.

One-Direction

The **One-Direction** option is used to add the thickness on one side of the sketch. The thickness can be specified in the **Direction 1 Thickness** spinner provided below this drop-down list. For the closed sketches, the direction can be inside or outside the sketch. Similarly, for open sketches, the direction can be below or above the sketch. You can reverse the direction of thickness using the **Reverse Direction** button available on the right of this drop-down list. This button will be available only when you select the **One-Direction** option from this drop-down list.

Mid-Plane

The **Mid-Plane** option is used to add the thickness equally on both the sides of the sketch. The value of the thickness of the thin feature can be specified in the **Direction 1 Thickness** spinner provided below this drop-down list.

Two-Direction

The **Two-Direction** option is used to create a thin feature by adding different thickness on both the sides of the sketch. The thickness values in direction 1 and direction 2 can be specified in the **Direction 1 Thickness** spinner and the **Direction 2 Thickness** spinner respectively. These spinners are displayed below the **Type** drop-down list automatically when you select the **Two-Direction** option from this drop-down list.

DYNAMICALLY ROTATING THE VIEW OF THE MODEL

SolidWorks allows you to dynamically rotate the view in the 3D space so that the solid models in the current file can be viewed from all directions. This allows you to visually maneuver around the model so that all the features in the model can be clearly viewed. This tool can be invoked even when you are inside some other tool. For example, you can invoke this tool when the **Extrude Feature** dialog box is displayed. You can freely rotate the model in the 3D space or rotate it around a selected vertex, edge, or face. Both the methods of rotating the model are discussed next.

Rotating the View Freely in 3D Space

Toolbar:	View > Rotate View
Menu:	View > Modify > Rotate

To rotate the view freely in 3D space, choose the **Rotate View** button from the **View** toolbar. You can also invoke this tool by choosing the **Rotate View** option from the shortcut menu that is displayed when you right-click in the drawing window. When you are inside some other tool use the right mouse button in the drawing area and choose the **View > Rotate View** option from the shortcut menu to invoke the rotate view tool. When you invoke this tool, the cursor will be replaced by the rotate view cursor. Now, press the left mouse button and drag the cursor to rotate the view. Figure 4-31 shows a model being viewed from different directions by rotating the view.

Figure 4-31 *Rotating the view to display the model from different directions*

Rotating the View Around a Selected Vertex, Edge, or Face

To rotate the view around a selected vertex, edge, or face, invoke this tool and move the rotate view cursor close to the vertex, edge, or the face around which you want to rotate the view. When

> **Tip**. *To resume rotating the view freely after you have completed rotating it around a selected vertex, edge, or face, double-click anywhere in the drawing area. Now when you drag the cursor, you will notice that the view is rotated freely in 3D space.*
>
> *If a 3 button mouse is configured to your system, you can press and drag the middle mouse button to rotate the model freely in 3D space. Note that in this case the rotate view cursor will not be displayed.*

it is highlighted, select it using the left mouse button. Next, drag the cursor to rotate the view around the selected vertex, edge, or face.

MODIFYING THE VIEW ORIENTATION

As mentioned earlier, when you invoke the **Extrude Boss/Base** tool or the **Revolve Boss/Base** tool, the view is automatically changed to a 3D view and the preview of the model is displayed. SolidWorks allows you to manually change the view orientation using some predefined standard views or user-defined views. The standard views are available in the **Standard View** toolbar shown in Figure 4-32. The various tools available in this toolbar are discussed next.
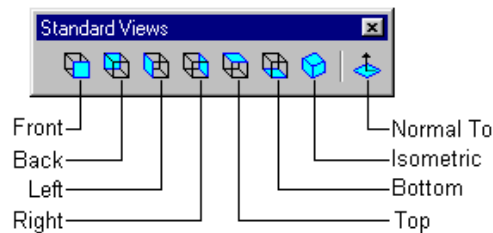

Figure 4-32 Standard View toolbar

Front

The **Front** button is chosen to reorient the view to the front view. This is the default view that is current when you open a new file. The hotkey for the front view is CTRL+1.

Back

The **Back** button is chosen to reorient the view to the back view. The hotkey for the back view is CTRL+2.

Left

The **Left** button is chosen to reorient the view to the left view. The hotkey for the left view is CTRL+3.

Right

The **Right** button is chosen to reorient the view to the right view. The hotkey for the right view is CTRL+4.

Top

The **Top** button is chosen to reorient the view to the top view. The hotkey for the top view is CTRL+5.

Bottom

The **Bottom** button is chosen to reorient the view to the bottom view. The hotkey for the bottom view is CTRL+6.

Isometric

The **Isometric** button is chosen to reorient the view to the isometric view. You can view the model with all three axes in this view. The hotkey for the isometric view is CTRL+7.

Normal To

The **Normal To** button is chosen to reorient the view normal to a selected face or plane. This button will be available only in the sketching environment or when you select a planar face or a plane.

View Orientation

You can also invoke these standard views using the **Orientation** dialog box. This dialog box is invoked by choosing the **View Orientation** button from the **View** toolbar. This dialog box can also be invoked by pressing the SPACEBAR from the keyboard. Note that when you invoke this dialog box by pressing the SPACEBAR, the dialog box will be displayed at the location where the cursor is placed currently. The **Orientation** dialog box is shown in Figure 4-33.
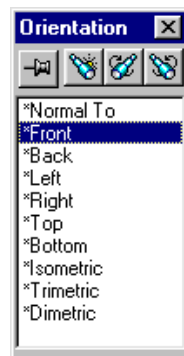
*Figure 4-33 **Orientation** dialog box*

You can invoke the view from this dialog box by double-clicking it. You will notice that in addition to the standard views, two more additional views are displayed. These are the trimetric view and the dimetric view. These options can be used to change the current view to trimetric or dimetric. The buttons that are available on top of this dialog box are discussed next.

Push-Pin

You will notice that the **Orientation** dialog box is automatically closed when you select a view, select a point somewhere on the screen, or invoke a tool. If you want that this dialog box should be retained on the screen, you can pin it at a location by choosing the **Push-Pin** button. This is the first button on top of this dialog box. Move the dialog box to the desired location and then choose this button. The dialog box will be pinned to that location and will not close when you perform any operation.

New View

The **New View** button is chosen to create a user-defined view and save it in the list of the views in the **Orientation** dialog box. Using the various drawing display tools and the **Rotate View** tool, modify the current view and then choose this button. When you choose this button, the **Named View** dialog box will be displayed. Enter the name of the view in the **View Name** edit box and then choose the **OK** button. You will notice that a user-defined view is created and it is saved in the list available in the **Orientation** dialog box.

