



# Chapter 7

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## Advanced Modeling Tools-II

### Learning Objectives

**After completing this chapter, you will be able to:**

- *Mirror features, faces, and bodies.*
- *Create linear patterns.*
- *Create circular patterns.*
- *Create sketch-driven patterns.*
- *Create curve-driven patterns.*
- *Create table-driven patterns.*
- *Create rib features.*
- *Display the section view of the model.*

## ADVANCED MODELING TOOLS

Some of the advanced modeling options were discussed in Chapter 6, Advanced Modeling Tools-I. In this chapter you will learn about more advanced modeling tools, using which you can capture the design intent of the model. The rest of the advanced modeling tools are discussed in later chapters.

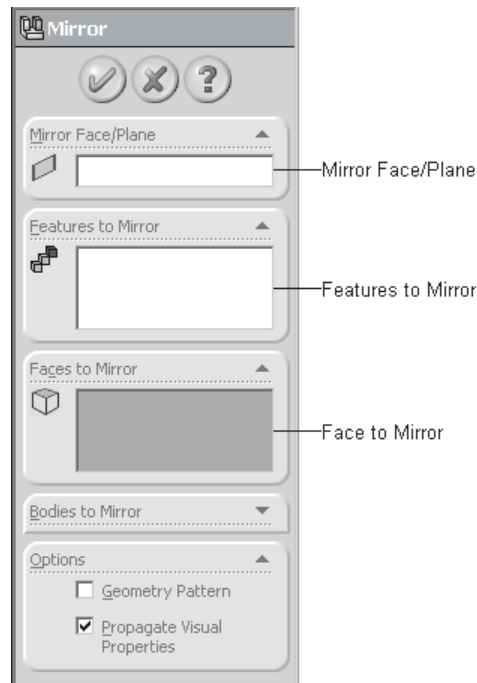
### Creating Mirror Features

**CommandManager:** Features > Mirror  
**Menu:** Insert > Pattern/Mirror > Mirror  
**Toolbar:** Features > Mirror



The **Mirror** tool is used to copy or mirror the selected feature, face, or body about a specified mirror plane, which can be a reference plane or a planar face. To use this tool, choose the **Mirror** button from the **Features CommandManager**, or choose **Insert > Pattern/Mirror > Mirror** to invoke the **Mirror**

**PropertyManager**, as displayed in Figure 7-1. The confirmation corner is also displayed in the drawing area.



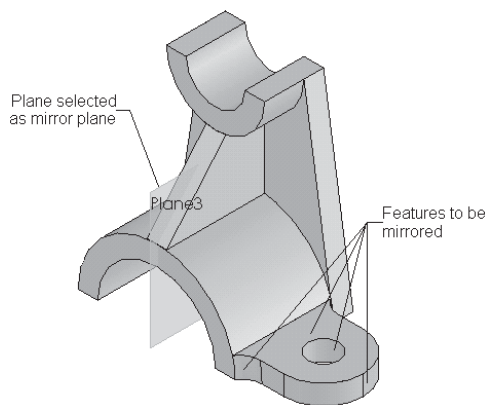
*Figure 7-1 The Mirror PropertyManager*

The options that are used to mirror features, faces, and bodies are discussed next.

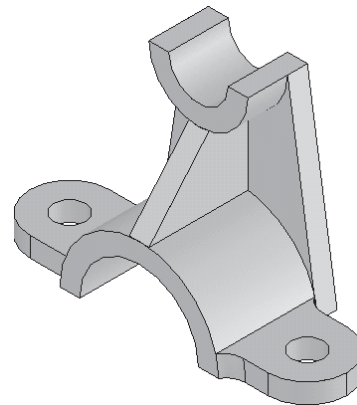
### Mirroring Features

You can mirror the selected feature along the specified mirror plane or face by using this

feature. To do so, invoke the **Mirror PropertyManager**. You are prompted to select a plane or a planar face about which to mirror, followed by the features to be mirrored. Select a plane or a planar face that will act as mirror plane or mirror face. After selecting the mirror plane or face, the selection mode of the **Features to Mirror** selection box is invoked and you are prompted to select features to mirror. Select the feature or features from the drawing area or from the **FeatureManager Design Tree** which is displayed in the drawing area. When you select the features to be mirrored, the preview of the mirrored image is displayed in the drawing area. After selecting all the required features, choose the **OK** button from the **Mirror PropertyManager**. Figure 7-2 shows the mirror plane and features to be mirrored and Figure 7-3 shows the resulting mirrored features.



**Figure 7-2** Mirror plane and the features to be mirrored



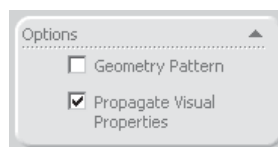
**Figure 7-3** The resulting mirror feature



**Tip.** You can also preselect the mirror plane or mirror face and the features to be mirrored before invoking the **Mirror PropertyManager**.

### Mirroring with and without Geometric Pattern

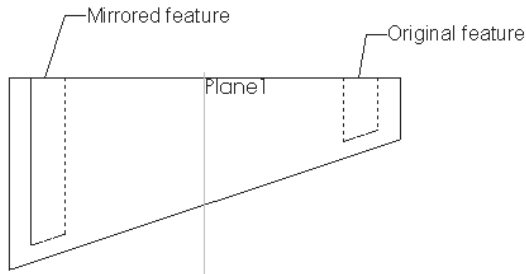
When you create a mirror feature, you are provided with an option known as the geometric pattern. This option is available in the **Options** rollout shown in Figure 7-4.



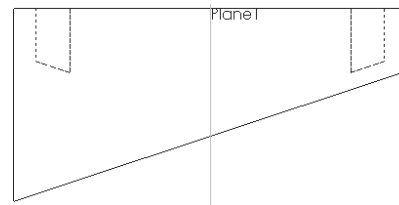
**Figure 7-4** The **Options** rollout

By default, the **Geometry Pattern** check box is cleared. Therefore, if you mirror a feature that is related to some other entity, the same relationship will be applied to the mirrored feature. Consider a case in which an extruded cut is created using the **Offset From Surface** option. If you mirror the cut feature along a plane, the same relationship will be applied to the mirrored

cut feature. The mirrored cut feature will be created with the same end condition of feature termination. Figure 7-5 shows a hole feature created on the right and mirrored along **Plane 1**, with the **Geometry Pattern** check box cleared.



**Figure 7-5** Mirror feature created with the **Geometry Pattern** check box cleared

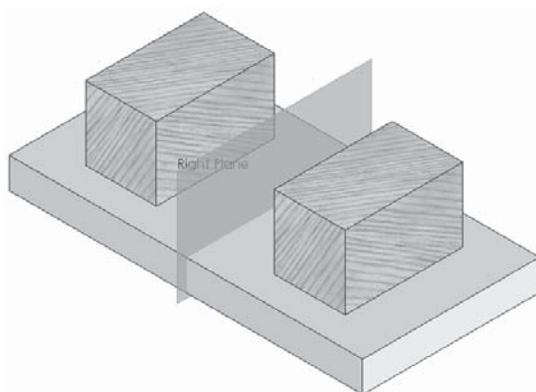


**Figure 7-6** Mirror feature created with the **Geometry Pattern** check box selected

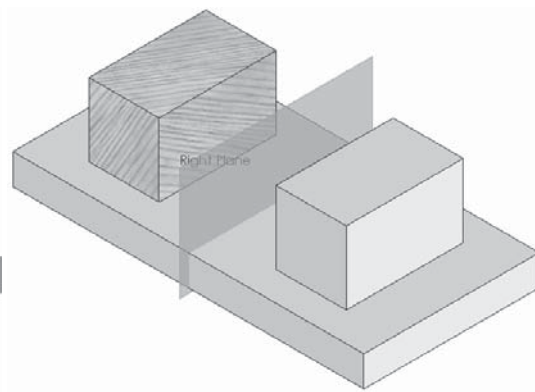
If you select the **Geometry Pattern** check box, the mirror feature created will not depend on the relational references. It will just create a replica of the selected geometry. Figure 7-6 shows the mirror feature created with the **Geometry Pattern** check box selected.

### Propagating Visual Properties While Mirroring

With this release of SolidWorks, the **Propagate Visual Properties** check box is provided in the **Options** rollout. This check box is selected by default and is used to transfer the visual properties assigned to the feature or parent body to the mirrored instance. Note that the visual properties will be visible only after you exit this tool. These visual properties include colors and textures applied to the features or part bodies. If you clear this check box, then the color or texture applied on faces, features, or bodies will be reflected in the resulting mirrored instance. Figure 7-7 shows the mirrored feature with the **Propagating Visual Properties** check box selected and Figure 7-8 shows the mirrored features with this check box cleared.



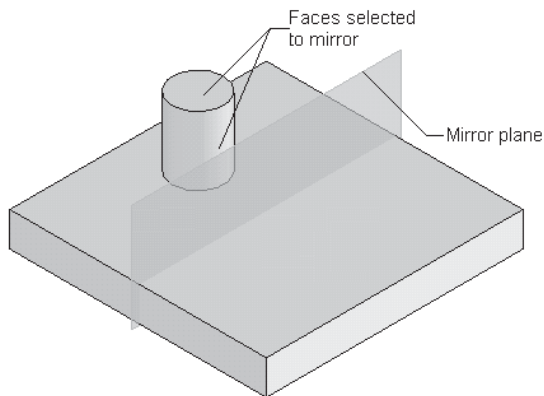
**Figure 7-7** Mirror feature with the **Propagating Visual Properties** check box selected



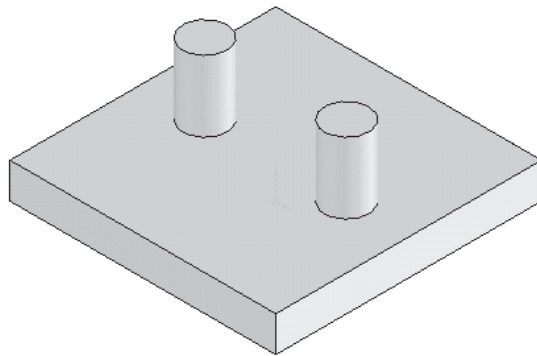
**Figure 7-8** Mirror feature with the **Propagating Visual Properties** check box cleared

## Mirroring Faces

Using this option, you can mirror faces along a mirror plane or mirror face. To use this option, invoke the **Mirror PropertyManager**. You are prompted to select a plane or a planar face about which to mirror. Select the planar face or a plane about which the selected faces will be mirrored. Now, click once in the **Faces to Mirror** selection box to invoke the selection mode, and select the faces to be mirrored. The selected faces must form a closed body. Else, the feature creation is not possible. Use the **OK** button from the **Mirror PropertyManager** to end the feature creation. Figure 7-9 shows the faces and the mirror plane to be selected. Figure 7-10 shows the resulting mirror feature created.



**Figure 7-9** Mirror plane and faces selected to mirror



**Figure 7-10** The resulting mirror created



### Note

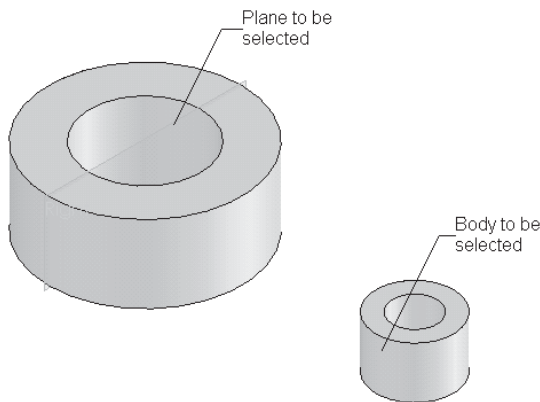
The following are some of the factors that should be considered, while creating a mirror feature by mirroring the faces along the selected plane or planar face:

1. If the replica of the faces is not coincident to the parent part body, SolidWorks will give an error while creating the mirror feature.
2. If the replica of the faces exists on faces other than the original face, SolidWorks will give an error while creating the mirror feature.
3. If the selected faces form a complex geometry, SolidWorks will give an error while creating the mirror feature.
4. If the mirrored faces exist on more than one face, SolidWorks will give an error while creating the mirror feature.
5. The selected faces should form a closed body. If the selected faces do not form a closed body, SolidWorks will give an error while creating the mirror feature.

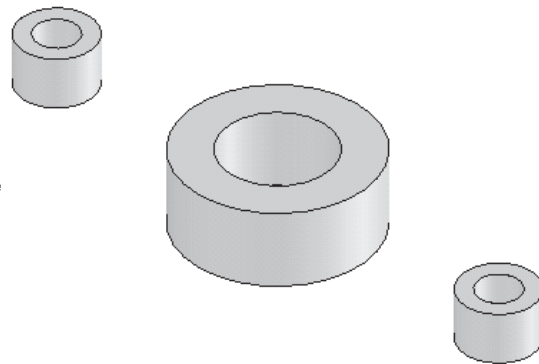
## Mirroring Bodies

As discussed in the earlier chapters, the SolidWorks supports the multibodies environment. Therefore, using the **Mirror** tool, you can also mirror the disjoint bodies. To mirror a body

along a plane, invoke the **Mirror PropertyManager** and select a plane or a planar face that will act as a mirror plane. Invoke the **Bodies to Mirror** rollout and select the body from the drawing area. Alternatively, you can expand the **FeatureManager Design Tree** flyout and select the body to be mirrored from the **Solid Bodies** folder. The name of the selected body is displayed in the **Solid/Surface Bodies to Mirror** selection box. The preview of the mirrored body is displayed in the drawing area. Choose the **OK** button from the **Mirror PropertyManager**. Figure 7-11 shows the plane and the body selected to be mirrored. Figure 7-12 shows the resulting mirrored feature.

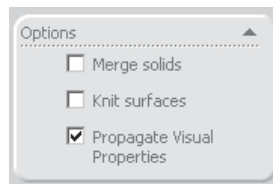


**Figure 7-11** Selecting the mirror plane and body to be mirrored



**Figure 7-12** Resulting mirrored feature

The options available in the **Options** rollout, while mirroring the bodies are discussed next. The **Options** rollout with these options, is displayed in Figure 7-13.



**Figure 7-13** The **Options** rollout

### **Merge solids**

The **Merge solids** option is used to merge the mirrored body with the parent body. Consider a case, in which you mirror a body along a selected plane or a planar face of the same body and the resulting mirrored body is joined to the parent body. In this case, if you select the **Merge solids** check box, the resulting mirrored body will merge with the parent body to become a single body. If the **Merge solids** check box is cleared, the resulting body will be joined with the parent body, but it will not merge with the parent body, therefore, resulting in two separate bodies.

**Knit surfaces**

If you mirror a surface body, then the **Knit Surface** check box is selected to knit the mirrored and the parent body together.



**Tip.** As discussed earlier, the design intent is captured in the model using the mirror option. Therefore, if you modify the parent feature, face, or body the same will be reflected on the mirrored feature, face, or body.

If you want to mirror all the features of the model using the **Features to Mirror** option, you need to select all the features. But for using **Bodies to Mirror** option, you need to select the body from the **Solid Bodies** folder. By selecting the body, all the features are added to the mirror image.

**Creating Linear Pattern Features**

<b>CommandManager:</b>	Features > Linear Pattern
<b>Menu:</b>	Insert > Pattern/Mirror > Linear Pattern
<b>Toolbar:</b>	Features > Linear Pattern



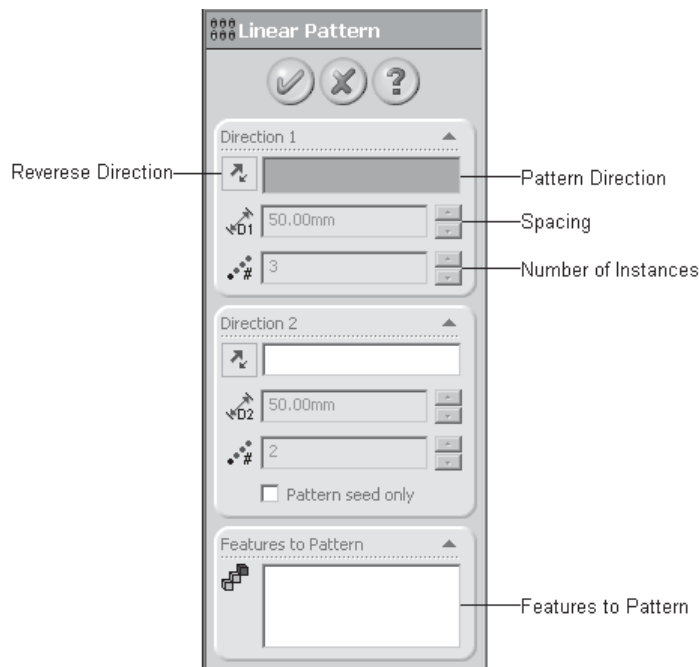
As discussed in the previous chapters, you can arrange the sketched entities in a particular arrangement or pattern. In the same manner, you can also arrange the features, faces, and bodies in a particular pattern. In SolidWorks, you are provided with various types of patterns such as linear patterns, circular patterns, sketch-driven patterns, curve-driven patterns, and table-driven patterns.

In this section, you will learn to create linear patterns. The other type of patterns are discussed later in this chapter.

To create a linear pattern, choose the **Linear Pattern** button from the **Features CommandManager**, or choose **Insert > Pattern/Mirror > Linear Pattern** from the menu bar. The **Linear Pattern PropertyManager** is invoked, and the confirmation corner is also displayed. A partial view of the **Linear Pattern PropertyManager** is displayed in Figure 7-14. The various options available in the **Linear Pattern PropertyManager** are discussed next.

