Chapter 6

Advanced Modeling Tools-I

Learning Objectives

After completing this chapter you will be able to:

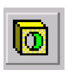
- *Create holes using the Simple Hole option.*
- *Create standard holes using Hole Wizard.*
- *Apply simple and advanced fillets.*
- *Understand various selection methods.*
- *Chamfer the edges and vertices of the model.*
- *Create the Shell feature.*

ADVANCED MODELING TOOLS

This chapter discusses various advanced modeling tools available in SolidWorks that assist you in creating a better and accurate design by capturing the design intent in the model. For example, in the previous chapters you learned how to create a hole using the cut option. But in this chapter you will create holes using the **Simple Hole** option and **Hole Wizard** option. Using the hole wizard, you can create the standard holes classified on the bases of the industrial standard, screw type, and size. Hole wizard of the SolidWorks is one of the largest standard industrial virtual hole generation machine available in any CAD package. You will also learn about some other advanced modeling tools such as fillet, chamfer, and shell in this chapter.

Creating Simple Hole

Toolbar:	Features > Simple Hole	*(Customize to Add)*
Menu:	Insert > Features > Hole > Simple	



The **Simple Hole** option is used to apply simple hole feature. In previous chapter you learned to create holes by sketching a circle and then using the cut option to complete hole creation. But using the simple hole option you do not need to create a sketch of the hole. This is the reason the holes created with this option act as a placed feature. To create a hole using this option, first you need to select the plane on which you want to place the hole feature. This is because this option is available only after you select the placement plane. Now, choose the **Simple Hole** button from the **Features** toolbar or choose **Insert** > **Features** > **Hole** > **Simple** from the menu bar. The **Hole PropertyManager** is displayed as shown in Figure 6-1. Confirmation corner is also displayed in the drawing area.

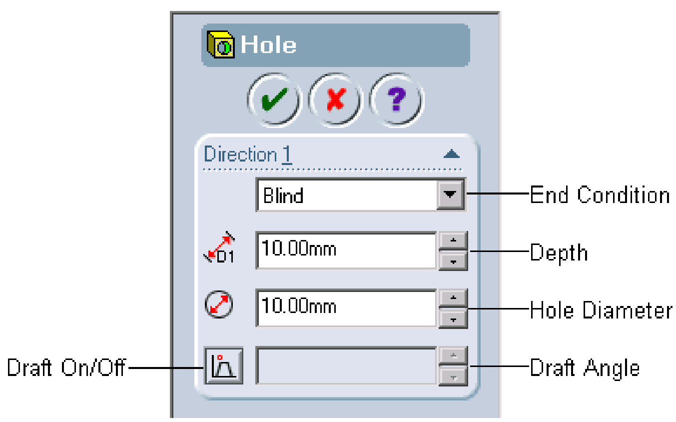


Figure 6-1** The **Hole PropertyManager

When you invoke the **Hole PropertyManager**, the preview of the hole feature is displayed in the drawing area in temporary graphics with the default values. The preview of the hole feature with default value is shown in Figure 6-2. Using the **End Condition** drop-down list, specify the termination type and set the value of the hole diameter in the **Hole Diameter** spinner. You can also specify a draft angle in the hole feature using the **Draft On/Off** button

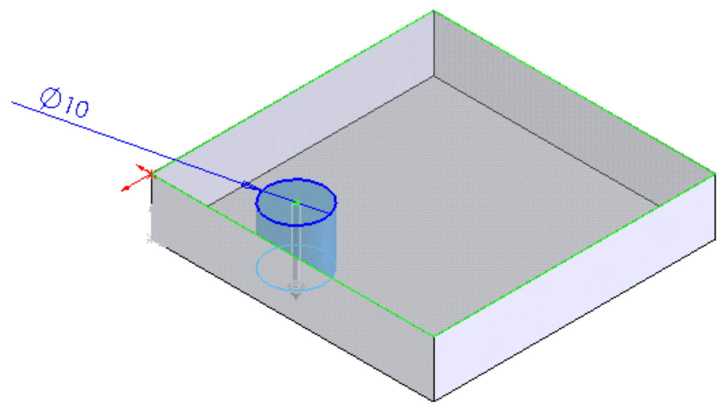
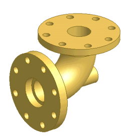


Figure 6-2 *Preview of the hole being created using the* ***Simple Hole*** *option*



Tip. *It is recommended that before creating the hole feature, you create a point that will define the placement of the hole feature. While creating the hole feature, you can dynamically move the hole feature by selecting the center point of the sketch and dragging the cursor to the point sketched earlier. The cursor will snap to that point and coincident relation will be appiled between the sketched point and the center point of the hole.*

and you can set the value of the draft angle using the **Draft Angle** spinner. The preview of the draft angle is also displayed in the drawing area in temporary graphics. After setting all the parameters choose the **OK** button from the **Hole PropertyManager** or choose the **OK** option from the confirmation corner.