Update Standard Views

The **Update Standard Views** button is chosen to modify the orientation of the standard views. For example, if you want that the view that is displayed when you invoke the **Back** option from this dialog box should be the front view, then change the current view to the back view by double-clicking it in the **Orientation** dialog box. Now, select the **Front** option from the list of the views available in the **Orientation** dialog box and then choose the **Update Standard Views** button. The **SolidWorks** warning box will be displayed and you will be informed that if you change the standard view, all the other named views in the model will also be changed. If you make the change using the **Yes** button then the current view that was originally the back view will become the front view. Also, all the other views will be modified automatically.

Reset Standard Views

The **Reset Standard Views** button is chosen to reset the standard settings of all the standard views in the current drawing. When you choose this button, the **SolidWorks** warning box will be displayed and you will be prompted to confirm whether you want to reset all the standard views to their original settings or not. If you choose **Yes,** all the standard views will be reset to their default settings.

Previous View

The **Previous View** option is used to display the previous orientation of the model. You can undo upto last 10 views. This option is used only when you change the view of the model or a drawing or a sketch one or more than one time.

DISPLAY MODES OF THE MODEL

SolidWorks provides you with various predefined modes to display the model. You can select any of these display modes from the **View** toolbar. These modes are discussed next.

Shaded

The **Shaded** mode is the default mode in which the model is displayed. When you open a new file and create a base feature, you will notice that it is automatically shaded. This is because the **Shaded** button is chosen by default in the **View** toolbar.

Fast HLR/HLV

Sometimes when you rotate the view of a large assembly, or a model with large number of features with **Shaded** or the **Hidden Lines Removed** shading modes, the regeneration of the model takes a lot of time. This can be avoided by choosing the **Fast HLR/HLG** button in combination with the other shading modes. Choosing this button speeds up the regeneration time and you can easily rotate the view. This is a toggle mode and is turned on when you choose this button. This button is chosen in combination with any of the other display modes.

Hidden Lines Removed

When you choose the **Hidden Lines Removed** button, the hidden lines in the model will not be displayed. Only those edges will be displayed that should be displayed in the current view.

Hidden Lines Visible

When you choose the **Hidden Lines Visible** button, the model is displayed in the wireframe and the hidden lines in the model will be displayed as dashed lines.

Wireframe

When you choose the **Wireframe** button, all the hidden lines will be displayed along with the visible lines in the model. If you set this display mode for complex models, sometimes it becomes difficult to recognize the visible lines and the hidden lines.

Perspective

You can display the perspective view of a model using the **Perspective** button from the **View** toolbar. You can create the perspective view of any type of view such as Shaded, Wireframe, Hidden In Gray, or Hidden Lines Removed. You can also save the perspective view as a named view. Choose **View > Modify > Perspective** from the menu bar to invoke the **Perspective View PropertyManager**. Using this **PropertyManager** you can modify the observer position using the **Object Sizes Away** spinner. Figure 4-34 shows a perspective view.

> **Tip**. *When you rotate the view with the current display mode set to **Hidden Lines Removed**, the hidden lines in the model are automatically displayed while the view is being rotated. If you do not want to display the hidden lines, choose the **Fast HLR/HLV** button in combination with the **Hidden Lines Removed** button in the **View** toolbar and then rotate the view. You will notice that the hidden lines are no more displayed in the model.*

Figure 4-34 *Perspective view of a model*

Display HLR Edges In Shaded Mode

This button is not available in the **View** toolbar by default. You will have to add this button to the **View** toolbar using the **Customize** dialog box. You can also invoke this display mode by choosing **View > Display > HLR Edges In Shaded Mode** from the menu bar. If this button is chosen, the model will be displayed in shaded mode and the visible edges of the model are also highlighted.

Shadows In Shaded Mode

The **Shadows In Shaded Mode** button is used to display the shadow of the model. A light appears from the top of the model to display the shadow in the current view. When you activate the shadow in the shaded mode then the performance of the system is affected during the dynamic orientation. The position of the shadow is not changed when you rotate the model in the 3D space. To change the placement of shadow first remove the shadow in the shaded model using the **Shadow in Shaded Mode** button and rotate the model; after rotating the model use the same button to activate the shadow in the shaded mode. Figure 4-35 shows a model with shadow in the shaded mode.
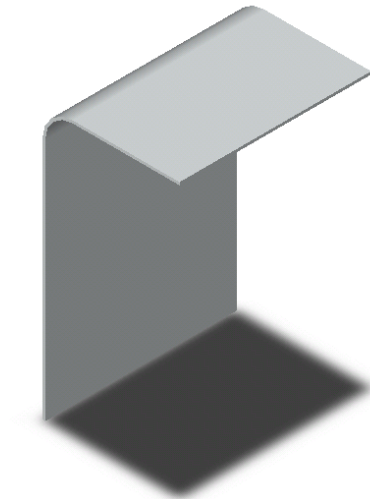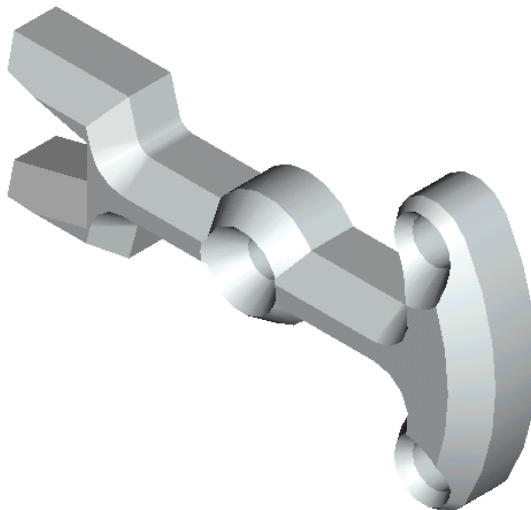
Figure 4-35 *Shadow in active shaded mode*

TUTORIALS

Tutorial 1

In this tutorial you will open the sketch drawn in Tutorial 2 of Chapter- 3. You will then convert this sketch into an extruded model by extruding it in two directions as shown in Figure 4-36. The parameters for extruding the sketch are given next.

Figure 4-36 *Model for Tutorial 1*

Direction 1
Depth = 10mm
Draft angle = 35°

Direction 2
Depth = 15mm
Draft angle = 0°

After creating the model, you will rotate the view using the **Rotate View** tool and then modify the standard views such that the front view of the model becomes the top view. You will then save the model with the current settings. **(Expected time: 30 min)**

The steps that will be followed to complete this tutorial are given next:

a. Open the file of Tutorial 2 of Chapter- 3, refer to Figure 4-37.
b. Save this file in the *c04* directory with a new name.
c. Invoke the **Extrude Boss/Base** tool and convert the sketch into a model, refer to Figures 4-38 and 4-39.
d. Rotate the view using the **Rotate View** tool to view the model from all the directions, refer to Figure 4-40.
e. Invoke the **Orientation** dialog box and then modify the standard view, refer to Figure 4-41.

Opening the File of Tutorial 2 of Chapter- 3

Since the file that you require is saved in the *\My Documents\SolidWorks\c03* directory, you will have to select this directory and then open the *c03-tut2.SLDPRT* file.