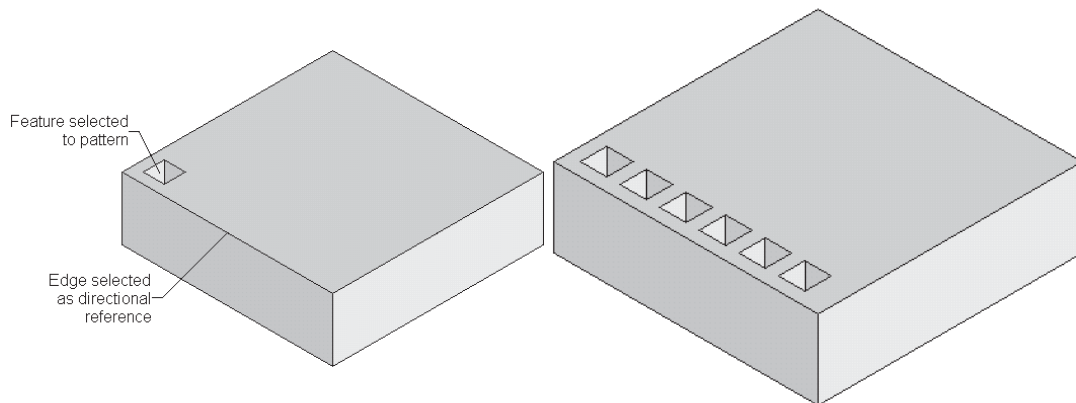
**Linear Pattern in One Direction**

When you invoke the **Linear Pattern PropertyManager**, the **Direction 1** rollout and the **Features** rollout are invoked by default. Also, you are prompted to select an edge or axis for direction reference and face of the feature for pattern features. Therefore, first you need to select an edge or axis as a direction reference. Select an edge or axis as the direction reference. The name of the selected reference will be displayed in the **Pattern Direction** selection box of the **Direction 1** rollout. The selected reference is displayed in green and the **Direction 1** callout is attached to it. The **Direction 1** callout has two edit boxes to define the number of instances and spacing. You are also provided with the **Reverse Direction** arrow along with the selected reference. Now, select a face of the feature to be patterned. The name of the selected feature will be displayed in the **Features to pattern** area of the **Features to Pattern** rollout. The preview of the pattern is displayed in the drawing area with the default values. Set the value of the center to center spacing between the pattern instances in the **Spacing** spinner. Set the value of the number of instances to be patterned in the **Number of**



**Figure 7-14** Partial view of the **Linear Pattern PropertyManager**

**Instances** spinner. You can also set these values in the **Direction 1** callout. Using the **Reverse Direction** button from the **PropertyManager** or the **Reverse Direction** arrow from the drawing area, you can reverse the direction of the pattern feature creation. Figure 7-15 shows the feature and edge to be selected for directional reference and Figure 7-16 shows the model after the pattern creation.



**Figure 7-15** Feature and the edge to be selected

**Figure 7-16** Linear pattern created using the **Direction 1** option

### Linear Pattern in Two Directions

As discussed earlier, you can create a linear pattern of features, faces, and bodies by defining

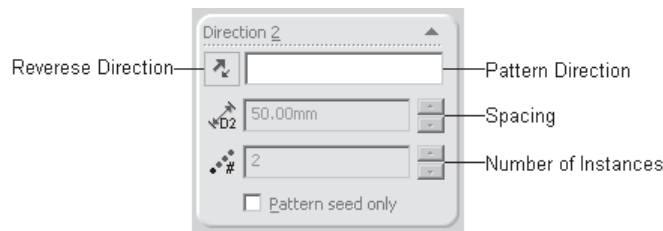




**Tip.** When you select the feature to be patterned, its dimensions are also displayed in the drawing area. You can also select the dimensions as the directional reference.

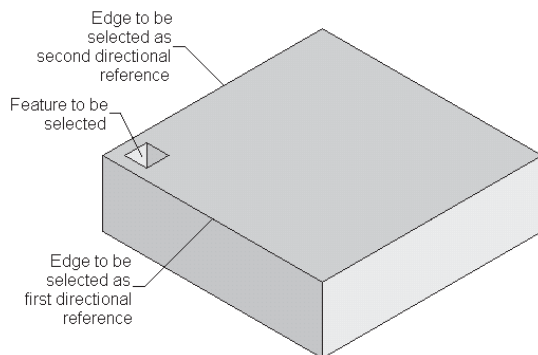
As discussed earlier, you can mirror the faces and bodies. In addition you can also pattern them. It should be noted that the selected faces must form a closed body, otherwise the patterning of faces will give an error.

a single direction using the **Direction 1** rollout. You can also define the parameters in the **Direction 2** rollout to define the pattern in the second direction. The **Direction 2** rollout is shown in Figure 7-17. When you define the pattern in the second direction, the entire row created by specifying the parameters in the first direction is patterned in the second direction.

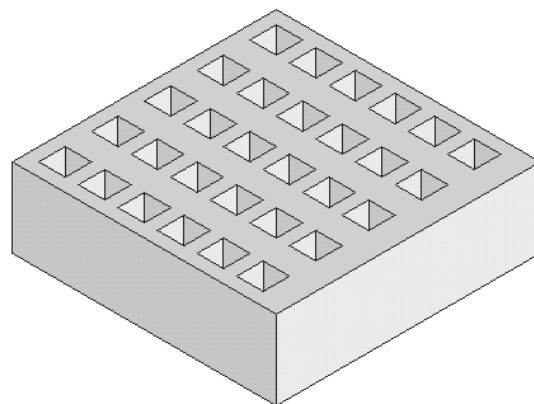


**Figure 7-17** The **Direction 2** rollout

To create a pattern by specifying the parameters in both the directions, select the feature to be patterned, and invoke the **Linear Pattern PropertyManager**. Select the first directional reference. Next, specify the parameters in the **Direction 1** rollout. Now, select the second directional reference. If the **Direction 2** rollout is not invoked by default in the **Linear Pattern PropertyManager**, use the blue arrow in the **Direction 2** callout to invoke it. The options available in the **Direction 2** rollout are the same as those discussed in the **Direction 1** rollout. Figure 7-18 shows the directional references and the feature to be selected. Figure 7-19 shows the linear pattern created using the **Direction 1** and **Direction 2** rollouts.

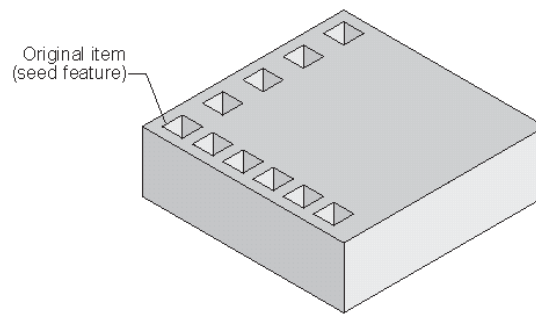


**Figure 7-18** References and feature to be selected



**Figure 7-19** Linear pattern created using the **Direction 1** and **Direction 2** rollouts

By default, all rows of the instances created in the first direction are patterned in the second direction also. This is because the **Pattern seed only** check box in the **Direction 2** rollout is cleared. You can select this check box to pattern only the original selected feature (also called seed feature) in the second direction. Figure 7-20 shows the pattern created with the **Pattern seed only** check box selected.



**Figure 7-20** Linear pattern created with the **Pattern seed only** check box selected

### Instances to Skip

Using the **Instances to Skip** option, you can skip some of the instances from the pattern. These instances are not actually deleted. These only disappear from the pattern feature and you can resume them at any time of your design cycle. To skip the pattern instances, invoke the **Instances to Skip** rollout from the **Linear Pattern PropertyManager**. The **Instances to Skip** rollout is displayed in Figure 7-21.

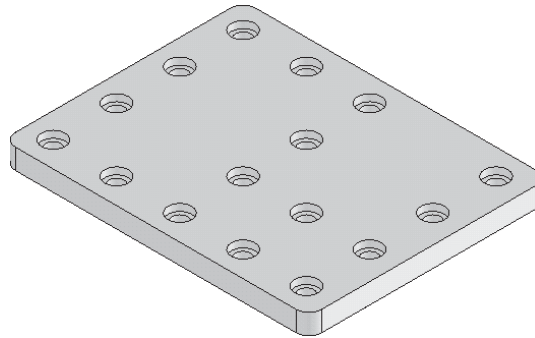


**Figure 7-21** The **Instances to Skip** rollout

As soon as you invoke this rollout, pink dots are displayed at the center of all pattern instances except the parent instance. Therefore, you cannot skip the parent instance. Now, move the cursor to the pink dot of the instance to be skipped. The cursor will be replaced by the instance to skip the cursor and the position of that instance, in the form of a matrix, is displayed in the tooltip below this cursor. Use the left mouse button to skip that instance. The pink dot will be replaced by a red dot and the preview of that instance will disappear from the pattern. The position of the skipped instance is displayed in the **Instances to Skip** selection box of the **Instances to Skip** rollout. Figure 7-22 shows a pattern created with some instances skipped. You can resume the skipped instances by deleting the position of the instance from the **Instances to Skip** selection box or selecting the red dot from the drawing area.

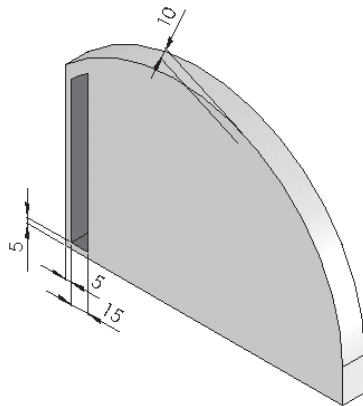
### Pattern Using a Varying Sketch

The **Vary Sketch** option is used in a pattern where the shape and size of each pattern instance

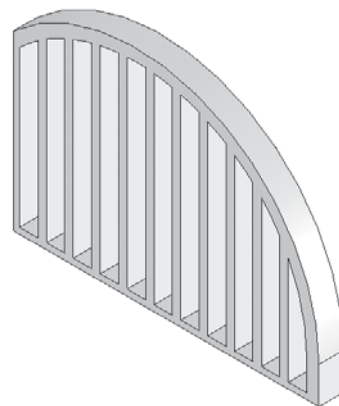


**Figure 7-22** Linear pattern created with some instances skipped

is controlled by the relations and dimensions of the sketch of that feature. In this type of pattern, the dimension of the sketch of the feature to be patterned is selected as the directional reference, which drives the shape and size of the sketch of the feature to be patterned. In Figure 7-23 a cut feature is created on the base feature. Figure 7-24 shows the linear pattern created using the varying sketch option. For creating this type of pattern, the sketch of the feature to be patterned should be in relation with the geometry along which it will vary. The dimensions of the sketch should allow it to change the shape and size easily. You should also provide a linear dimension that will drive the entire sketch and will also be the directional reference. From the **FeatureManager Design Tree** select the feature to be patterned and then invoke the **Linear Pattern PropertyManager**. Now, select the dimension to specify the directional reference and set the value of spacing and the number of instances. Invoke the **Options** rollout and select the **Vary sketch** check box. As you select it, the preview of the pattern disappears from the drawing area. Choose the **OK** button from the **Linear Pattern PropertyManager** to end the feature creation.



**Figure 7-23** Cut feature created on the base feature



**Figure 7-24** Linear pattern created with the **Vary sketch** check box selected

The **Geometry pattern** option available in the **Options** rollout of the **Linear Pattern PropertyManager** is the same as that discussed earlier in the **Mirror PropertyManager**.

### Propagating Visual Properties While Patterning

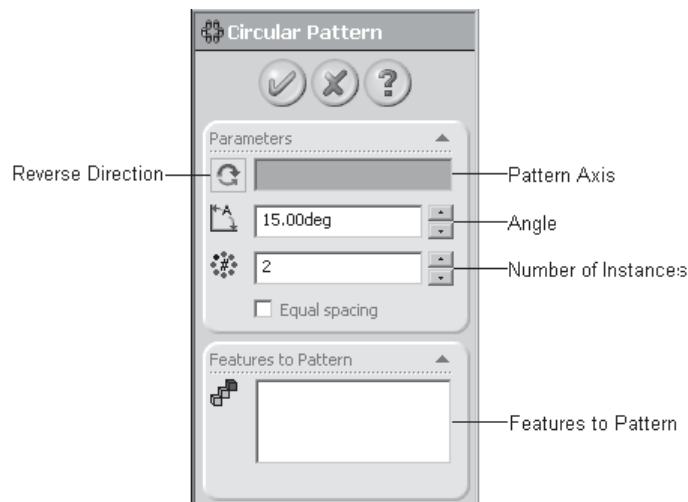
With this release of SolidWorks, the **Propagate Visual Properties** check box is provided in the **Options** rollout. This check box is selected by default and is used to transfer the visual properties assigned to the feature or parent body to the patterned instances. These visual properties includes colors and textures applied to the features or part bodies after you exit this tool. If you clear this check box, then color or texture applied on faces, features, or bodies will be not be reflected in the resulting mirrored instance.

### Creating Circular Pattern Features

<b>CommandManager:</b>	Features > Circular Pattern
<b>Menu:</b>	Insert > Pattern/Mirror > Circular Pattern
<b>Toolbar:</b>	Features > Circular Pattern



As discussed in the previous chapters, you can arrange the sketched entities in a circular pattern using the **Circular Sketch Step and Repeat** option. In this section, you will learn to create the circular pattern of a feature, face, or body by using the **Circular Pattern** tool. To invoke this tool, choose the **Circular Pattern** button from the **Features CommandManager** or choose **Insert > Pattern/Mirror > Circular Pattern** from the menu bar; the **Circular Pattern PropertyManager** is invoked and the confirmation corner is displayed. The partial view of the **Circular Pattern PropertyManager** is as shown in Figure 7-25.

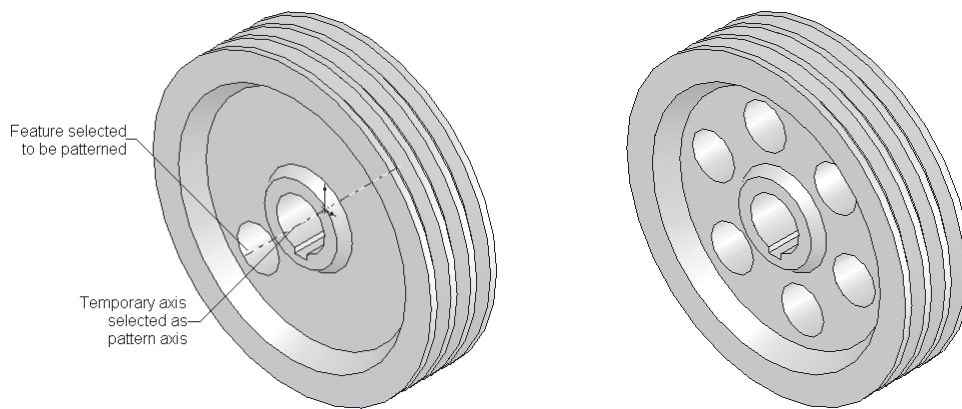


*Figure 7-25 Partial view of the Circular Pattern PropertyManager*

After invoking the **Circular Pattern PropertyManager**, you are prompted to select an edge or axis for the direction reference, and a face of the feature to be patterned. If the base feature of the model is a circular feature, the temporary axis is created automatically at the center of the model. Choose **View > Temporary Axes** from the menu bar to display the

temporary axis. Else, before invoking the **Circular Pattern** tool, you need to create an axis. Now, select the axis that will act as the pattern axis, and the feature to be patterned; the preview of the pattern feature with the default values is displayed in the drawing area. The **Direction 1** callout is also displayed with the **Reverse Direction** arrow in the drawing area. By default, the **Equal Spacing** check box is cleared. Therefore, you need to set the value of the incremental angle between the instances in the **Total Angle** spinner. Set the value of number of instances to pattern in the **Number of Instances** spinner. If you select the **Equal Spacing** check box, you need to enter the value of the total angle, along which all the instances of the pattern will be placed. The angular spacing between the instances will be automatically calculated.

The **Reverse Direction** button available on the left of the **Pattern Axis** selection box is used to change the direction of rotation. By default, the direction of the pattern creation is clockwise. If you choose this button then the resulting pattern will be created in the counterclockwise direction. You can also change the direction of the pattern creation using the **Reverse Direction** arrow from the drawing area. Figure 7-26 shows the feature and the temporary axis being selected. Figure 7-27 shows the resulting pattern feature.