1. Start SolidWorks by double-clicking its shortcut icon at the desktop of your computer. Close the **Tip of the Day** dialog box.

2. Choose the **Open Document** option from the **Welcome to SolidWorks 2003** window.

3. Select the *\My Documents\SolidWorks\c03* directory.

   All the files that were created in Chapter 3 will be displayed in this directory.

4. Select the *c03-tut02.SLDPRT* file and then choose the **Open** button.

   Since the sketch was saved in the sketching environment in Chapter 3, it will open in the sketching environment.

Saving the File in the c04 Directory

It is recommended that when you open a file of some other chapter, you should save it in the directory of the current chapter with some other name before you proceed with modifying the file. This is because if you save the file in the current directory, the original file of the other chapter will not be modified.

1. Choose **File > Save As** from the menu bar to display the **Save As** dialog box.

Since the *c03* directory was selected last to open the file, it will be the current directory.

2. Choose the **Up One Level** button available on the right of the **Save in** drop-down list to move to the *SolidWorks* directory.

3. Create a new directory with the name *c04* using the **Create New Folder** button.

4. Make the *c04* directory current by double-clicking it.

5. Enter the new name of the drawing as *c04-tut01.SLDPRT* in the **File name** edit box and then choose the **Save** button to save the document.

    The file will be saved with the new name and the new file will now be opened on the screen. The sketch that will be displayed on the screen is shown in Figure 4-37.
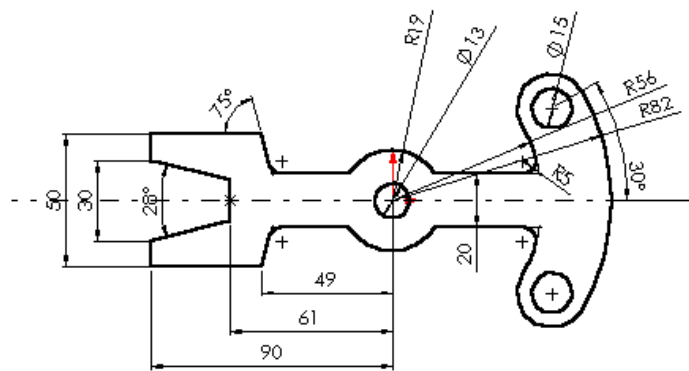
Figure 4-37 Sketch that will be opened on the screen

Extruding the Sketch

Next, you will invoke the **Extrude Boss/Base** tool and extrude the sketch using the parameters given in the tutorial description.

1. Choose the **Extrude Boss/Base** button from the **Features** toolbar to invoke the **Extrude PropertyManager** as shown in Figure 4-38.

    Since the sketch is closed, therefore, only the **Direction 1** rollout will be displayed in the **Extrude PropertyManager**. You will notice that the view is automatically changed to a 3D view. The preview of the feature, in the temporary shaded graphics with the default values, is shown in the drawing area.

2. Choose the **Draft On/Off** button from the **Direction 1** rollout and then set the value of the **Draft Angle** spinner to **35**.
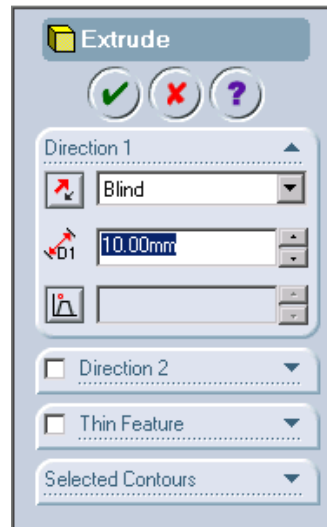
Figure 4-38 Extrude Feature dialog box

These are the settings in direction 1. Now, you need to specify the settings for direction 2.

3. Select the **Direction 2** check box to invoke the **Direction 2** rollout.

   You will notice that the default values in this rollout are the same as you specified in the **Direction 1** rollout.

   Since the **Draft On/Off** button is selected when you invoke the **Direction 2** rollout, therefore, you need to turn this button off. This is because you do not require the draft angle in the second direction.

4. Choose the **Draft On/Off** button from the **Direction 2** rollout. Set the value of the **Depth** spinner to **15** since the depth in the second direction is 15mm.

   This completes all the settings for the model in both the directions.

5. Choose the **OK** button to create the feature or choose the **OK** icon from the confirmation corner.

   It is recommended that you change the view to isometric view after creating the feature so that you can properly view the feature.

6. Choose the **Isometric** button from the **Standard Views** toolbar. The isometric view of the resultant solid model is shown in Figure 4-39.

Rotating the View

As mentioned earlier, you can rotate the view so that you can view the model from all the directions.
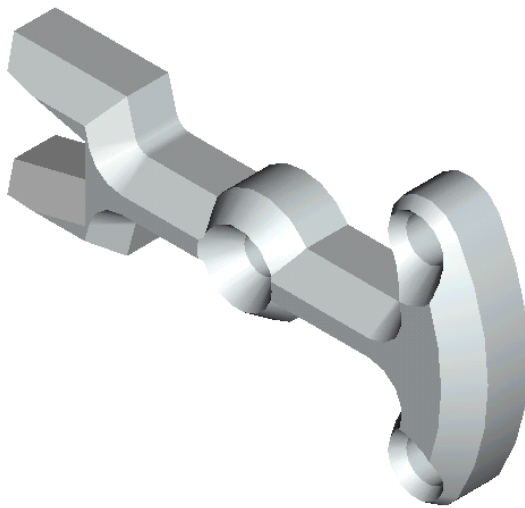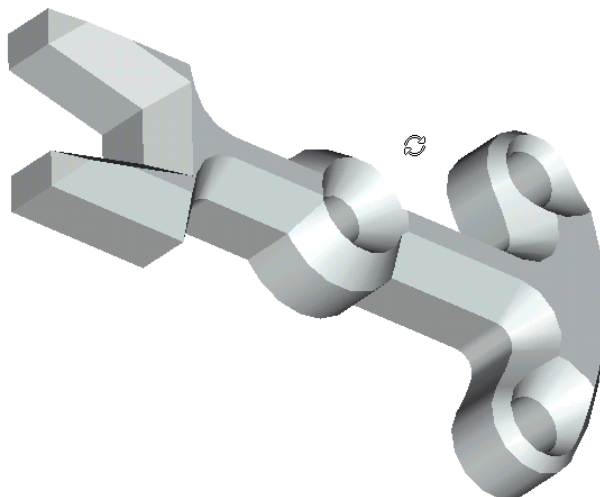
Figure 4-39 *Isometric view of the solid model*

1. Choose the **Rotate View** button from the **View** toolbar.

   The arrow cursor will be replaced by the rotate view cursor.

2. Press and hold down the left mouse button and drag the cursor on the screen to rotate the view as shown in Figure 4-40.

Figure 4-40 *Rotating the view to display the model from different directions*

   You will notice that the model is being displayed from different directions. Note that when you rotate the view, the model is not being rotated. The camera that is used to view the model is being rotated around the model.

3. After viewing the model from all the directions, choose the **Isometric** button again from the **Standard Views** toolbar to change the current view to the isometric view.

Modifying the Standard Views

As mentioned in the tutorial description, you need to modify the standard views such that the front view of the model becomes the top view. This is done using the **Orientation** dialog box.

1. Press the SPACEBAR on the keyboard to invoke the **Orientation** dialog box.

   The orientation dialog box is automatically closed as soon as you perform any other operation. Therefore, you will have to pin this dialog box so that it is not closed automatically.

2. Hold the **Orientation** dialog box by selecting it on the blue bar at top of this dialog box and then drag it to the top right corner of the drawing window.

3. Choose the **Push Pin** button to pin this dialog box at the top right corner of the drawing window. Pinning the dialog box ensures that the dialog box is not automatically closed when you perform any other operation.

4. Double-click the **Front** option in the list box of the **Orientation** dialog box.

   The current view will be automatically changed to the front view and the model will now be displayed from the front.

5. Select the **Top** option from the list box by selecting it once.

   Make sure you do not double-click this option. This is because if you double-click this option, the model will be displayed from the top.

6. Now, choose the **Update Standard Views** button to update the standard views.

   The **SolidWorks** warning box will be displayed and you will be warned that modifying the standard views will change the orientation of any named view in the drawing.

7. Choose **Yes** from this dialog box to modify the standard views.

8. Now, double-click the **Isometric** option provided in the list box of the **Orientation** dialog box. You will notice that the isometric view is different now, see Figure 4-41.

9. Choose the **Push Pin** button in the **Orientation** dialog box again and pick a point anywhere in the drawing area to close the dialog box.

Saving the Model

Since the name of the document was specified at the beginning, you just have to choose the save button now to save the file.
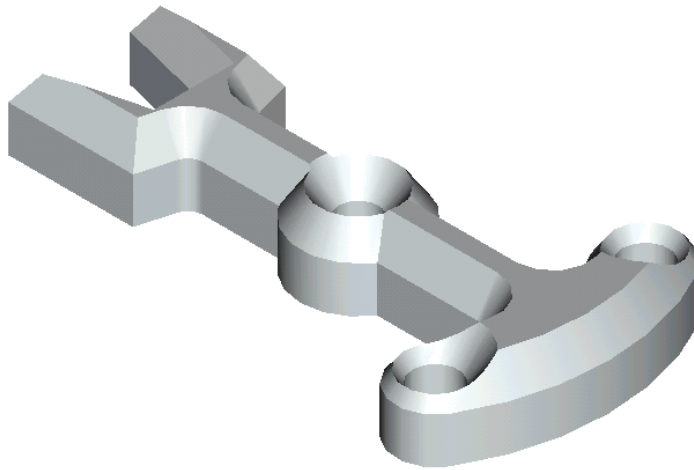
Figure 4-41 Model displayed from modified isometric view

1. Choose the **Save** button from the **Standard** toolbar to save the model.

    The model will be saved with the name *\My Documents\SolidWorks\c04\c04-tut01.SLDPRT*.

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

Tutorial 2

In this tutorial you will open the sketch drawn in Exercise 1 of Chapter- 3. You will then create a thin feature by revolving the sketch through an angle of 270-degree as shown in Figure 4-42. You will offset the sketch outwards while creating the thin feature.
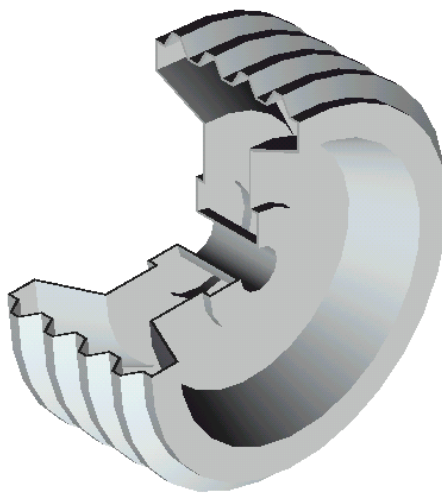
Figure 4-42 Revolved model for Tutorial 2

After creating the model you will set the display type to **Hidden Lines Removed** and will rotate the view to display the model from all the directions. **(Expected time: 30 min)**

The steps that will be followed to complete this tutorial are given next:

a. Open the sketch of Exercise 1 of Chapter- 3, refer to Figure 4-43.
b. Save it in the current directory.
c. Invoke the **Revolve Boss/Base** tool and revolve the sketch through an angle of 270-degree, refer to Figure 4-45.
d. Change the current view to isometric view and then change the display type to **Hidden Lines Removed**, refer to Figure 4-46.
e. Rotate the view using the **Rotate View** tool to display the model from different directions, refer to Figure 4-47.

Opening the file of Exercise 1 of Chapter- 3

Since the file that you require is saved in the *My Documents\SolidWorks\c03* directory, you will have to select this directory and then open the *c03-exr1.SLDPRT* file.

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

   The *c04* directory will be current in this dialog box.

2. Select the *My Documents\SolidWorks\c03* directory.

   All the files that were created in Chapter 3 will be displayed in this directory.

3. Select the *c03-exr01.SLDPRT* file and then choose the **Open** button.

   The file will be opened in the sketching environment. Also, you will notice that the file is maximized in the SolidWorks window. This is because in the previous tutorial you selected the option to maximize the files when opened.

Saving the File in the c04 Directory

As mentioned earlier, it is recommended that you save the file with a new name so that the original file is not modified.

1. Choose **File > Save As** from the menu bar to display the **Save As** dialog box.

   Since the *c03* directory was selected last to open the file, it will be the current directory.

2. Choose the **Up One Level** button available on the right of the **Save in** drop-down list to move to the *SolidWorks* directory.

3. Make the *c04* directory current by double-clicking it.

4. Enter the name of the document in the **File name** edit box as *c04-tut02.SLDPRT*. Choose the **Save** button to save the file. The sketch that will be displayed on the screen is shown in Figure 4-43.
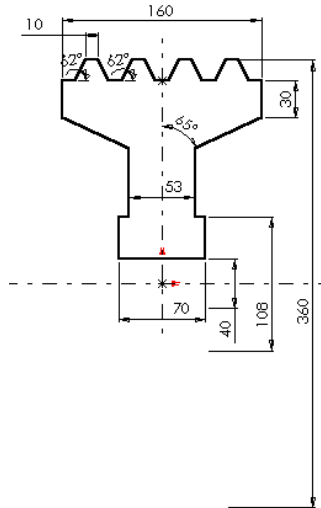
Figure 4-43 Sketch for the revolved model

Revolving the Sketch

The sketch consists of two centerlines. The first centerline was used to mirror the sketched entities and the other one was drawn to revolve the sketch. However, if you choose the **Revolve Boss/Base** button, the **Revolve PropertyManager** will not be displayed. Instead, the **SolidWorks** warning box will be displayed and you will be informed that the sketch should have either a single centerline or you should select a centerline before invoking this tool. Therefore, to revolve the sketches that have more than one centerlines, you need to first select the centerline and then invoke the **Revolve Boss/Base** tool.