**Figure 7-26** The reference to be selected for creating a circular pattern

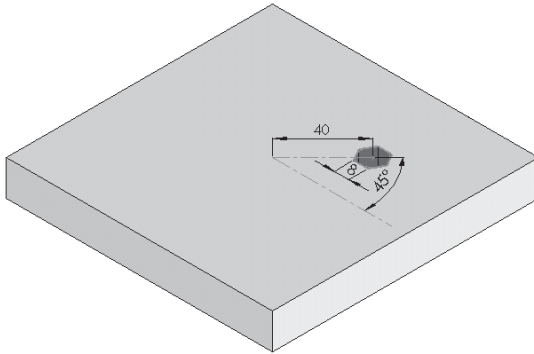
**Figure 7-27** The resulting circular pattern

### Circular Pattern Using a Dimensional Reference

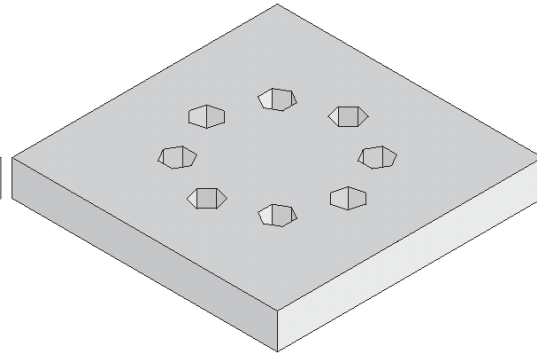
You can also create a circular pattern by selecting an angular dimension. To create a pattern using this option, you need to create an angular dimension in the sketch of the feature that is to be patterned. Invoke the **Circular Pattern PropertyManager** and select the feature to be patterned. The dimensions of the feature to be patterned will be displayed in the drawing area, as shown in Figure 7-28. Now, select the angular dimension and set the value of the total angle and spacing in the **Circular Pattern PropertyManager**. Figure 7-29 shows the pattern created by selecting the angular dimension as the angular reference. The other options available in the **Circular Pattern PropertyManager** are the same as those discussed earlier for the **Linear Pattern PropertyManager**.



**Tip.** In SolidWorks you can pattern a patterned feature. You can also pattern a mirrored feature. The mirror of the pattern feature is also possible in SolidWorks.



**Figure 7-28** Dimensions displayed after selecting the feature to be patterned



**Figure 7-29** Circular pattern created by selecting the angular dimension as the angular reference



**Tip.** Instead of setting the value of the angle and the number of instances in the respective spinners, you can also set the values in the callout. Entering values in the callout is a better option and a timesaving practice.

It is always a good practice to create patterns of features instead of creating complex sketches or repeatedly creating the same feature again and again. It also helps in capturing the design intent of the model. The patterns created in the **Part** mode are very useful in assembly modeling. You will learn more about it in later chapters.

## Creating Sketch-driven Patterns

<b>CommandManager:</b>	Features > Sketch Driven Pattern	(Customize to Add)
<b>Menu:</b>	Insert > Pattern/Mirror > Sketch Driven Pattern	
<b>Toolbar:</b>	Features > Sketch Driven Pattern	(Customize to Add)

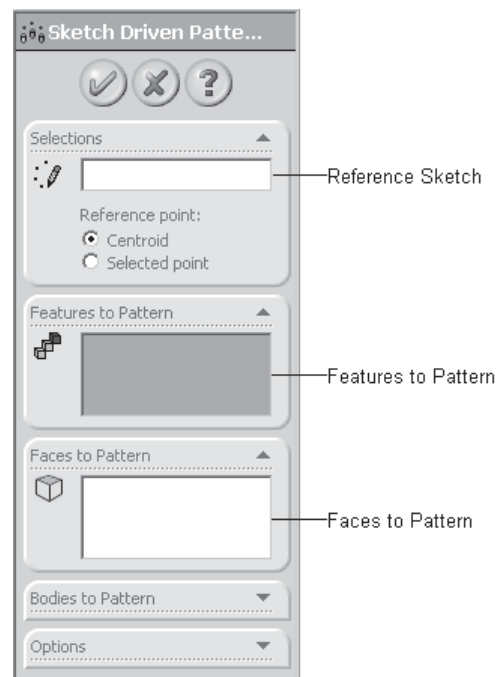


A sketch-driven pattern is created when the features, faces, or bodies are to be arranged in a nonuniform manner, which is neither rectangular nor circular. For creating a sketch-driven pattern, first you have to create an arrangement of the sketch points in a single sketch. This arrangement of sketch points will drive the instances in the pattern feature. After creating the feature to be patterned and placing the points in the sketch, invoke the **Sketch Driven Pattern PropertyManager**, as shown in Figure 7-30. You are prompted to select a sketch for pattern layout, and the face of the feature to be patterned. Select the feature or features to be patterned. Now, click in the **Reference Sketch** display box in the **Selections** rollout and select any one of the sketched point from the drawing area. You can also select the sketch from the **FeatureManager Design Tree** flyout. Choose the **OK** button from the **Sketch Driven Pattern PropertyManager**.

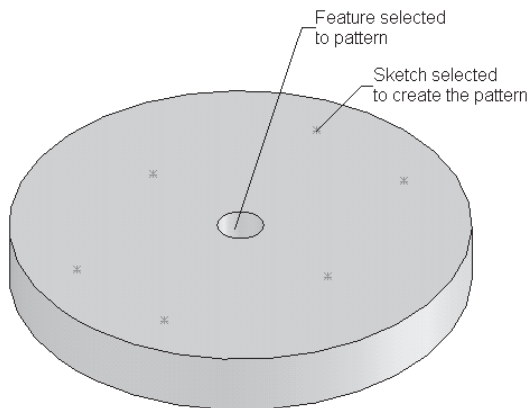


Figure 7-31 shows the feature and the sketch point to be selected and Figure 7-32 shows the resulting pattern feature.

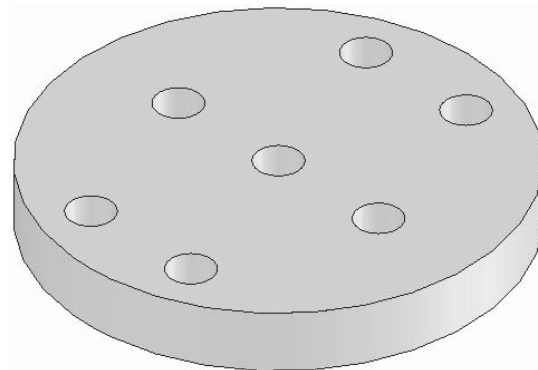
The options available in the **Sketch Driven Pattern PropertyManager** are discussed next.



**Figure 7-30** The Sketch Driven Pattern PropertyManager



**Figure 7-31** The feature and the sketch point to be selected



**Figure 7-32** The resulting pattern feature

### Sketch Driven Pattern Using a Centroid

When you invoke the **Sketch Driven Pattern PropertyManager**, the **Centroid** radio button is selected by default in the **Reference Point** area of the **Selections** rollout. Therefore, the pattern created is with reference to the centroid.

### Sketch Driven Pattern Using a Selected Point

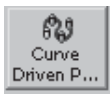
If you select the **Selected point** radio button from the **Reference Point** area of the **Selections** rollout, the pattern created is with reference to the selected point. When you select this radio

button, a **Reference Vertex** selection box is displayed. Select a vertex; the pattern will be created in reference to that vertex.

With this release of SolidWorks, you can also propagate the visual properties to the resulting patterned instances by keeping the **Propagate Visual Properties** check box in the **Options** rollout selected.

## Creating Curve-driven Patterns

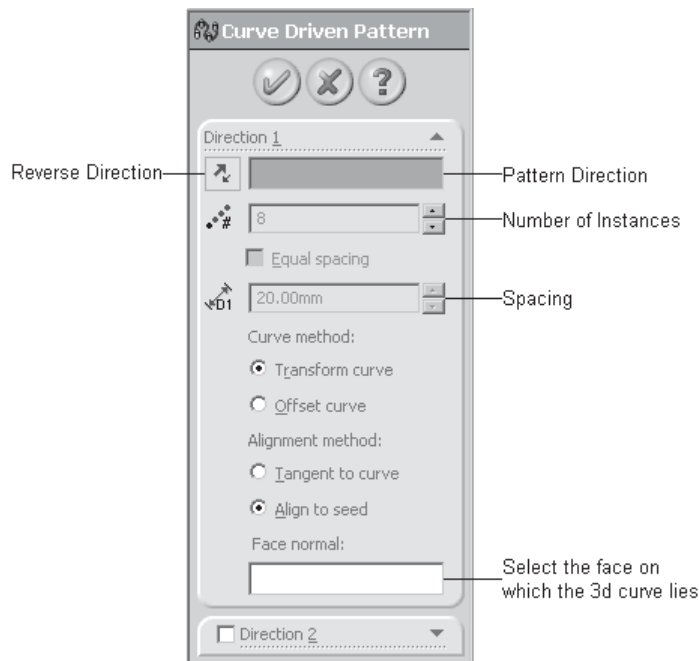
<b>CommandManager:</b>	Features > Curve Driven Pattern	(Customize to Add)
<b>Menu:</b>	Insert > Pattern/Mirror > Curve Driven Pattern	
<b>Toolbar:</b>	Features > Curve Driven Pattern	(Customize to Add)



The **Curve Driven Pattern** option is used to pattern the features, faces, or bodies along a selected reference curve. The reference curve can be a sketched entity or an edge, or an open profile, or a closed loop. To create a pattern using this option, choose the **Curve**



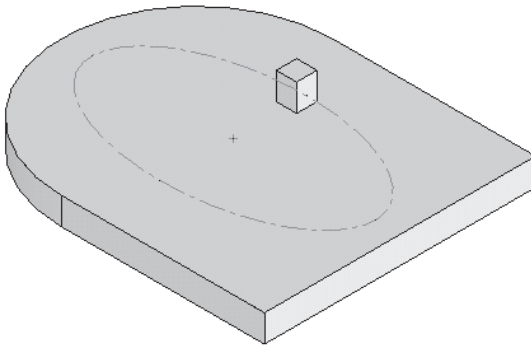
**Driven Pattern** button from the **Features CommandManager** or choose **Insert > Pattern/Mirror > Curve Driven Pattern** from the menu bar. When you choose this button, the **Curve Driven Pattern PropertyManager** is displayed, as shown in Figure 7-33. Figure 7-34 shows the feature and the curve that will be used to create the pattern. Figure 7-35 shows the resulting curve-driven pattern.



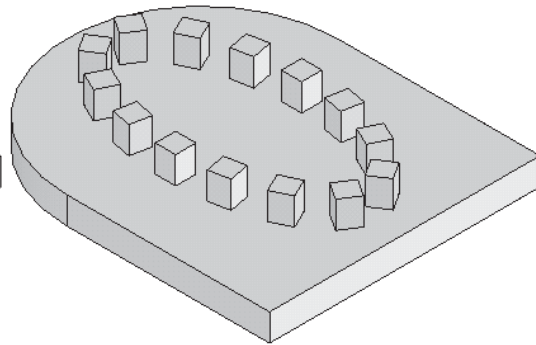
**Figure 7-33** Partial view of the **Curve Driven Pattern PropertyManager**

After invoking the **Curve Driven Pattern PropertyManager**, you are prompted to select an edge, curve, or sketch segment for pattern layout and select a face of feature to be patterned. Select the reference curve along which the feature, face, or body to be patterned. From this





**Figure 7-34** The feature and curve to be used to create the curve-driven pattern



**Figure 7-35** The resulting pattern feature

release of SolidWorks, you can also select 3D curves or sketches as the reference curve. You will learn more about 3D curves and sketches in later chapters. When you select the reference curve, its name is displayed in the **Pattern Direction** selection box and the **Direction 1** callout is also displayed. As discussed earlier, the **Direction 1** callout is divided in two areas. Select the feature to be patterned; the preview of the pattern is displayed in the drawing area. Set the various parameters available in the **Direction 1** rollout and choose the **OK** button from the **Curve Driven Pattern PropertyManager**.

The options available in the **Direction 1** rollout are discussed next.

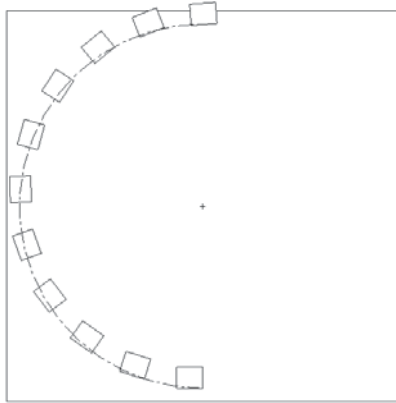
### Equal Spacing

The **Equal Spacing** check box is used to accommodate all the instances of the pattern along the selected curve. By default, this check box is cleared. Therefore, you have to specify the distance between the instances and the total number of instances to be created along the selected curve. When you select this check box, the **Spacing** spinner is not available and you have to specify only the total number of instances. The distance between the instances is calculated automatically.

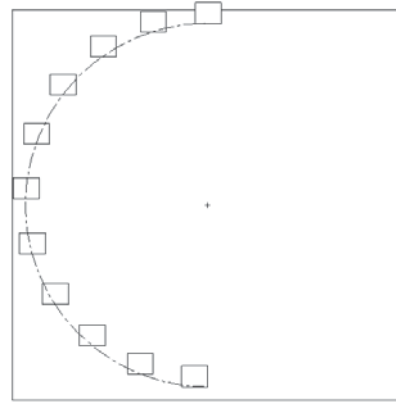
### Curve method and Alignment method

The **Curve method** area of the **Direction 1** rollout is used to specify the type of curve method to be followed while creating patterns. The two options available in this area are in the form of the **Transform curve** and **Offset curve** radio buttons. The **Alignment method** area of the **Direction 1** rollout is used to specify the type of alignment method to be applied. The two alignment methods are the **Tangent to curve** method and the **Align to seed** method. Figure 7-36 shows the curve-driven pattern created with selected **Transform curve** and the **Tangent to curve** radio buttons. Figure 7-37 shows the curve-driven pattern created with the **Transform curve** and **Align to seed** radio buttons selected.

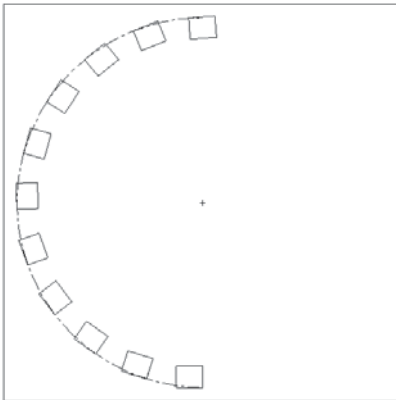
Figure 7-38 shows the curve-driven pattern created with the **Offset curve** and the **Tangent to curve** radio buttons selected. Figure 7-39 shows the curve-driven pattern created with the **Offset curve** and **Align to seed** radio buttons selected.



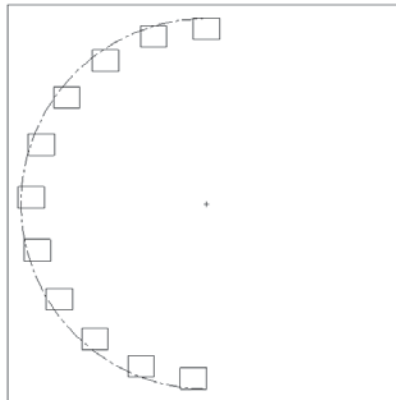
**Figure 7-36** Pattern created with **Transform curve** and **Tangent to curve** radio buttons selected



**Figure 7-37** Pattern created with **Transform curve** and **Align to seed** radio buttons selected

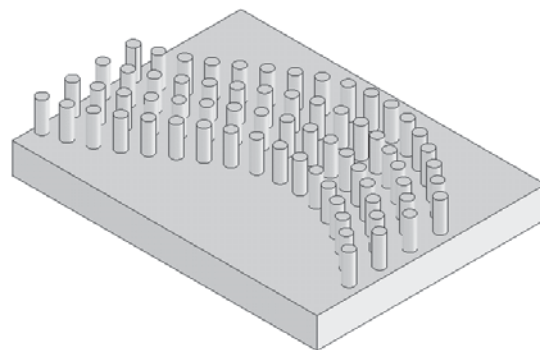


**Figure 7-38** Pattern created with **Offset curve** and **Tangent to curve** radio buttons selected



**Figure 7-39** Pattern created with **Offset curve** and **Align to seed** radio buttons selected

Other options available in the **Curve Driven PropertyManager** are the same as those discussed earlier for the mirror and other pattern features. By selecting the check box available in the **Direction 2** rollout, you can also specify the parameters in the second direction. Figure 7-40 shows the curve-driven pattern feature created with the pattern defined in the first and second direction.



**Figure 7-40** A curve-driven pattern created by specifying parameters in both the directions

Creating Table-driven Patterns

**CommandManager:**

Features > Table Driven Pattern

(Customize to Add)

**Menu:**

Insert > Pattern/Mirror > Table Driven Pattern

**Toolbar:**

Features > Table Driven Pattern

(Customize to Add)



The table driven pattern is created by specifying the X and Y coordinates with reference to a coordinate system. The instances of the selected features, faces, or bodies are created at the points specified using the X and Y coordinates. For creating this pattern, you first need to create a coordinate system using the **Coordinate System** button from the **Reference Geometry** toolbar. The coordinate system defines the direction along which the selected feature will be patterned. Choose the **Table Driven Pattern** button from the **Features CommandManager** or choose **Insert > Pattern/Mirror > Table Driven Pattern** from the menu bar. The **Table Driven Pattern** dialog box is displayed, as shown in Figure 7-41.