1. Select the horizontal centerline and choose the **Revolve Boss/Base** button from the **Features** toolbar.

   The current view will be changed to a 3D view and the **Revolve PropertyManager** will be displayed. The confirmation corner will also be displayed. Since the sketch is closed, therefore, only the **Revolve Parameters** rollout will be displayed in this **PropertyManager**. The preview of a complete revolved feature in temporary shaded graphics will be displayed on the screen. Since the preview of the model is not displayed properly in the current view, therefore, you have to use the zoom to fit option. Also, since you need to create a thin feature, you need to select the **Thin Feature** check box from the **Revolve PropertyManager**.

2. Choose the **Zoom to Fit** button from the **View** toolbar or press the F key from the keyboard.

3. Set the value of the **Angle** spinner to **270**.

The preview of the revolved model will also be modified accordingly. Note that if the horizontal centerline was drawn from left to right, then the direction of revolution has to be reversed.

4. Select the **Thin Feature** check box to invoke the **Thin Feature** rollout as shown in Figure 4-44.

Figure 4-44 Thin Feature rollout

5. Set the value of the **Wall Thickness** spinner to **5**.

You will notice that the preview of the thin feature is shown outside the original sketch.

6. Choose the **OK** button to create the revolved feature or choose the **OK** icon from the confirmation corner.

You will notice that the revolved feature is created. As evident from the model, the thin features are hollowed inside and have some wall thickness.

7. Choose the **Isometric** button from the **Standard Views** toolbar to change the view to isometric view. The revolved feature is shown in Figure 4-45.
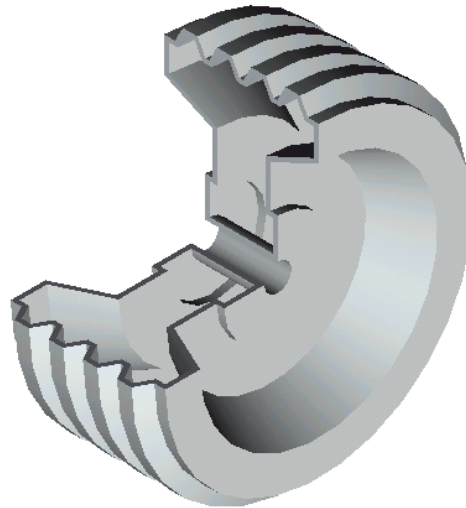
Figure 4-45 Model created by revolving the sketch

Note
*If you are working on Windows XP operating system, the shadow will be automatically displayed.*

Changing the Display Type

As mentioned in the tutorial description, you need to change the display type to **Hidden Lines Removed**. In this type of display mode, the model will be shown with hidden lines removed. You can set this display type by choosing its button available in the **View** toolbar.

1. Choose the **Hidden Lines Removed** button from the **View** toolbar.

   You will notice that the model is no more shaded. However, at the same time the hidden lines in the model will be suppressed and will not be displayed. The model with this display type is shown in Figure 4-46.
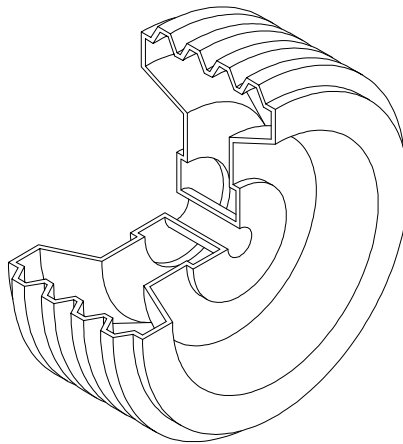
*Figure 4-46 Model displayed in **Hidden Lines Removed** mode*

Rotating the View

Next, you need to rotate the view so that you can view the model from all the directions. As mentioned earlier, the view can be rotated using the **Rotate View** tool.

1. Choose the **Rotate View** button from the **View** toolbar.

   The arrow cursor will be replaced by the rotate view cursor.

2. Press the left mouse button and drag the cursor on the screen to rotate the view. Figure 4-47 shows the model being rotated with hidden lines removed.
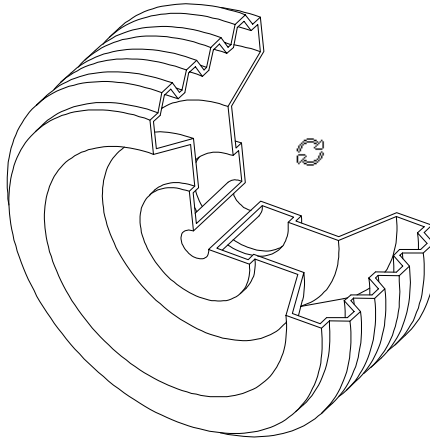
   Note
   *If the hidden lines are displayed while rotating the model, you need to set the hidden line display option. Choose **View** > **Display** > **Use Fast HLR/HLV** from the menu bar.*

3. Choose the **Isometric** button from the **Standard Views** toolbar to change the current view to isometric view.

Figure 4-47 Rotating the model with hidden lines suppressed

Saving the Sketch

Since the name of the document was specified at the beginning, you just have to choose the save button now to save the file.

1. Choose the **Save** button from the **Standard** toolbar to save the model.

    The model will be saved with a name *\My Documents\SolidWorks\c04\c04-tut02.SLDPRT*.

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

Tutorial 3

In this tutorial you will create the model shown in Figure 4-48. The dimensions of the model are shown in Figure 4-49. The extrusion depth of the model is 20mm. After creating the model, rotate the view and then change the view back to isometric view before saving the model.

(Expected time: 45 min)

The steps that will be used to complete this tutorial are given next:

a. Open a new part file and then switch to the sketching environment.
b. Draw the outer loop of the sketch and then draw the sketch of three inner cavities. Finally, draw the six circles inside the outer loop, refer to Figures 4-50 through 4-54.
c. Invoke the **Extrude Boss/Base** tool and extrude the sketch through a distance of 20mm, refer to Figure 4-55.
d. Rotate the view using the **Rotate View** tool.
e. Change the current view to isometric view and then save the model.
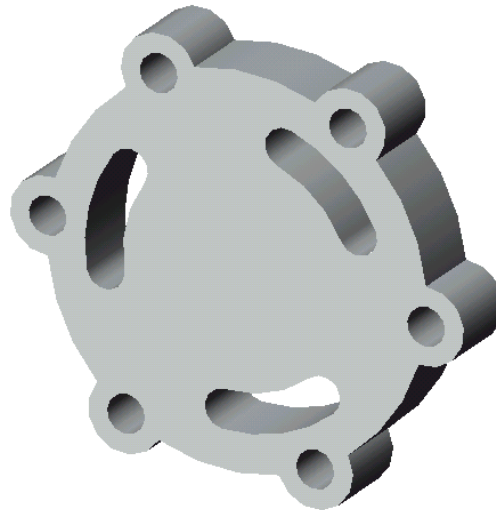
Figure 4-48 *Model for Tutorial 3*
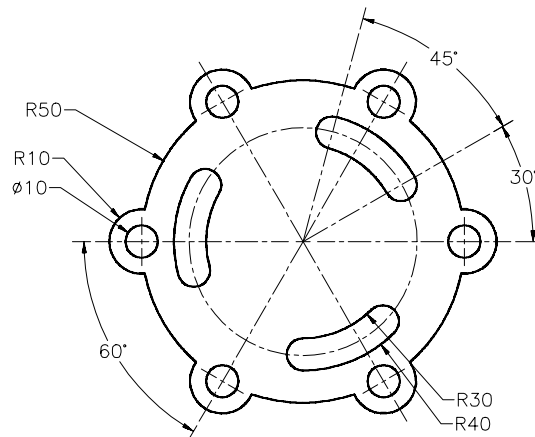
Figure 4-49 *Dimensions of the model for Tutorial 3*

Opening a New Part File

1. Choose the **New** button from the **Standard** toolbar and open a new part file using the **New SolidWorks Document** dialog box.

2. Choose the **Sketch** button from the **Sketch** toolbar to switch to the sketching environment for drawing the sketch.

Drawing the Outer Loop

When the sketch consists of more than one closed loop, it is recommended that you add relations and dimensions to the outer loop first so that it is fully defined. Next, draw the

inner loops one by one and add relations and dimensions to them. Therefore, you will first draw the outer loop first and then add the relations and dimensions to it.

1. Draw a circle in the first quadrant and then dimension it so that it is forced to a diameter of 100mm.

2. Locate the center of the circle at a distance of 70mm along X and Y directions from the origin by adding dimensions in both the directions. Choose the **Zoom to Fit** button to fit the display on the screen.

3. Draw a horizontal centerline from the center of the circle.

4. Draw a circle at the intersection of the centerline and the bigger circle.

5. Trim the part of the sketch so that the sketch looks similar to the one shown in Figure 4-50.
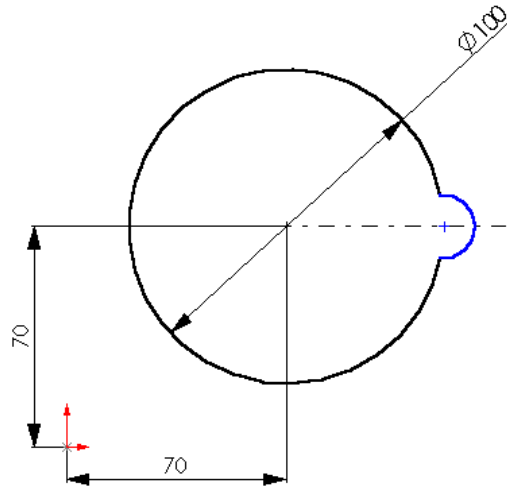
Figure 4-50 Sketch after trimming the unwanted portion

6. Dimension the smaller arc so that it is forced to a radius of 10mm.

7. Add the **Coincident** relation to the centerpoint of the smaller arc and the circumference of the outer arc.

    You will notice that as you add the dimension and relations to the sketch, it turns black in color. This suggests that the sketch is fully defined.

    Next, you will create a circular pattern of the smaller arc. The total number of instances in the pattern is 6 and the total angle is 360-degree.

8. Select the smaller arc using the **Select** tool and then choose the **Circular Sketch Step and Repeat** button from the **Sketch Tools** toolbar.

    The **Circular Sketch Step and Repeat** dialog box will be displayed and the cursor will be replaced by the circular pattern cursor.

9. Move the circular pattern cursor at the control point available at the end of the arrow that is displayed at the origin.

    The circular pattern cursor will turn yellow in color.

10. Press and hold the left mouse button down at the control point and then drag it to the center of the outer arc in the sketch. Release the left mouse button when the cursor turns yellow in color.

11. Set the value of the **Number** spinner in the **Step** area to **6**. Accept all the other default values and choose the **OK** button to create the pattern.

    You will notice that all the instances of the pattern are black in color. This is because you have already applied the dimensions and relations to the original instance and so the other instances are also fully defined.

12. Trim the unwanted portion of the outer arc using the **Sketch Trim** tool. This completes the outer loop. The sketch at this stage should look similar to the one shown in Figure 4-51.
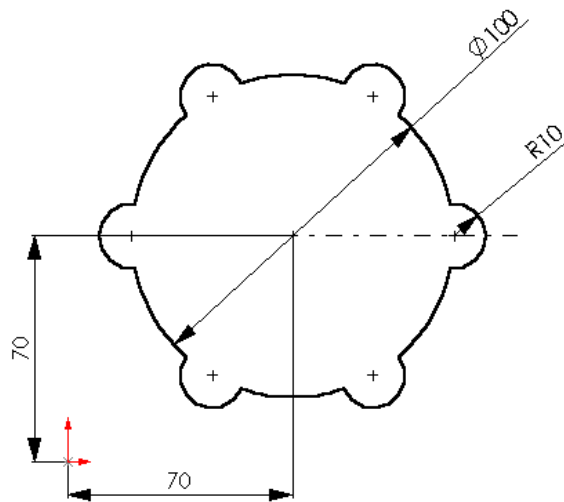
Figure 4-51 *Outer loop of the sketch*
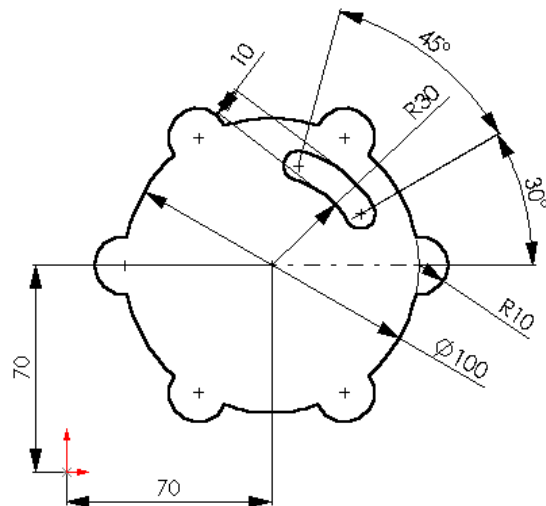
Drawing the Sketch of the Inner Slots

Now, you need to draw the sketch of the inner cavities. You will draw the sketch of one of the cavities and then add the required relations and dimensions to it. Next, you will create a circular pattern of this cavity. The number of instances in the circular pattern will be 3.

1. Using the **Centerpoint Arc** tool, draw an arc with the center at the centerpoint of the outer arc of 100mm diameter.