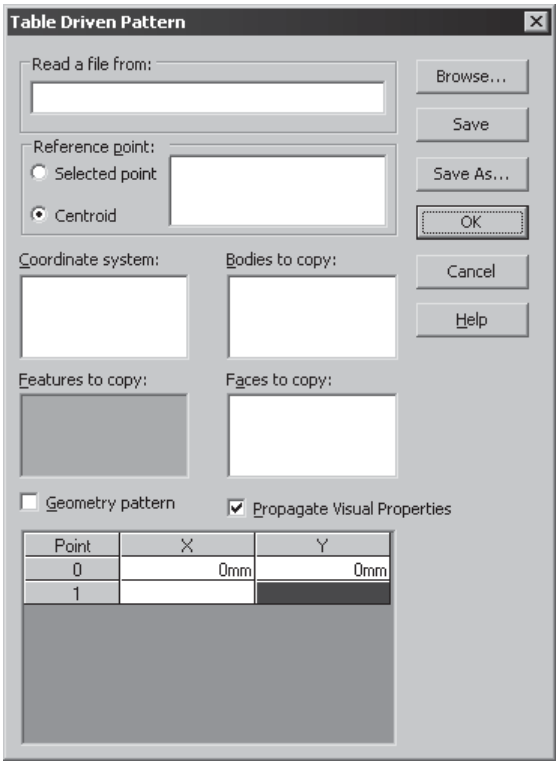
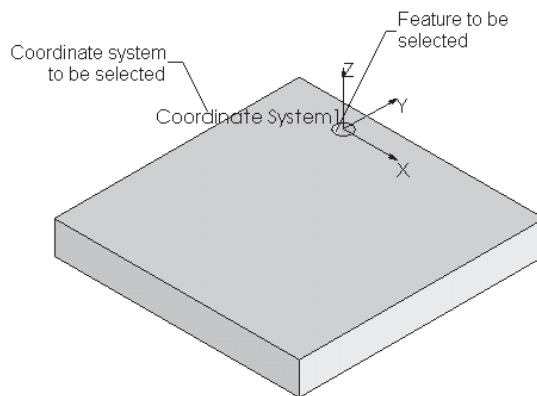
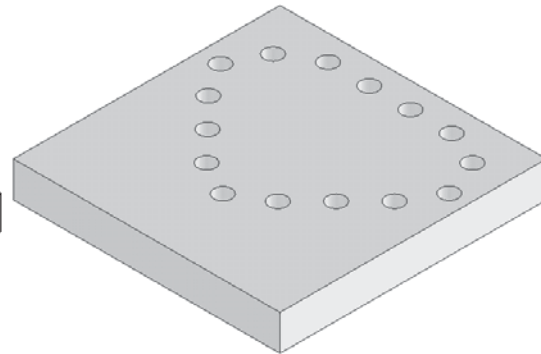


Figure 7-41 The Table Driven Pattern dialog box

Select the feature to be patterned and the coordinate system, from the drawing area or from the **FeatureManager Design Tree**. Enter the coordinates for creating the instances in the **Coordinate points** area of the **Table Driven Pattern** dialog box. As you enter the coordinates for the instances, the preview of the pattern is displayed in the drawing area. After entering all the coordinate points, choose the **OK** button from the **Table Driven Pattern** dialog box. Figure 7-42 shows the feature and the coordinate system to be selected. Figure 7-43 shows the



**Figure 7-420** Feature and the coordinate system to be selected for creating a table-driven pattern



**Figure 7-43** The resulting pattern after specifying the coordinate points

table-driven pattern created after entering the coordinate values in the **Table Driven Pattern** dialog box.

You can also save the table-driven pattern file and retrieve the same coordinates by simply browsing the saved file using the **Browse** button from the **Table Driven Pattern** dialog box. You can also simply write the coordinates in a text file and browse the same file while creating a table-driven pattern. The other options available in this dialog box are the same as those discussed earlier.

## Creating Rib Features

<b>CommandManager:</b>	Features > Rib
<b>Menu:</b>	Insert > Features > Rib
<b>Toolbar:</b>	Features > Rib



Ribs are defined as the thin walled structures that are used to increase the strength of the entire structure of the component, so that it does not fail under an increased load. In SolidWorks, the ribs are created using an open sketch as well as a closed sketch. To create a rib feature, invoke the **Rib PropertyManager** and select the plane on which you need to draw the sketch for creating the rib feature. Draw the sketch and exit the sketching environment. Specify the rib parameters in the **Rib PropertyManager** and view the detailed preview using the **Detailed Preview** button. The **Rib** tool is invoked by choosing the **Rib** button from the **Features CommandManager** or by choosing **Insert > Features > Rib** from the menu bar. After invoking the **Rib** tool, draw the sketch and exit the sketching environment; the **Rib PropertyManager** is displayed, as shown in Figure 7-44.

The preview of the rib feature with the direction arrow and the confirmation corner is displayed in the drawing area. Figure 7-45 shows the sketch drawn for the rib feature and Figure 7-46 shows the resulting rib feature.

The options available in the **Rib PropertyManager** are discussed next.

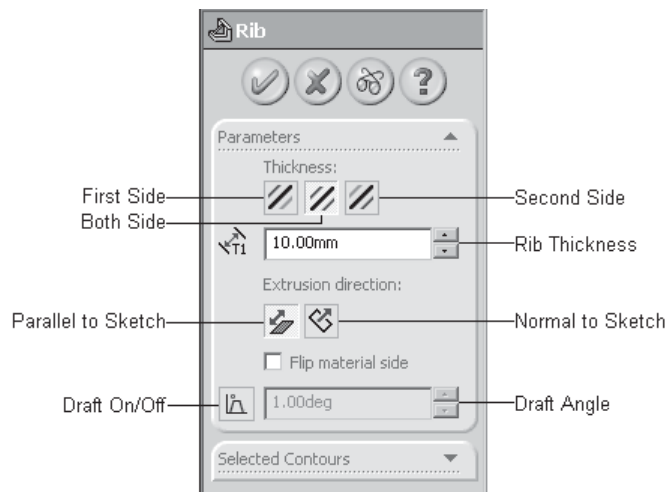


Figure 7-44 The Rib PropertyManager

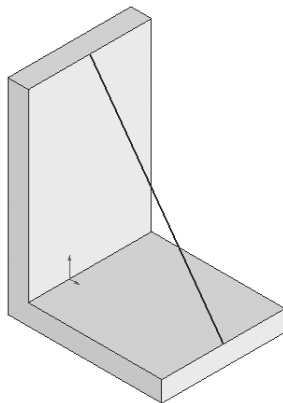


Figure 7-45 Sketch for the rib feature

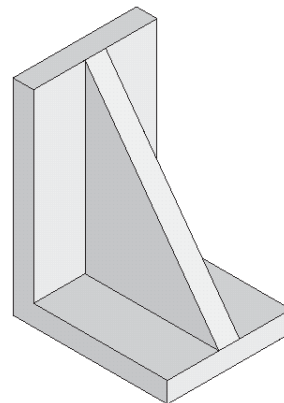
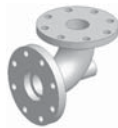


Figure 7-46 The resulting rib feature



**Tip.** You can also create the rib feature by first drawing the sketch and then invoking the **Rib** tool within the sketching environment. If you exit the sketching environment after creating the sketch for a rib feature, invoke the **Rib** tool and select the sketch from the drawing area.

### Thickness

The **Thickness** area of the **Parameters** rollout is used to specify the side of the rib thickness and the thickness of the rib feature. The buttons available in this area are used to control the side on which you want to add the rib thickness. By default, the **Both Sides** button is chosen. Therefore, the rib is created on both sides of the sketch. You can choose the **First Side** or **Second Side** button to create ribs on either side of the sketch. The **Rib Thickness** spinner in this area of the **Parameters** rollout is used to specify the rib thickness.

## Extrusion direction

The **Extrusion direction** area of the **Parameters** rollout is used to specify the method of extruding the closed or open sketch. When you invoke the **Rib PropertyManager**, by default the option that is suitable for creating the rib feature will be active, depending on the geometric conditions. The options available in this area are discussed next.

### Parallel to Sketch

The **Parallel to Sketch** option is used to extrude the sketch in a direction that is parallel to both the sketch and sketching plane. When you invoke the **Rib PropertyManager** and the sketch created for the rib feature is a continuous single entity open sketch, then this option is selected by default. Figure 7-47 shows an open sketch suitable for creating a rib using the **Parallel to Sketch** option. Figure 7-48 shows the rib feature created using the sketch.

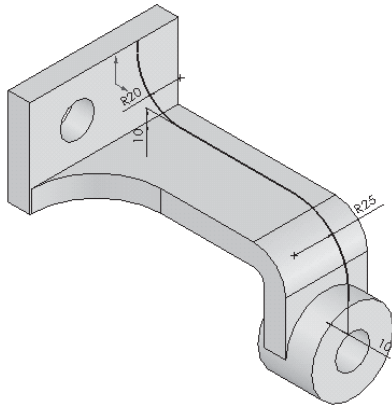


Figure 7-47 An open sketch for the rib feature

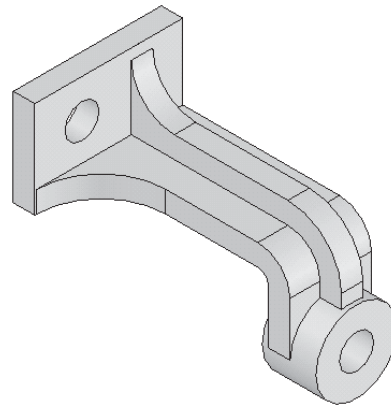


Figure 7-48 The resulting rib feature

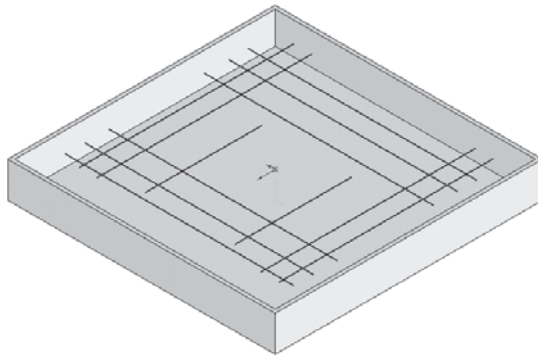
### Normal to Sketch

The **Normal to Sketch** option is used to create a rib feature when the sketch of the rib feature is a closed loop sketch, or it consists of multiple sketched entities. The sketch with multiple entities can be closed loops or open profiles. If you draw a sketch with a closed loop or with multiple sketched entities and invoke the **Rib** tool, the **Normal to Sketch** button will be selected by default. You can also choose the **Normal to Sketch** button from the **Extrusion Direction** area to use this option. Figure 7-49 shows a multiple entity sketch for the rib feature. Figure 7-50 shows the resulting rib feature.

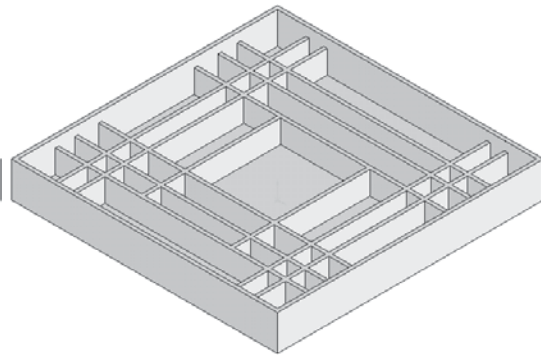
When this option is selected, the **Type** area is displayed under the **Draft Angle** spinner. The options available in the **Type** area are discussed next.

### Type

The **Type** area is available only when you choose the **Normal to Sketch** button from the **Extrusion direction** area of the **Parameters** rollout. The **Type** area is provided with two radio buttons, **Linear** and **Natural**. These radio buttons are used if the endpoints of the open sketch for the rib are not coincident with the faces of the existing feature. If the **Linear Radio** button is selected, the rib is created by extending



**Figure 7-49** Multiple entities for the rib feature



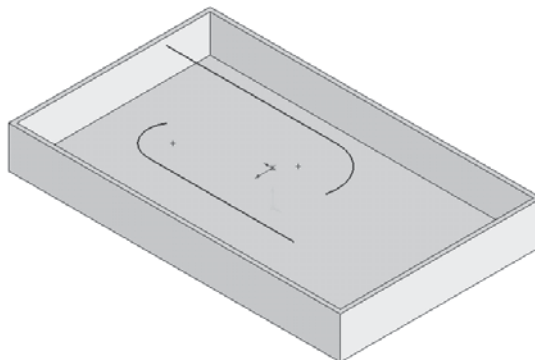
**Figure 7-50** The resulting rib feature



**Tip.** You will observe that the endpoints of the sketched lines drawn in Figure 7-47 do not merge with the model edges. However, the rib created using this sketch merges with the model edges. This is because while creating the sketch for the rib feature, you do not have to create a complete sketch. The ends of the rib feature automatically extend to the next surface.

the sketch normal to the sketched entity direction. The sketch will be extended up to a point where it meets the boundary.

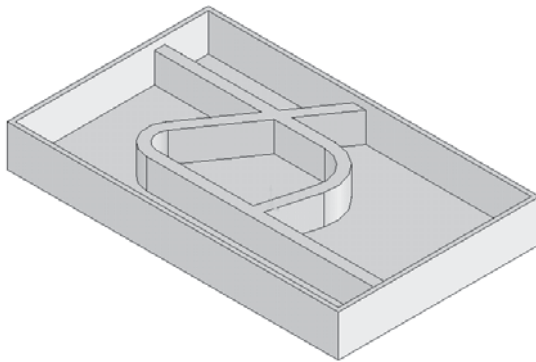
On the other hand, if the **Natural** radio button is selected, the rib feature is created by extending the sketch along the direction of the sketched entities. For example, consider the sketch shown in Figure 7-51 which shows a multiple entity sketch created for the rib feature.



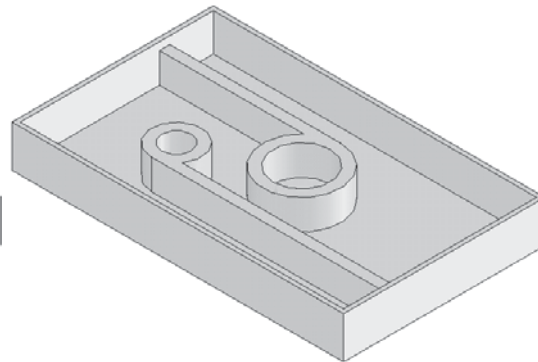
**Figure 7-51** Sketch for the rib feature

Figure 7-52 shows a rib feature created by extending the sketch normal to the arc and the line. This is because the **Linear** radio button is selected. Similarly, in Figure 7-53, the feature is created by extending the sketch along the line and arc using the **Natural** radio button. This is the reason a circular feature is created at the end where the sketch has the arc.





**Figure 7-52** Rib feature created with the **Linear** radio button selected from the **Type** area of the **Rib PropertyManager**



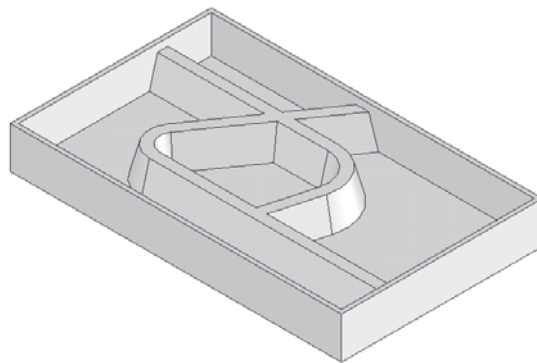
**Figure 7-53** Rib feature created with the **Natural** radio button selected from the **Type** area of the **Rib PropertyManager**

### Flip material side

The **Flip material side** check box is selected to reverse the direction of material addition, while creating the rib feature. You can also reverse the direction of material addition using the **Flip material side** arrow available in the drawing area.

### Draft On/Off

The **Draft On/Off** button is used to add the taper to the faces of the rib feature. When you choose the **Draft On/Off** button, the **Draft Angle** spinner is invoked. If you are creating a rib feature using multiple sketched entities, you can add only a simple draft to it. Figure 7-54 shows the draft angle added to the rib feature. By default, the draft is added inwards to the rib feature.

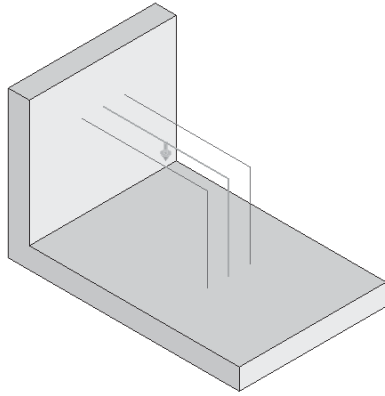


**Figure 7-54** Draft angle added to the rib feature

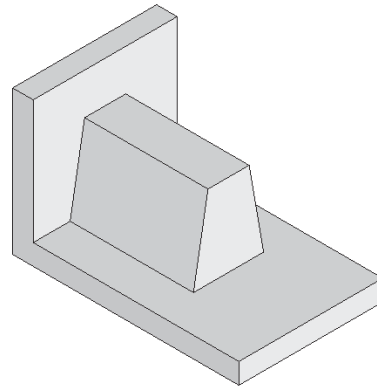
Using the **Draft outward** check box you can add the draft outwards. If the rib feature to be created consists of a single continuous sketch and if you choose the **Draft On/Off** button, the **Next Reference** button is displayed below the **Draft Angle** spinner. A reference arrow is also displayed in the drawing area. Using the **Next Reference** button, you can cycle through the reference along which you want to add the draft angle.



Figure 7-55 shows the preview of the rib feature and Figure 7-56 shows the resulting rib feature with the **Draft outward** check box selected.

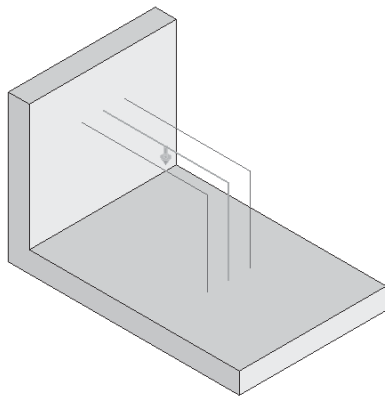


**Figure 7-55** Preview of the rib feature

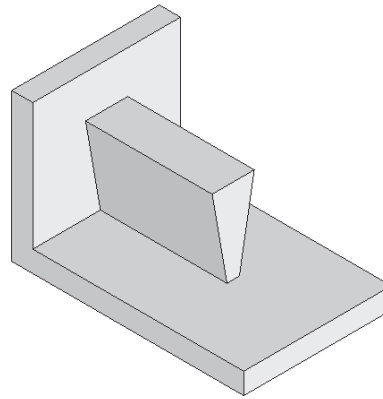


**Figure 7-56** The resulting rib feature with the **Draft outward** check box selected

Figure 7-57 shows the preview of the rib feature and Figure 7-58 shows the resulting rib feature with the **Draft outward** check box cleared.



**Figure 7-57** Sketch selected to create a rib feature with an inward draft



**Figure 7-58** The resulting rib feature

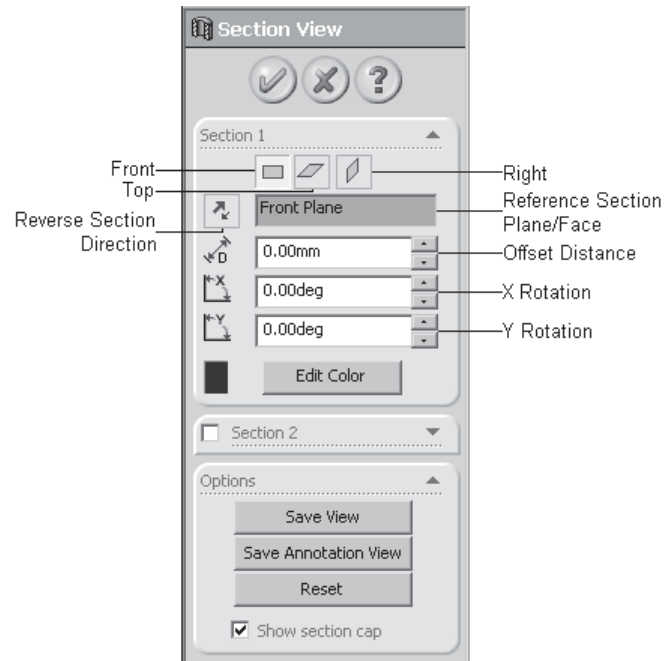
## Displaying the Section View of the Model

<b>CommandManager:</b>	View > Section View (Customize to Add)
<b>Menu:</b>	View > Display > Section View
<b>Toolbar:</b>	View > Section View



The **Section View** tool is used to display the section view of the model by cutting it using a plane or face. To display the section view of a model, choose the **Section View** button from the **View CommandManager**, or choose **View > Display > Section View** from the menu bar. You need to invoke the **View CommandManager**,

if it not is not available by default. Move the cursor on any one of the buttons available in **CommandManager** and right-click to invoke the shortcut menu. Choose the **Customize CommandManager** option from it. The **Customizing CommandManager** menu is displayed. Select the **View** check box to display the **View CommandManager** and click once anywhere in the drawing area. On invoking this tool, the **Section View PropertyManager** is displayed, as shown in Figure 7-59.



*Figure 7-59 The Section View PropertyManager*

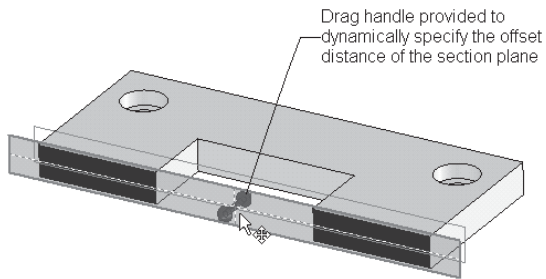
By default, the **Front Plane** is automatically selected in the **Section View PropertyManager** and the section view of the model, created using the **Front Plane** as the section plane, is displayed in the drawing area. A drag handle is provided in the drawing area to dynamically adjust the offset distance of the section plane, as shown in Figure 7-60.

If you need to select the **Right Plane** or **Top Plane** as the section plane, choose the respective buttons available in the **Section 1** rollout. You can also select a face or a user-defined plane as the section plane. You can also specify the offset distance using the **Offset Distance** spinner. You will observe that as you modify the offset distance, the preview of the section view is automatically modified. You can also rotate the section plane along the X axis and the Y axis using the **X Rotation** and **Y Rotation** spinners respectively.

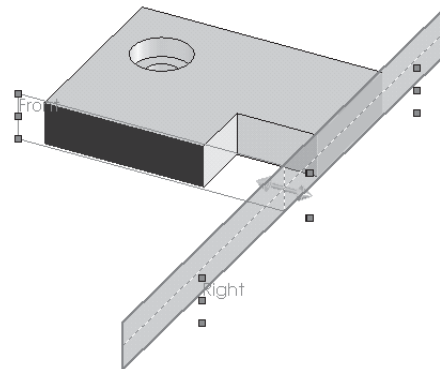
The **Edit Color** button is used to modify the color of the preview of the section cap. This color is displayed only when the **Section View PropertyManager** is active. When you exit the **PropertyManager**, the color is not displayed in the section view.

To create a half section view, you need to invoke the **Section 2** rollout and specify the section

plane in it. You can also specify the offset distance and the rotation of the plane, as discussed earlier for **Section 1**. After setting all the parameters, choose the **OK** button from the **Section View PropertyManager**. Figure 7-61 shows the preview of the model after defining the section plane.



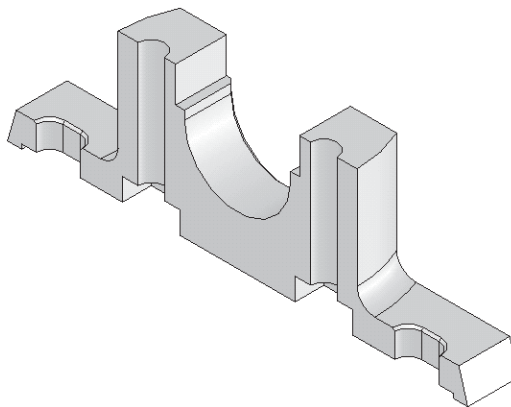
**Figure 7-60** Drag handle for dynamic offset



**Figure 7-61** Section preview after selecting the second section plane

You can also define the third section plane using the **Section 3** rollout. This rollout is displayed only after you invoke the **Section 2** rollout.

The options available in the **Options** rollout are used to save the section view as a named view. You can retrieve the saved section view from the **Orientation** dialog box at any stage of your design cycle. To save a named view, choose the **Save View** button from the **Options** rollout. The **Named View** dialog box is displayed; specify the name of the view and choose the **OK** button from this dialog box. Using the **Reset** button you can reset the section view setting to the default settings. The **Show section cap** check box is selected by default. If you clear this check box, the preview of the section view is not capped. Figure 7-62 shows the section view of a model.



**Figure 7-62** Section view of a model



**Tip.** You can also rotate the section plane dynamically by moving the cursor on the edge of the section plane. The cursor will be replaced by the rotate cursor. Drag the cursor to dynamically rotate the section plane.

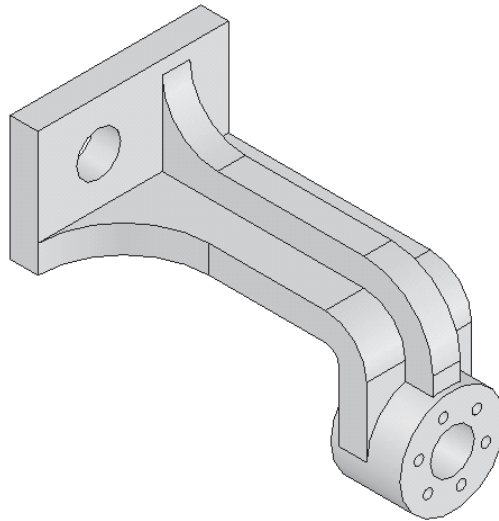
To switch back to the full view mode, you need to choose the **Section View** button from the **View CommandManager**. You can also select any face of the sectioned model and then right-click to invoke the shortcut menu. Choose the **Section View** option from the shortcut menu.

To modify the section view, select any face of the sectioned model and invoke the shortcut menu. Choose the **Section View Properties** option from it. The **Section View PropertyManager** is displayed and you can modify the section view.

## TUTORIALS

### Tutorial 1

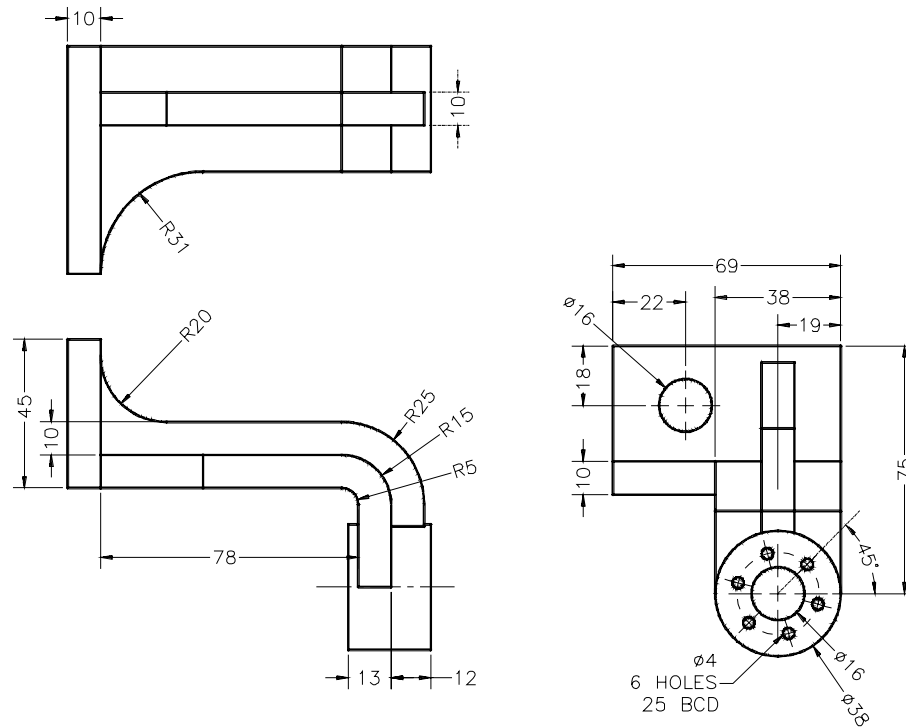
In this tutorial, you will create the model shown in Figure 7-63. The dimensions of the model are shown in Figure 7-64. **(Expected time: 30 min)**



**Figure 7-63** Solid model for Tutorial 1

The steps to be followed to complete this tutorial are discussed next.

- Create the base feature of the model by extruding a rectangle of 69 mm x 45 mm, created on the **Right Plane** to a depth of 10 mm, refer to Figure 7-65.
- Create the second feature, which is created by extruding the sketch created on the back face of the base feature, refer to Figure 7-66.
- The third feature of the model is a circular feature, refer to Figure 7-67.



**Figure 7-64** Views and dimensions of the model for Tutorial 1

- d. Create the hole on the specified BCD and pattern the hole feature using the circular pattern option.
- e. Create the hole feature on the base feature, refer to Figure 7-68.
- f. Create a fillet feature to add the required fillets, refer to Figures 7-69 and 7-70.
- g. Create the rib feature, refer to Figures 7-71 and 7-72.

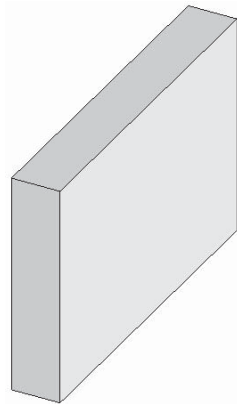
### Creating the Base Feature

1. Start SolidWorks and then start a new part document from the **New SolidWorks Document** dialog box.

It is evident from the model that the sketch of its base feature is drawn on the **Right Plane**. Therefore, you need to select the **Right Plane** from the **FeatureManager Design Tree** to create the base feature.

2. Select the **Right Plane** from the **FeatureManager Design Tree** and choose the **Extruded Boss/Base** button from the **Features CommandManager**. The sketching environment is invoked and the **Right Plane** is oriented normal to the view.

3. Draw the sketch of the base feature of the model, which consists of a rectangle of dimensions 69 mm x 45 mm.
4. Add the required relations and dimensions to the sketch. Exit the sketching environment.
5. Set the value of the **Depth** spinner to 10 mm and exit the **Extrude PropertyManager**. The base feature of the model is shown in Figure 7-65.



*Figure 7-65 Base feature of the model*

### Creating the Second Feature of the Model

The second feature of the model is also an extruded feature. Draw the sketch for the second feature on its back face, and extrude this sketch to the given depth.

1. Select the back face of the base feature as the sketching plane and invoke the **Extruded Boss/Base** tool.
2. Draw the sketch of the second feature and add the required relations and dimensions.
3. Exit the sketching environment and select the **Reverse Direction** button from the **PropertyManager**.
4. Set the value of the **Depth** spinner to **38** and end the feature creation.

The model, after creating the second feature, is shown in Figure 7-66.



#### Note

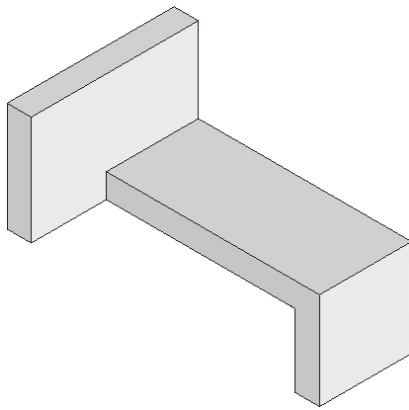
*You can also create the first and second features using the contour selection method.*

### Creating the Third Feature

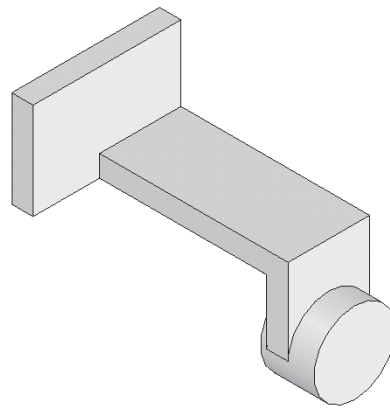
The third feature of this model is a circular extruded feature, which is created by extruding a circular sketch on both sides of the sketching plane. The sketch for this feature is drawn on the right planar face of the second feature.

1. Select the right planar face of the second feature as the sketching plane and invoke the **Extruded Boss/Base** tool.
2. Draw the sketch using the **Circle** tool. Add the required relations and dimensions it.
3. Exit the sketching environment and set the value of the **Depth** spinner available in the **Direction 1** rollout to **12**. Since you have to extrude the sketch in both the directions with variable values, therefore, you need to invoke the **Direction 2** rollout. Set the value of the **Depth** spinner available in the **Direction 2** rollout to **13** and end the feature creation.

Figure 7-67 shows the model, after adding the third feature.



**Figure 7-66** Second feature added to the model



**Figure 7-67** Third feature added to the model

### Creating the Fourth Feature

The fourth feature of this model is a hole feature. You will create a hole using the **Simple Hole** option on the right face of the third feature. To create a hole using this option, you need to place a point on which the hole will be placed.

1. Select the right face of the third feature as the sketching plane for placing the sketch point and invoke the sketching environment. The center point of the hole to be created, using the **Hole PropertyManager**, will be placed coincident to this sketched point.
2. Place a point on the right face of the circular feature.
3. Add the **Concentric** relation between the sketched point and the circular edge of the third feature and exit the sketching environment.
4. Now, select the face on which the point is placed. Choose the **Simple Hole** button from the **Features CommandManager**, or choose **Insert > Features > Hole > Simple** from the menu bar to invoke the **Hole PropertyManager**.

5. Choose the **Through All** option from the **End Condition** drop-down list and set the value of the **Hole Diameter** spinner to **16**.
6. Select the center point of the hole feature and drag the cursor to the sketched point drawn earlier. Release the left mouse button when the cursor snaps the sketched point.
7. Choose the **OK** button from the **Hole PropertyManager**.

### Creating the Fifth Feature

1. Using the procedure given in the previous section, create the fifth feature, which is also a hole feature placed on the same placement plane. The hole feature is created using the **Through All** option, with the diameter of the hole as **4**. Next, define the placement of the feature by adding the required relations and dimensions.

### Patterning the Hole Feature

After creating the fifth feature, which is a hole feature, you will pattern it using the **Circular Pattern** tool.

1. In case all the buttons of the **CommandManager** are not visible, select the black arrow; a flyout appears. Choose the **Circular Pattern** button from the **CommandManager** flyout.



The **Circular Pattern PropertyManager** is invoked and you are prompted to select an edge or axis for direction reference; select a face of the feature for pattern features.

To create a circular pattern, you need an edge or axis that will be used as the central axis. As discussed in the earlier chapters, when you create a circular feature, a temporary axis is automatically created passing through its center. In this tutorial, the feature to be patterned is created on a circular feature. Therefore, you will display the temporary axis of the circular feature which will be used as the central axis.

2. Choose **View > Temporary Axes** from the menu bar to display the temporary axis.

The temporary axes are displayed in the model.

3. Select the temporary axis that passes through the center of the first hole feature.

The **Direction 1** callout is displayed and is attached to the selected axis.

4. Click in the **Features to Pattern** selection box to invoke the selection mode.
5. Select the smaller hole from the drawing area, or expand the **FeatureManager Design Tree**, which is displayed in the drawing area, and select the **Hole2** feature.