2. Dimension this arc such that it is forced to a radius of 30mm. Also, add the angular dimensions to the arc, refer to Figure 4-52. The arc will turn black in color, suggesting that it is fully defined.

3. Offset the last arc outward through a distance of 10mm using the **Offset Entities** tool.

    The new arc created using the **Offset** tool is also black in color. Also, a dimension with the value 10mm will be created between the two arcs.

4. Close the two ends of the arc using the **Tangent Arc** tool. This completes the sketch of one of the inner cavities. All the entities in the sketch at this stage should be displayed in black color as shown in Figure 4-52.

Figure 4-52 *Sketch after drawing the sketch of the inner cavity*

Next, you will create a circular pattern of the inner slot. This is done using the **Circular Sketch Step and Repeat** tool.

5. Select all the entities in the sketch of the inner slot and then choose the **Circular Sketch Step and Repeat** button from the **Sketch Tools** toolbar.

    The **Circular Sketch Step and Repeat** dialog box will be displayed and the center of the circular pattern is again placed at the origin.

6. Hold the left mouse button down at the control point provided at the end of the arrow displayed at the origin and drag it to the center of the outer arc in the sketch.

7.   Set the value of the **Number** spinner in the **Step** area to **3** and then choose the **OK** button
     to create the circular pattern.

     This completes the sketch of the inner cavities. The sketch after creating the circular pattern
     of the inner cavities is shown in Figure 4-53.
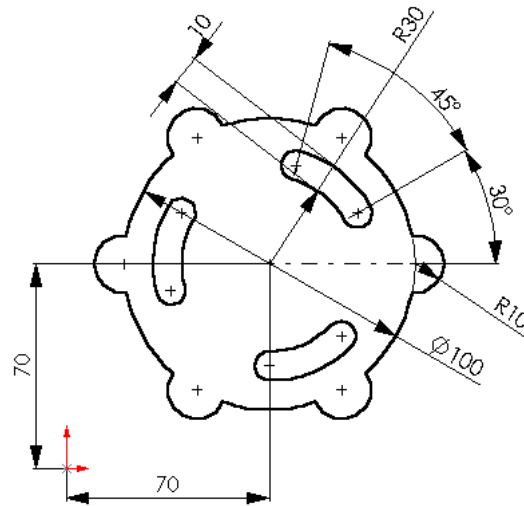
Figure 4-53 *Sketch after creating the circular pattern*

Drawing the Sketch for the Holes

Next, you need to draw the sketch for the holes. You will draw one of the circles and then
add dimension to it. Then you will create a circular pattern of the circle.

1.   Taking the centerpoint of one of the arcs on the outer arc of 100mm diameter, draw a circle
     and then dimension it to force it to a diameter of 10mm.

     The circle will turn black in color when you dimension it.

2.   Select the circle using the **Select** tool and then choose the **Circular Sketch Step**
     and Repeat button from the **Sketch Tools** toolbar.

3.   Drag the center of the circular pattern to the center of the outer arc.

4.   Set the value of the **Number** spinner in the **Step** area to **6**. Choose **OK** to create the pattern.
     All the instances in the pattern will be displayed in black color.

     This completes the sketch of the model. The final sketch of the model is shown in Figure 4-54.

Extruding the Sketch

The next step after drawing the sketch is to extrude it. The sketch will be extruded using the
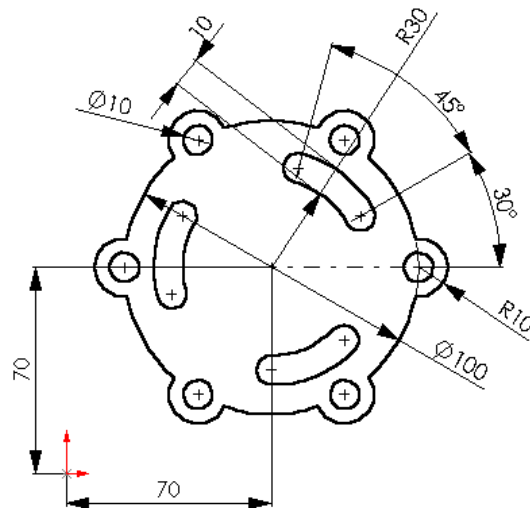**Extrude Boss/Base** tool.

Figure 4-54 *Final sketch of the model*

1. Choose the **Extrude Boss/Base** button from the **Features** toolbar to invoke the **Extrude PropertyManager**.

    The current view will be changed to a 3D view and the **Extrude PropertyManager** will be displayed. Also, the preview of the model as it will be created using the default values will be displayed on the screen.

2. Set the value of the **Depth** spinner to **20** and then choose the **OK** button to extrude the sketch.

3. Choose the **Isometric** button from the **Standard Views** toolbar to change the view to isometric view. The completed model for Tutorial 3 is shown in Figure 4-55.

Rotating the View

1. Choose the **Rotate View** button from the **View** toolbar.

    The arrow cursor will be replaced by the rotate view cursor.

2. Press the left mouse button and drag the cursor on the screen to rotate the view.

3. Choose the **Isometric** button from the **Standard Views** toolbar.

Saving the Sketch

Since the document has not been saved even once until now, therefore, when you choose the **Save** button from the **Standard** toolbar, the **Save As** dialog box will be displayed. You can enter the name of the document in this dialog box.
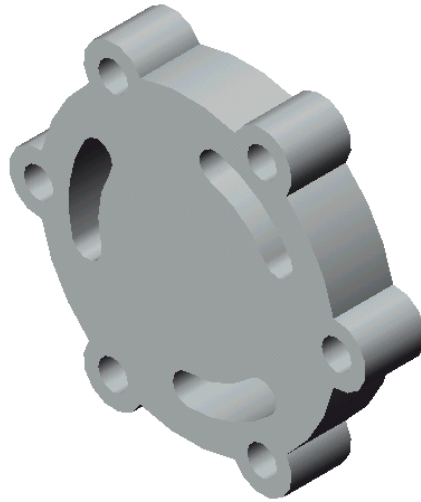
Figure 4-55 *Final model for Tutorial 3*

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below:

   \My Documents\SolidWorks\c04\c04-tut03.SLDPRT.

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 the answers given at the end of this chapter.

1. In SolidWorks, a sketch is revolved using the **Base-Extrude PropertyManager**. (T/F)

2. You can also specify the depth of extrusion dynamically in the preview of the extruded feature. (T/F)

3. You can invoke the drawing display tools such as **Zoom to Fit** while the preview of a model is displayed on the screen. (T/F)

4. When you rotate the view with the current display mode set to Hidden Lines Removed, the hidden lines in the model are automatically displayed while the view is being rotated. (T/F)

5. _____ tool is used to display the perspective view of a model.

6. The **Cap Ends** check box is displayed in the **Extrude Thin Feature** dialog box only when the sketch selected to create a thin feature is _____.