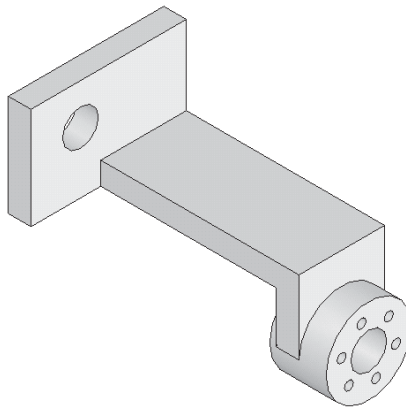
The preview of the pattern feature is displayed with the default settings.



6. Select the **Equal spacing** check box, if cleared and set the value of the **Number of Instances** spinner to **6**. Choose the **OK** button from the **PropertyManager**.
7. Choose **View > Temporary Axes** to hide the temporary axes.

### Creating the Seventh Feature

1. The seventh feature of this model is a hole feature. You will create this hole feature using the procedure given to create the fourth feature. This hole feature will be placed on the right planar face of the base feature. Therefore, after selecting the right planar face of the base feature, place the hole feature. Next, define the placement of the hole feature by adding the required relations and dimensions. Figure 7-68 shows the model after adding the hole features.



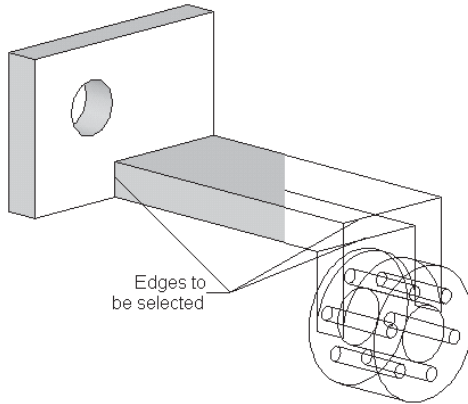
*Figure 7-68 Model after adding all the hole features*

### Creating the Fillet Feature

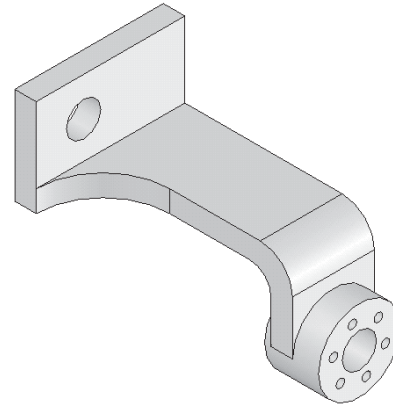
Next, you need to create the fillet feature. It is evident from the model that the fillets to be added to the model are of different radii. In SolidWorks, you can specify different radii to individual selected edges, faces, or loops in a single fillet feature.

1. Choose the **Fillet** button from the **Features CommandManager** to invoke the **Fillet PropertyManager**.  
  
You are prompted to select edges, faces, features, or loops to be filleted.
2. Select the **Multiple radius fillet** check box from the **Items To Fillet** rollout of the **Fillet PropertyManager**.
3. Select the edges to fillet, as shown in Figure 7-69. As the **Multiple radius fillet** check box is selected, therefore, each selected edge has a separate **Radius** callout.
4. Modify the value of the radii, as required in the respective **Radius** callouts.
5. Choose the **OK** button from the **Fillet PropertyManager**.

The isometric view of the model, after adding the fillet feature, is shown in Figure 7-70.



**Figure 7-69** Edges to be selected



**Figure 7-70** Model after adding the fillet feature

### Creating the Rib Feature

The next feature that you need to create is a rib feature. The sketch of the rib feature needs to be drawn on a sketching plane at an offset distance from the back planar face of the model. Therefore, first you need to create a reference plane at an offset distance from the back planar face of the model.

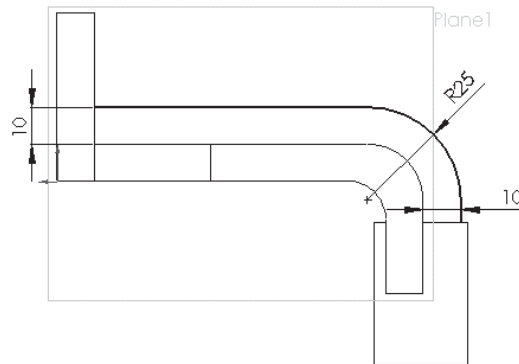
1. Invoke the **Plane PropertyManager**.
2. Rotate the model and select its back planar face. Choose the **Reverse direction** check box under the **Distance** spinner and set the value of the **Distance** spinner to **19**.
3. Choose the **OK** button from the **Plane PropertyManager** to end the feature creation.

A new plane is created at an offset distance from the back planar face of the model.

4. Choose the **Rib** button from the **Features CommandManager**.
5. Create the sketch for the rib feature on the newly created plane and add the required relations and dimensions to the sketch, as shown in Figure 7-71.
6. Exit the sketching environment. The **Rib PropertyManager** is displayed.

The preview of the rib feature is displayed in the drawing area and you will observe that the direction of material addition is displayed by an arrow in the drawing area. The direction of material addition is opposite to the required direction. Therefore, you need to flip its direction.

7. Select the **Flip material side** check box to flip the direction of the material addition.

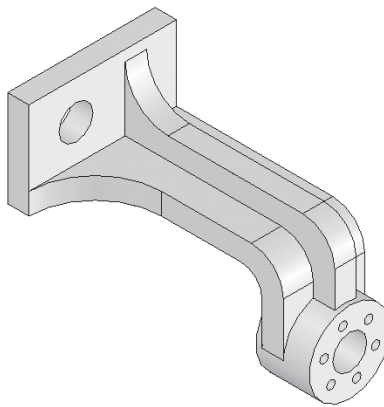


**Figure 7-71** Sketch for the rib feature

The default value of the rib thickness is 10, which is the required value, and so you do not need to change it.

8. Choose **OK** from the **Rib PropertyManager**. Also, hide the newly created plane.

The last feature of the model is the fillet feature. Add the fillet feature on the left edge of the rib using the **Fillet** tool. Figure 7-72 shows the isometric view of the final model.



**Figure 7-72** The final solid model

### Saving the Model

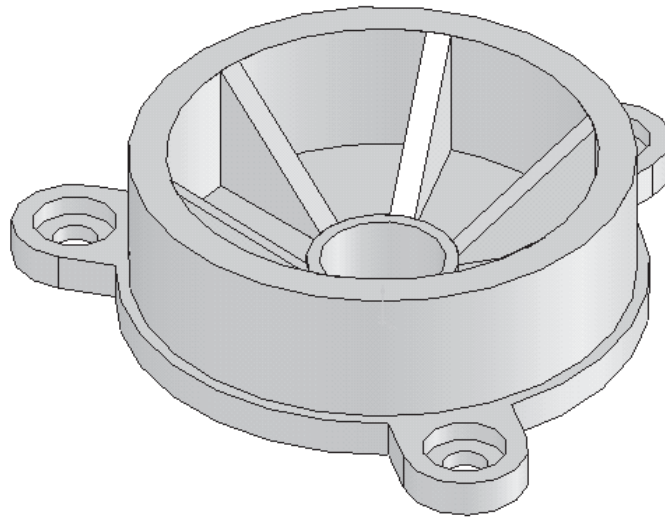
1. Create a *c07* folder in the *SolidWorks* folder and choose the **Save** button from the **Standard** toolbar. Save the model with the name given below.

`\My Documents\SolidWorks\c07\c07tut1.sldprt`

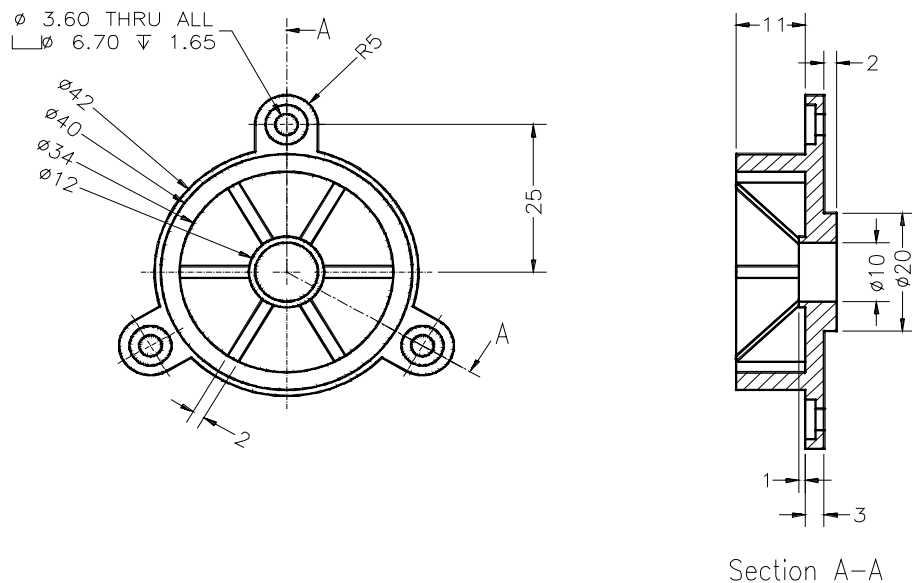
2. Choose **File > Close** from the menu bar to close the document.

## Tutorial 2

In this tutorial you will create the model shown in Figure 7-73. The dimensions of the model are shown in Figure 7-74. **(Expected time: 30 min)**



*Figure 7-73 Model for Tutorial 2*



*Figure 7-74 Front view and aligned section view with dimensions*

The steps to be followed to complete this tutorial are discussed next.

- Create the base feature of the model by revolving the sketch along a centerline, refer to Figures 7-75 and 7-76.
- Create the second feature, which is on the outer periphery of the base feature, by extruding the sketch from the sketch plane to a selected surface, refer to Figures 7-77 and 7-78.
- Place a counterbore hole feature on the top face of the second feature using the hole wizard.
- Pattern the second and third features along the temporary axis using the **Circular Pattern** tool, refer to Figure 7-79.
- Create the rib feature, refer to Figure 7-80.
- Pattern the rib feature along a temporary axis using the **Circular Pattern** tool, refer to Figure 7-81.

### Creating the Base Feature

Start a new SolidWorks part document. First you need to create the base feature of the model by revolving the sketch along the axis of revolution. The axis of revolution will be a centerline and the sketch for the base feature will be drawn on the **Right Plane**.

- Invoke the **Revolved Boss/Base** tool and select the **Right Plane** as the sketching plane.
- Create the sketch for the base feature and add the required relations and dimensions to the sketch, as shown in Figure 7-75.

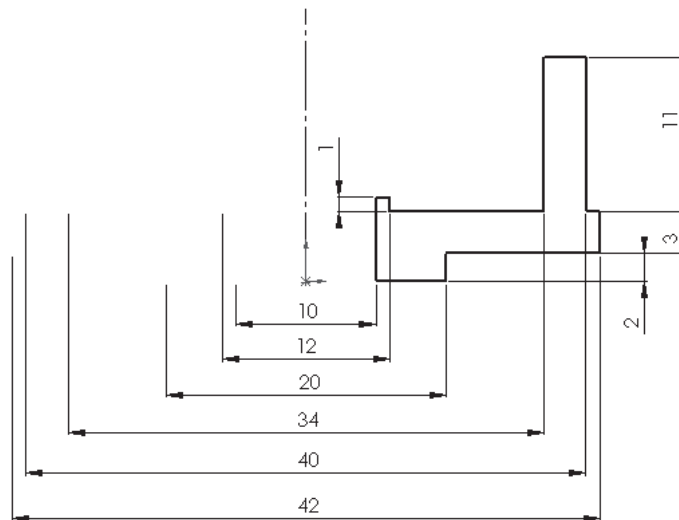
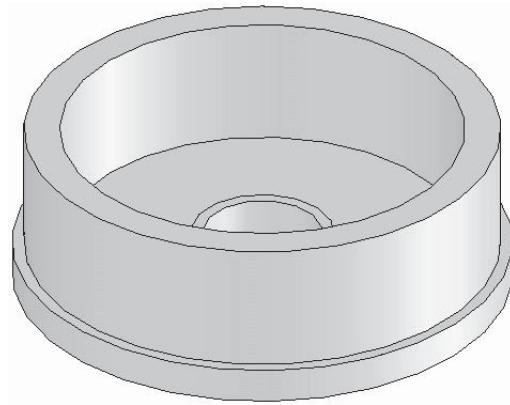


Figure 7-75 Sketch for the base feature

- Exit the sketching environment and set the value of the **Angle** spinner to **360**.
- Choose the **OK** button from the **Revolve PropertyManager**. The base feature created after revolving the sketch is shown in Figure 7-76.

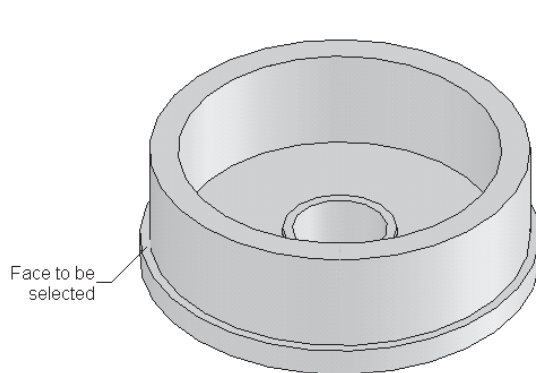


*Figure 7-76 Base feature of the model*

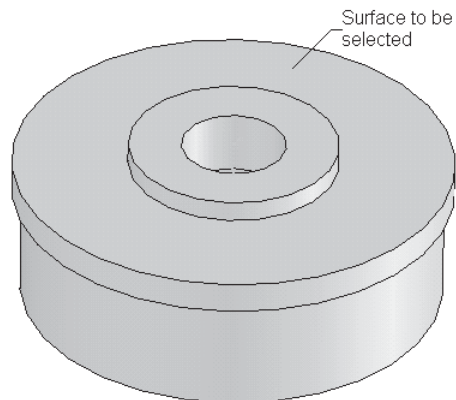
### Creating the Second Feature

The second feature is created by extruding a sketch up to the selected surface.

1. Invoke the **Extruded Boss/Base** tool and select the face shown in Figure 7-77 as the sketching plane.
2. Create the sketch of the second feature and add the required relations and dimensions to the sketch.
3. Exit the sketching environment. Use the **Up To Surface** option to extrude the sketch. The surface to be selected is shown in Figure 7-78.



*Figure 7-77 Face to be selected*



*Figure 7-78 Surface to be selected*

4. Choose the **OK** button from the **Extrude PropertyManager**.

### Creating the Hole Feature

It is evident from Figure 7-74 that a counterbore hole needs to be added to the model by

using the **Hole Wizard**. Before invoking this tool, select the placement plane for the hole feature.

1. Select the top face of the second feature as the placement plane for the hole feature.
2. Choose the **Hole Wizard** button from the **Features CommandManager** to invoke the **Hole Definition** dialog box.

The preview of the hole feature with the default settings in the **Hole Definition** dialog box is displayed in the drawing area.

3. Invoke the **Counterbore** tab, if it is not invoked by default and select the **Ansi Metric** option from the **Standard** drop-down list to specify the standard to be used.
4. Choose the **Socket Button Head Cap Screw - ANSI B18.3.4M** option from the **Screw Type** drop-down list to specify the type of screw.
5. Select the **M3** option from the **Size** drop-down list to specify the size of the fastener to be used in the hole. Leave all the default options as they are and choose the **Next** button from the **Hole Definition** dialog box.

The **Hole Placement** dialog box is displayed.

6. Invoke the **Add Relations PropertyManager** and apply a concentric relation between the center point of the hole feature and the circular edge of the second feature.
7. Choose the **Finish** button from the **Hole Placement** dialog box to end the feature creation.

### Patterning the Features

After creating the second and third feature, you need to pattern them about the temporary axis using the **Circular Pattern** tool.

1. Choose the **Circular Pattern** button from the **Features CommandManager** to invoke the **Circular Pattern PropertyManager**.



You need to define an axis as the direction reference to create a circular pattern. Therefore, display the temporary axes by choosing **View > Temporary Axes** from the menu bar.

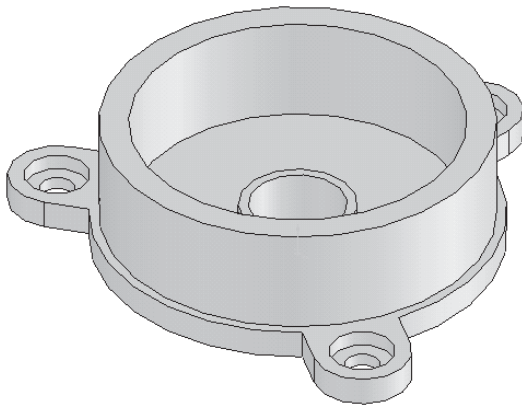
2. Select the temporary axis that passes through the center of the model and invoke the selection mode in the **Features to Pattern** area. The **Direction 1** callout is displayed.
3. Select the second and third features from the drawing area or from the **FeatureManager Design Tree** that is displayed in the drawing area.
4. Set the value of the **Number of Instances** spinner to **3** and make sure the **Equal spacing** check box is selected. Choose the **OK** button from the **Circular Pattern PropertyManager**.

5. Hide the temporary axes. The model, after patterning the features, is shown in Figure 7-79.

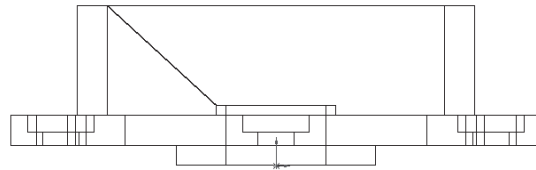
### Creating the Rib Feature

The next feature is a rib. The sketch for the rib feature will be created on the **Front Plane**.

1. Choose the **Rib** button from the **Features CommandManager** and select the **Front Plane** from the **FeatureManager Design Tree**.
2. Set the display model to wireframe, create the sketch for the rib feature, and add the required relations, as shown in Figure 7-80.
3. Exit the sketching environment and set the value of the **Rib Thickness** spinner to **2**.



**Figure 7-79** Model after patterning the features



**Figure 7-80** Sketch for the rib feature

Leave all the other default options as they are and choose the **OK** button from the **Rib PropertyManager**.

4. Change the model display mode to shaded.
5. Using the **Circular Pattern** tool, create six instances of the rib feature. The final model after, creating all the features, is shown in Figure 7-81.

### Saving the Model

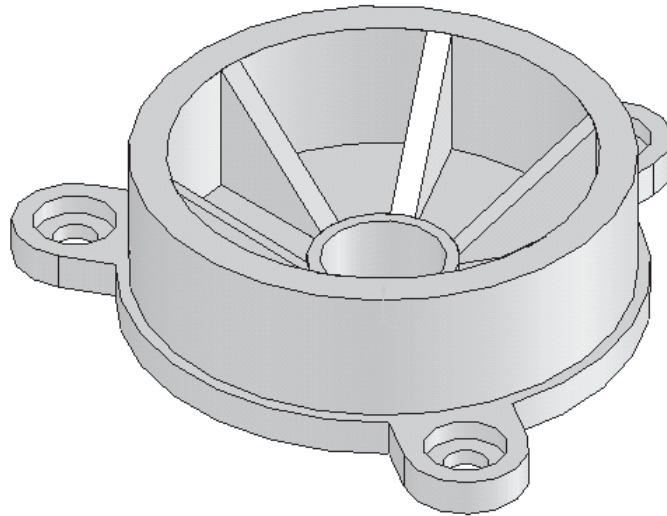
1. Choose the **Save** button from the **Standard** toolbar and save the model in the *c07* folder with the name given below.

*\My Documents\SolidWorks\c07\c07tut2.sldprt*

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

In this tutorial you will create the cylinder head of a two-stroke automobile engine shown in





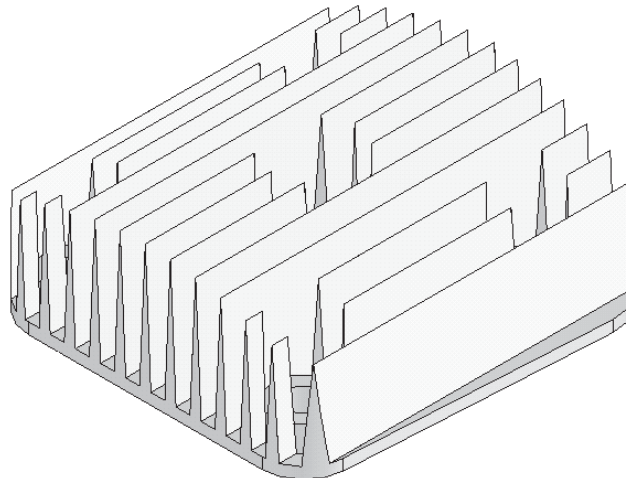
*Figure 7-81 Final solid model*

### Tutorial 3

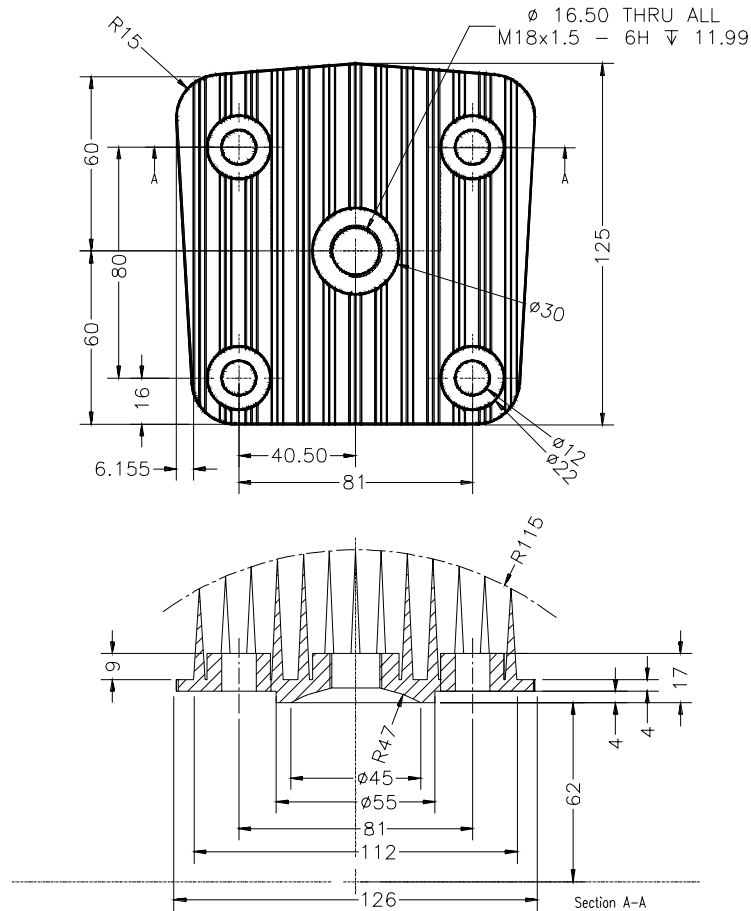
Figure 7-82. The dimensions of the model are shown in Figure 7-83. You will also create a section view of the model using the **Section View** tool.

**(Expected time: 1 hr)**

The steps to be followed to complete this tutorial are discussed next.



*Figure 7-82 Model for Tutorial 3*



**Figure 7-83** Top view and the section front view with dimensions

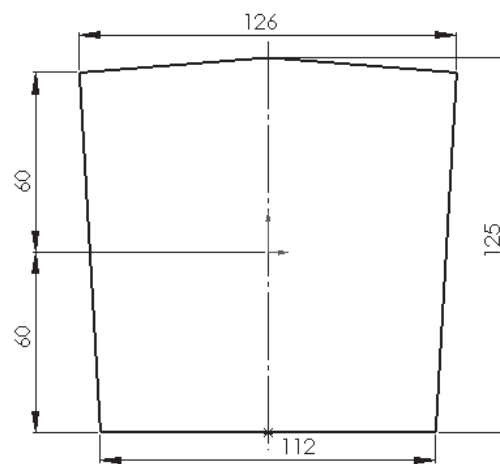
- Create the base feature of the model by extruding a polygon to the given depth, refer to Figure 7-84.
- Add the fillet to the base feature.
- Create a circular feature at the bottom face of the base feature.
- Create the revolve cut feature to create the dome of the cylinder head, refer to Figures 7-85 and 7-86.
- Create the left fin of the cylinder head by extruding the sketch. The sketch for this feature should be carefully dimensioned and defined, refer to Figure 7-87.
- Use the **Vary sketch** option to pattern the fins, refer to Figure 7-88.
- Create other cut and extrude features to complete the model, refer to Figure 7-89.
- Create a tap hole using the hole wizard, refer to Figure 7-90.
- Create the section view of the model, refer to Figure 7-91.

### Creating the Base Feature

1. Create a new SolidWorks part document.

The base feature of the model will be created by extruding the sketch created on the **Top** plane.

2. Invoke the **Extruded Boss/Base** tool and select the **Top Plane** as the sketching plane.
3. Create the sketch for the base feature and add the required relations and dimensions to the sketch as shown in Figure 7-84.



*Figure 7-84 Sketch for the base feature*

4. Exit the sketching environment and extrude the sketch to a depth of 4 mm.

### Creating the Second Feature

The second feature of the model is the fillet feature. You need to fillet all the vertical edges of the base feature using the given radius.

1. Invoke the **Fillet** tool and set the value of the **Radius** spinner to **15**. Select all the vertical edges of the base feature to add the fillet feature.
2. Choose the **OK** button from the **Fillet PropertyManager**.

### Creating the Third Feature

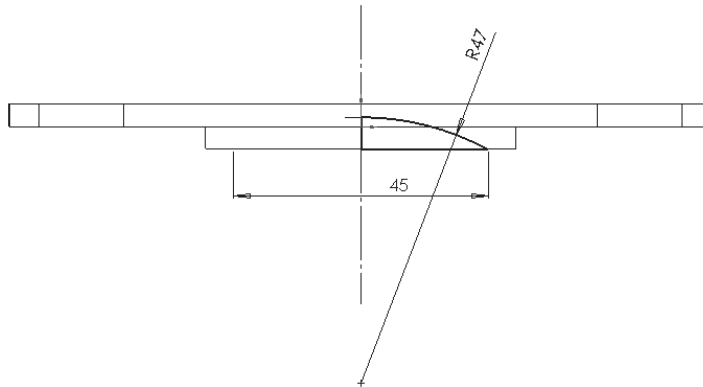
After creating the base and adding fillet at its vertical edges, you will create the third feature of the model, which is a circular extruded feature. The sketch of the feature will be drawn on the bottom face of the base feature and it will be extruded to the given depth.

1. Invoke the **Extruded Boss/Base** tool and select the bottom face of the base feature as the sketching plane.
2. Create a circle of 55 mm diameter with its center point at the origin.
3. Exit the sketching environment and extrude the sketch to a depth of 4 mm.

### Creating the Fourth Feature

The fourth feature is a revolved cut feature whose sketch will be drawn on the **Front Plane**. After drawing the sketch, apply the required relations and dimensions to it.

1. Invoke the **Revolved Cut** tool and select the **Front Plane** from the **FeatureManager Design Tree**.
2. Draw the sketch for the revolved cut feature and add the required relations and dimensions as shown in Figure 7-85. You need to apply the vertical relation between the center point of the arc and the origin to fully define it.



**Figure 7-85** Sketch for the revolve cut feature

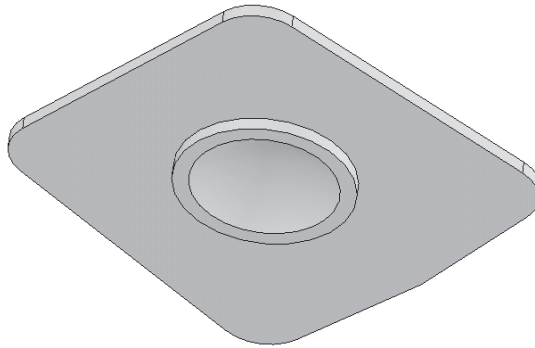


#### Note

When you draw a sketch for the revolved cut feature for this tutorial, draw a horizontal centerline, such that the start point of the centerline is merged with the upper endpoint of the arc. Also, ensure a tangent relation between the arc and the centerline exists to maintain the tangency of the arc.

3. Select the vertical centerline and exit the sketching environment. Make sure that the **Angle** spinner value is **360**.
4. Choose the **OK** button from the **Cut-Revolve PropertyManager**.

The rotated model, after creating the fourth feature, is shown in Figure 7-86.

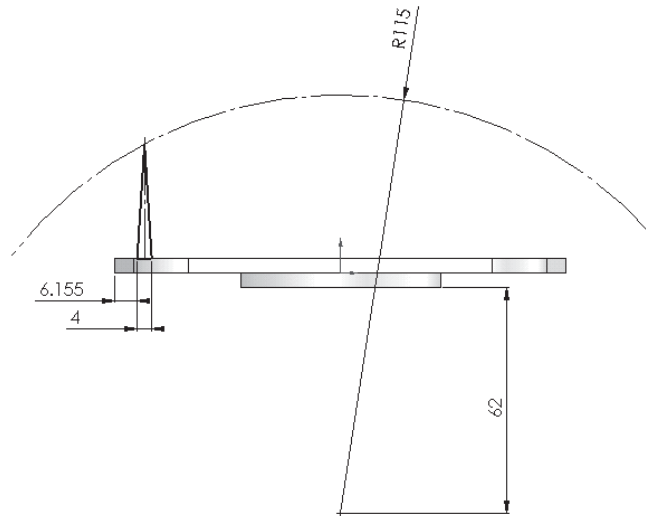


*Figure 7-86 Cut revolve feature added to the model*

### Creating the Fifth Feature

Next, you will create the left fin of the cylinder head. It will be created by extruding a sketch to both the directions using the **Through All** option. The sketch of this feature, drawn on the **Front Plane**, will be dimensioned and defined such that the length of the fin is driven by a construction arc and a horizontal dimension. The detailed step by step procedure of drawing, dimensioning, and defining the sketch is discussed next.

1. Invoke the **Extruded Boss/Base** tool and select the **Front Plane** from the **FeatureManager Design Tree**.
2. Using the **Line** tool draw the triangle and then draw a vertical centerline that passes through the upper vertex to the triangle, refer to Figure 7-87.
3. Invoke the **3 Pt Arc** tool and draw the arc as shown in Figure 7-87. Select the arc and select the **For Construction** check box from the **Options** rollout of the **Arc PropertyManager**.
4. Invoke the **Add Relations PropertyManager** and add the coincident relation between the upper vertex of the triangle and the centerline.
5. Add the midpoint relation between the lower endpoint of the centerline and the horizontal line of the triangle. Make sure that the **Coincident** relation between the upper vertex of the triangle and the centerline exists. Also, add the vertical relation to the centerline, if it is missing.
6. Add the coincident relation between the upper vertex of the triangle and the arc.
7. Add the required dimensions and relations to fully define the sketch, as shown in Figure 7-87.
8. Exit the sketching environment and extrude the sketch in both the directions using the **Through All** option. You will notice that the fin extends out of the base feature at both



**Figure 7-87** Sketch for the fin of the cylinder head



**Tip.** It is evident from Figure 7-87 that one of the horizontal dimensions has a value of 6.155. By default the primary unit precision is set to two decimal places. Therefore, for defining a dimension value with more number of decimal places, you need to select the dimension and set the precision value to the required decimal places from the **Primary Unit Precision** drop-down list from the **Dimension PropertyManager**.

the ends. You will learn how to remove the unwanted material of the fin later in this tutorial.

### Patterning the Fifth Feature

You will pattern the fin using the **Vary Sketch** option from the **Linear Pattern** tool. Using the **Vary sketch** option the geometry of each instance of the pattern varies according to the driven dimension and the relation added to the sketch of the feature to be patterned.

1. Choose the **Linear Pattern** button from the **Features CommandManager** to invoke the **Linear Pattern PropertyManager**.



You are prompted to select the directional reference.

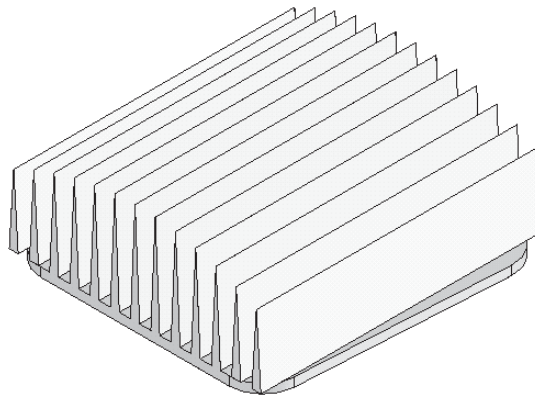
2. The fifth feature is selected by default in the **Features to Pattern** selection box. Select the fifth feature from the drawing area if it is not selected.
3. Select the horizontal dimension with the value of **6.155** as the directional reference in the **Pattern Direction** selection box of the **Direction 1** rollout.
4. Set the value of the **Spacing** spinner to **9**. Set the value of the **Number of Instances** spinner to **13**. Choose the **Reverse Direction** button.

5. Invoke the **Options** rollout and select the **Vary Sketch** check box from this rollout.

The preview of the pattern is not displayed in the drawing area.

6. Choose the **OK** button from the **Linear Pattern PropertyManager**.

The model, after adding the pattern feature, is shown in Figure 7-88.



**Figure 7-88** Model after patterning the fin of the cylinder head

### Creating the Cut Feature

The next feature that you will create is a cut feature. Using the **Rotate View** tool rotate the solid model and you will observe that the fins of the cylinder head that you patterned in the last feature extend beyond the boundary of the base feature. Therefore, to trim the extended portion of the fins, you will create a cut feature.

1. Invoke the **Extruded Cut** tool and select the top planar face of the base feature as the sketching plane.
2. Draw the sketch using the standard sketch tools. The sketch for this feature will be the outer profile of the base feature.

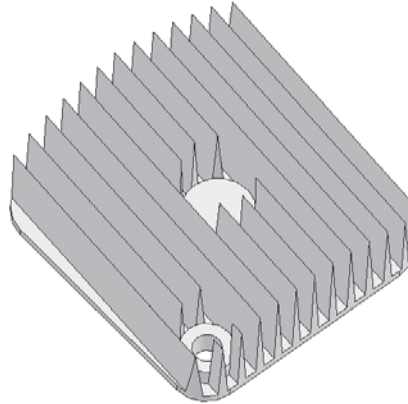


**Tip.** You can draw the sketch using the outer profile of the base feature using the **Convert Entities** tool. Select the lower flat face of the base feature and choose the **Convert Entities** button from the **Sketch** toolbar. You will notice that the sketch similar to the outer boundary of the base feature will be placed on the sketching plane.