7. The _____ check box is used to create a feature with different values in both the directions of the sketching plane.

8. The _____ check box is used to apply the automatic fillets while creating a thin feature.

9. The _____ button is used to display the shadow in the shaded mode.

10. To resume rotating the view freely after you have completed rotating it around a selected vertex, edge, or face, _____ any where in the drawing area.

REVIEW QUESTIONS
Answer the following questions:

1. You can also invoke the **Rotate View** tool by choosing the **Rotate View** option from the _____ that is displayed when you right-click in the drawing window.

2. When you choose the **Wireframe** button, all the _____ lines will be displayed along with the visible lines in the model.

3. You can also modify the parallel view to perspective view by choosing _____ from the menu bar.

4. When you invoke the **Extrude Boss/Base** tool or the **Revolve Boss/Base** tool, the view is automatically changed to a _____.

5. The thin revolved features can be created using a _____ or an _____ sketch.

6. Which one of the following buttons is chosen to modify the orientation of the standard views?

    (a) **Update Standard Views**   (b) **Reset Standard Views**
    (c) None   (d) Both

7. Which one of the following buttons is not available in the **View** toolbar by default?

    (a) **Hidden Lines Removed**   (b) **Hidden In Gray**
    (c) **Shaded**   (d) **Display HLR Edges In Shaded Mode**

8. Which one of the following parameters will not be displayed in the preview of the model?

    (a) Depth   (b) Draft angle
    (c) None   (d) Both

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9.   If the sketch is open, it can be converted into

    (a) Thin feature   (b) Solid feature
    (c) None   (d) Both

10. In SolidWorks, the circular pattern of the sketched entities is created using which one of the following tools?

    (a) **Circular Pattern**   (b) **Circular Sketch Step and Repeat**
    (c) None   (d) Both

EXERCISES

Exercise 1

Create the model shown in Figure 4-56. The sketch of the model is shown in Figure 4-57. Create the sketch and dimension the sketch using the autodimension option. The extrusion depth of the model is 15mm. After creating the model, rotate the view.   **(Expected time: 30 min)**
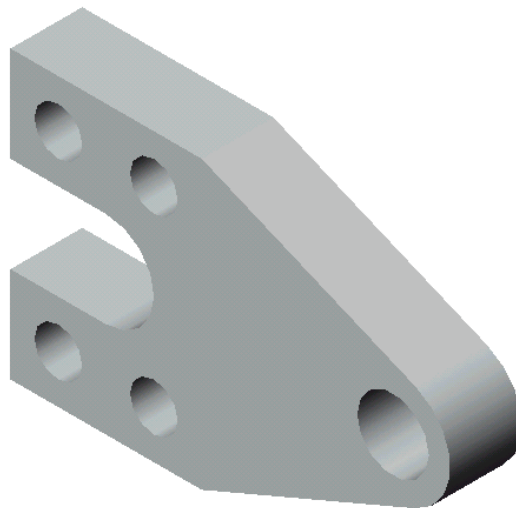
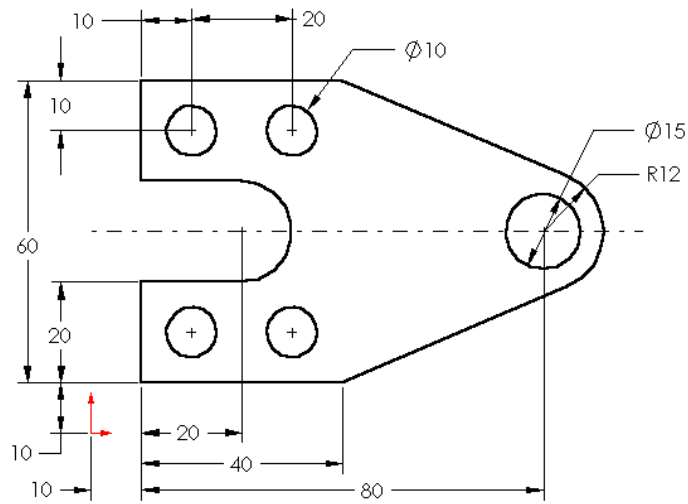***Figure 4-56*** *Model for Exercise 1*

Figure 4-57 *Sketch of the model for Exercise 1*

Exercise 2

Create the model shown in Figure 4-58. The sketch of the model is shown in Figure 4-59. Create the sketch and dimension the sketch using the autodimension tool. The extrusion depth of the model is 25mm. Modify the standard view such that the current front view of the model should be displayed when you invoke the top view. **(Expected time: 30 min)**
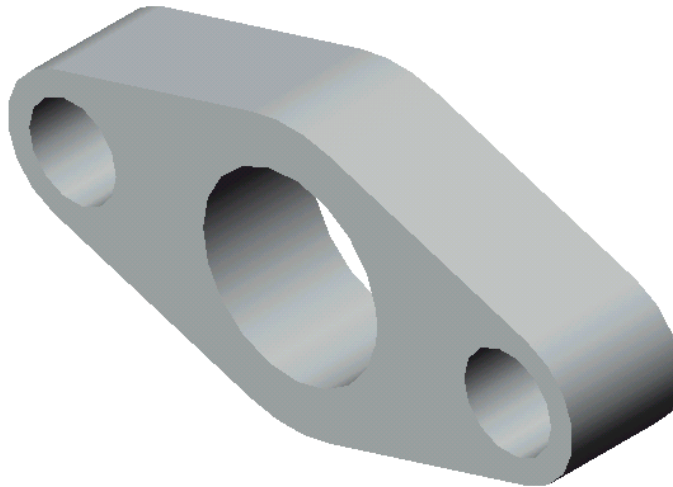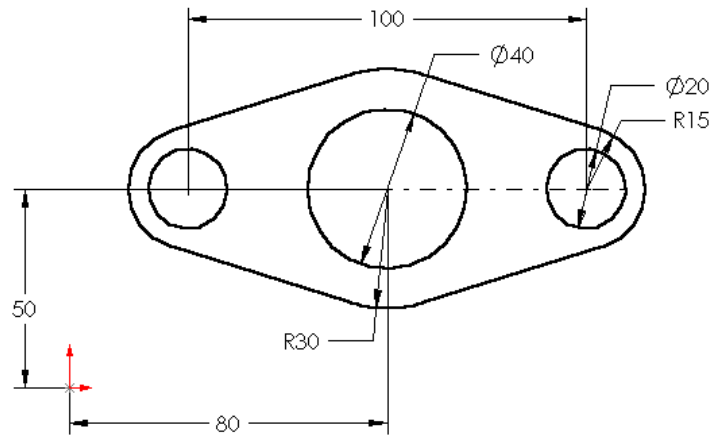

Figure 4-58 *Model for Exercise 2*

Figure 4-59 *Sketch of the model for Exercise 2*

Answers to Self-Evaluation Test
1. F, **2.** T, **3.** T, **4.** T, **5. Perspective**, **6.** closed, **7. Both Directions**, **8. Instance deleted**, **9.** shortcut menu, **10.** double-click