3. Exit the sketching environment and choose the **Reverse Direction** button from the **Direction 1** rollout and select the **Through All** option from the **End Condition** drop-down list.

Since the direction of the side from which the material is to be removed is opposite to the required direction, therefore, you need to flip the direction of material removal.

4. Select the **Flip side to cut** check box from the **Direction 1** rollout and choose the **OK** button from the **Cut-Extrude PropertyManager**.
5. Using the standard modeling tools shape the model, as shown in Figure 7-89.



*Figure 7-89 Model after adding other extrude and cut features*

### Patterning the Remaining Features

After creating all the features, you need to pattern the cut, extrude, and hole features created at the lower left corner of the model.

1. Invoke the **Linear Pattern PropertyManager** and select the cut, extrude, and hole features, created on the lower left corner of the model.
2. Select the two directional references to pattern the features in both the directions and set the values of the distances between the instances and the number of instances, refer to Figure 7-83.



**Tip.** The sketch sharing option is also available in SolidWorks. This allows you to use a sketch used earlier in creating a sketch feature. For sharing the sketch, select it from the **FeatureManager Design Tree**, after invoking the feature tool. You need to expand the created sketched feature to select the sketch for sharing from the **FeatureManager Design Tree**.

3. Choose the **OK** button from the **Linear PropertyManager**.

### Creating a Tapped Hole

The last feature of the model is a hole feature. You will create a tapped hole using the **Hole Wizard** and then define the placement of the hole.

1. Select the top face of the middle circular extrude feature as the placement plane for the hole feature.

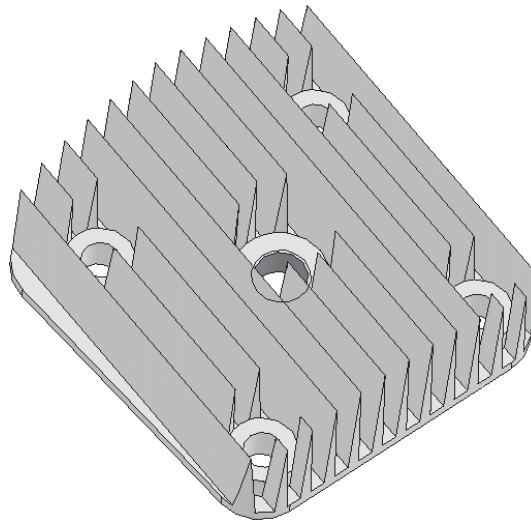


2. Invoke the **Hole Definition** dialog box by choosing the **Hole Wizard** button from the **Features CommandManager** and select the **Tap** tab. Select **ANSI Metric** from the **Standard** drop-down list.
3. Select the **M18x1.5** option from the **Size** drop-down list to define the size of the tap hole.
4. Select the **Through All** option from the **Tap Drill Type & Depth** drop-down list. Also, select the **Through All** option from the **Thread Type & Depth** drop-down list.
5. Select the **Add Cosmetic thread with thread callout** option from the **Add Cosmetic Thread** drop-down list. Choose the **Next** button from the **Hole Definition** dialog box. The **Hole Placement** dialog box is displayed.

The tapped hole will be placed by default on the placement plane. This default location is not the required position to place the hole. You need to define the placement of the tapped hole concentric with the center circular feature.

6. Invoke the **Add Relations PropertyManager** and add the **Concentric** relation between the center point of the tapped hole and the circular extruded feature of diameter 55 mm.
7. Choose the **Finish** button from the **Hole Placement** dialog box to end the tapped hole feature creation.

The rotated final model is shown in Figure 7-90.



*Figure 7-90 Final solid model*

### Displaying the Section View of the Model

Next, you will display the section view of the model. The section view of the model is created using the **Section View PropertyManager**.



**Tip.** You will observe that a graphic thread is displayed in the tapped hole. You can zoom the area for the better visualization of the graphic thread.

You will also observe that the cosmetic thread is displayed with the tapped hole feature. On orienting the model in the top view, you will observe the thread convention is visible. On orienting the model in the front, back, right, or left views, you will view the side convention of the thread.

You can also hide the cosmetic thread by selecting the cosmetic thread from the drawing area and right-clicking to invoke the shortcut menu. Choose the **Hide** option from the shortcut menu to hide the cosmetic thread.

1. Orient the model in the isometric view.
2. Choose the **Section View** button from the **View CommandManager**.



The **Front** view is selected as the section plane in the **Section View PropertyManager** by default. The preview of the section view, using the **Front Plane** as the section plane, is displayed in the drawing area.

3. Choose the **OK** button from the **Section View PropertyManager** to view the section of the model.

The section view of the model is shown in Figure 7-91.

4. Choose the **Section View** button again from the **View CommandManager** to return to the full view mode.

### Saving the Model

When you choose the **Save** button from the **Standard** toolbar, the **Save As** dialog box will be displayed because the document has not been saved until now. You can enter the name of the document in this dialog box.

1. Choose the **Save** button from the **Standard** toolbar and save the model in the *c07* folder with the name given below.

*\My Documents\SolidWorks\c07\c07tut3.sldprt*

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

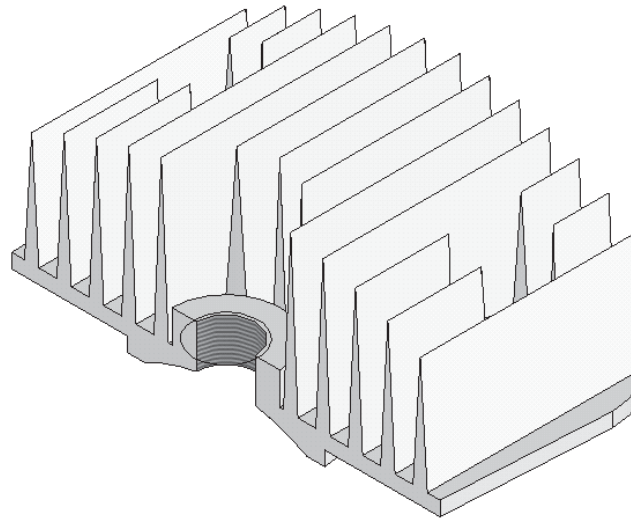


Figure 7-91 Section view of the model

## SELF-EVALUATION TEST

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

1. To invoke the **Mirror PropertyManager**, choose **View > Pattern/Mirror > Mirror** from the menu bar. (T/F)
2. If you modify the parent feature, then the same change will not reflect on the mirrored feature. (T/F)
3. You cannot preselect the mirror plane and the feature to be patterned before invoking the **Mirror** tool. (T/F)
4. You can mirror a single face using the **Mirror** tool. (T/F)
5. You can also pattern a pattern feature. (T/F)
6. The \_\_\_\_\_ **PropertyManager** is used to view the section view.
7. A \_\_\_\_\_ is provided in the drawing area to dynamically adjust the offset distance of the section plane.
8. \_\_\_\_\_ option is used to create a pattern by specifying the coordinates.
9. \_\_\_\_\_ option is used to create a pattern with respect to the sketched points.
10. Using the \_\_\_\_\_ rollout you can delete the pattern instances.

## REVIEW QUESTIONS

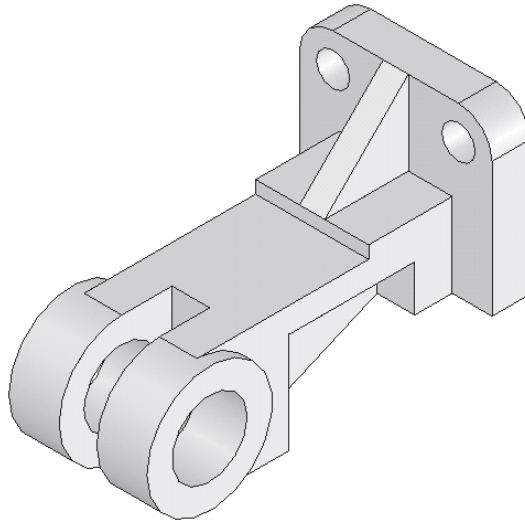
Answer the following questions.

1. The \_\_\_\_\_ check box is used to accommodate all the instances of a pattern along the selected curve.
2. Enter the coordinates for creating the instances in the \_\_\_\_\_ area of the **Table Driven Pattern** dialog box.
3. You need to invoke \_\_\_\_\_ to create a rib feature.
4. The \_\_\_\_\_ check box is used to transfer the visual properties assigned to the feature or parent body to the mirrored instance.
5. Using \_\_\_\_\_ check box from the **Section View PropertyManager**, you can create a section using an invisible plane normal to the eye view as the section plane.
6. When you choose the **Mirror** button from the **Features** tool, which **PropertyManager** is displayed?
  - (a) **Mirror Feature PropertyManager**
  - (b) **Mirror All PropertyManager**
  - (c) **Mirror PropertyManager**
  - (d) **Copy/Mirror PropertyManager**
7. Which option is used to mirror the exact geometry of the feature independent of the relationships between the geometries?
  - (a) **Same Mirror**
  - (b) **Geometry Pattern**
  - (c) **Geometry Copy**
  - (d) **Copy Geometry**
8. Which pattern is created along the sketched lines, arcs, or splines?
  - (a) Curve-driven pattern
  - (b) Sketch-driven pattern
  - (c) Geometry-driven pattern
  - (d) Linear pattern
9. Which dialog box is invoked to create a pattern by specifying the coordinate points?
  - (a) **Sketch Driven Pattern**
  - (b) **Table Driven Pattern**
  - (c) **Mirror**
  - (d) None of these
10. Which plane is selected by default when you invoke the **Section View PropertyManager** to view a section of the model?
  - (a) **Right**
  - (b) **Top**
  - (c) **Front**
  - (d) **Plane 1**

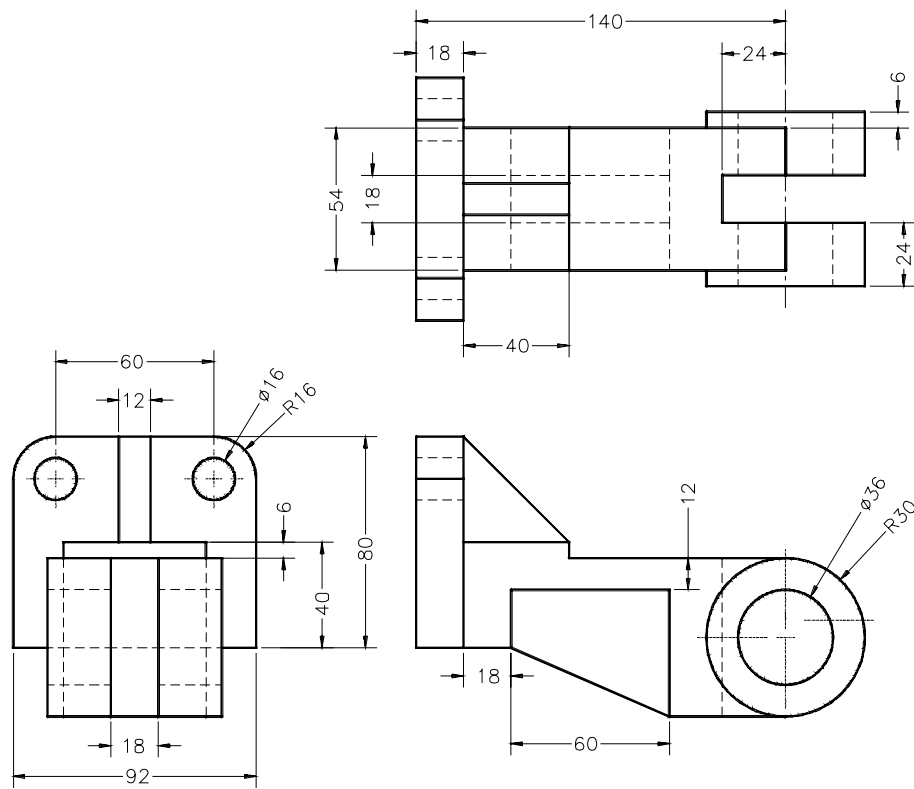
## EXERCISES

### Exercise 1

Create the model shown in Figure 7-92. The dimensions of the model are shown in Figure 7-93.  
(Expected time: 1 hr)



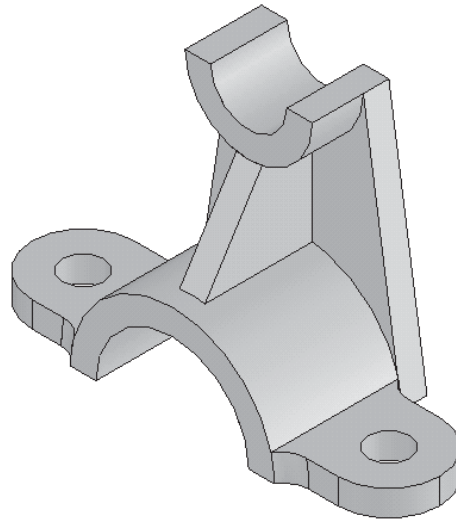
*Figure 7-92 Solid model for Exercise 1*



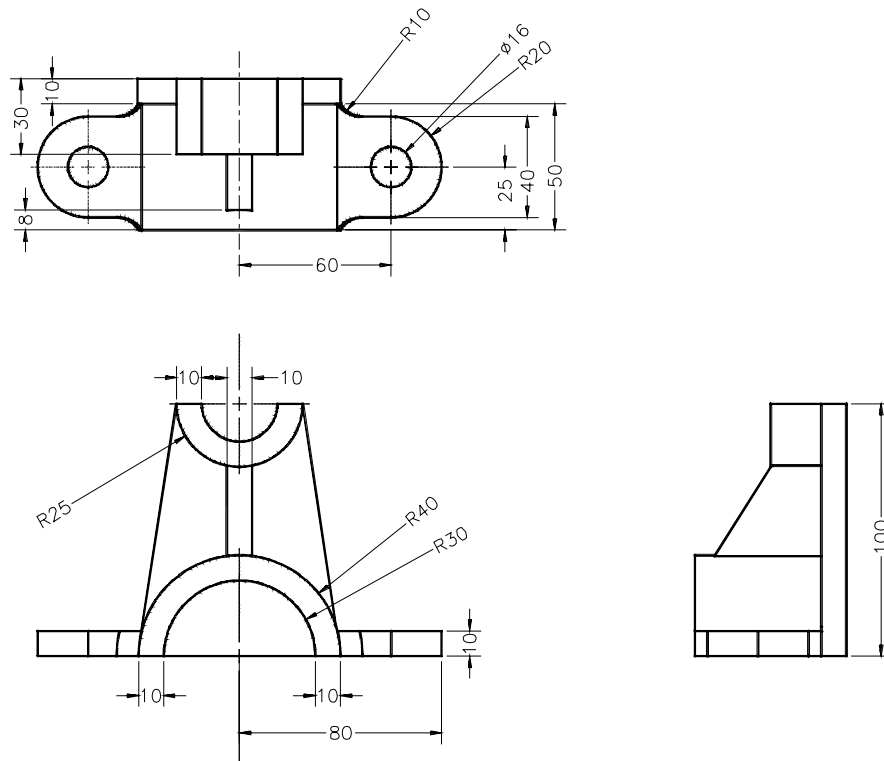
*Figure 7-93 Views and dimensions of the model for Exercise 1*

## Exercise 2

Create the model shown in Figure 7-94. The dimensions of the model are shown in Figure 7-95.  
(Expected time: 1 hr)



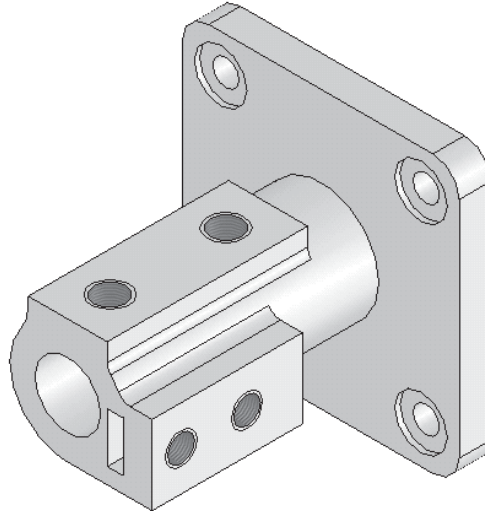
**Figure 7-94** Solid model for Exercise 2



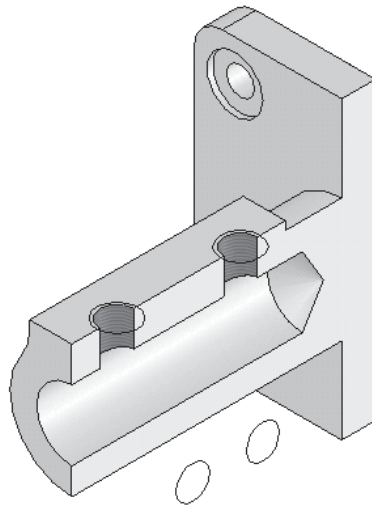
**Figure 7-95** Views and dimensions of the model for Exercise 2

### Exercise 3

Create the model shown in Figure 7-96. Next, create the section view of the model using the **Right Plane**. Figure 7-97 shows the section view of the model whose dimensions are shown in Figure 7-98. (Expected time: 45 min)



*Figure 7-96 Solid model for Exercise 3*



*Figure 7-97 Section view of the model*





**Answers to Self-Evaluation Test**

1. F, 2. F, 3. F, 4. F, 5. T, 6. Section View, 7. drag handle, 8. Table Driven Pattern, 9. Sketch Driven Pattern, 10. Instances to Skip