The hole feature created using this option is placed on the selected plane but the placement of the hole is not yet defined. Therefore, select the hole feature from the **FeatureManager Design Tree** and right-click to display the shortcut menu. Choose the **Edit Sketch** option from the shortcut menu. The sketching environment is invoked and you can apply the relations and dimensions to define the placement of the hole feature on the selected face and exit the sketching environment.

Creating the Standard Holes Using Hole Wizard

Toolbar:	Features > Hole Wizard
Menu:	Insert > Features > Hole > Wizard

The **Hole Wizard** option is used to add standard holes to the model. The holes applied using the hole wizard include the standard counterbore, countersink, drilled, tapped, and pipe tap holes. You can also create a user-defined counterbored drilled hole, counter-drilled drilled hole, counterbored hole, counterdrilled hole, countersunk hole, countersunk drilled hole, simple hole, simple drilled hole, tapered hole, and tapered drilled hole. You can control all the parameters of the holes including the termination options and you can also modify the holes according to your requirement after placing them. Thus, it results in the placement of standard parametric holes using this option. There are two methods to place a hole feature using the hole wizard. The first method is the preselection method. In this method you have to select a placement face or a placement plane before invoking this tool. The placement face can be a planar face or a curved face. Then choose the **Hole Wizard** button from the **Features** toolbar or choose **Insert > Features > Hole > Wizard** from the menu bar to invoke the **Hole Definition** dialog box. The preview of the hole feature is displayed in the graphics area. If the preview is not visible properly then move the **Hole Definition** dialog box to see the preview. If you change the settings and parameters or the type of hole, the preview of the hole will also get modified dynamically. The various options available in the **Hole Definition** dialog box are discussed next.

Counterbore Tab

The **Counterbore** tab, as shown in Figure 6-3, available in the **Hole Definition** dialog box is used to define the parameters of a counterbore hole. You can also add the frequently used counterbore holes in the favorites list options in this tab. The various options available in the **Counterbore** tab are discussed next.

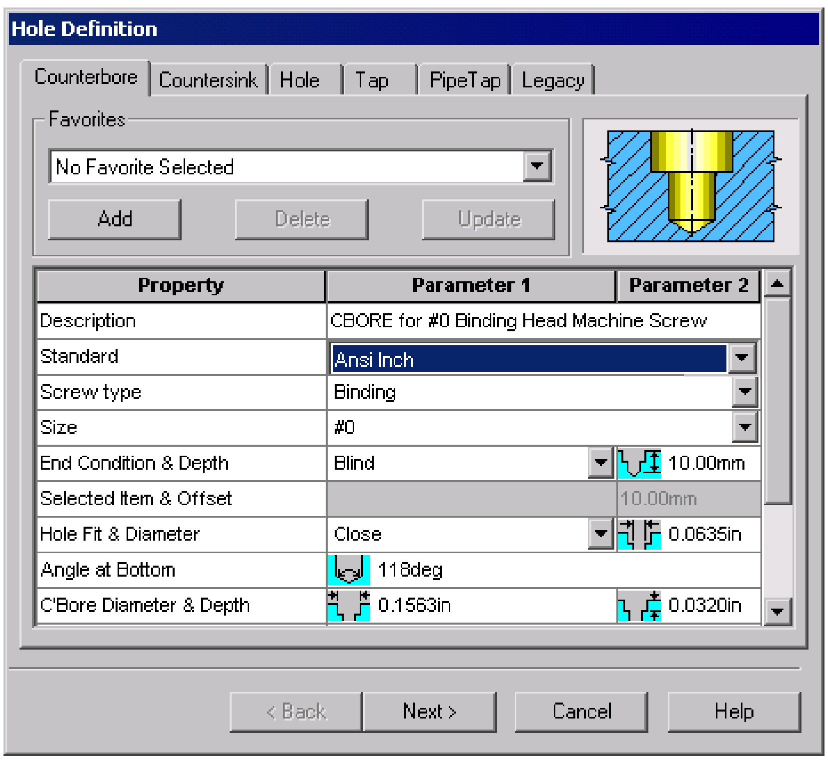


*Figure 6-3 The **Counterbore** tab of the **Hole Definition** dialog box*

Property

The **Property** column of the **Hole Definition** dialog box is used to define the various properties of the standard hole. The properties include the description, standard, screw type, size, and so on. The parameters available for all the properties are divided in two columns. The name of the columns are **Parameter 1** and **Parameter 2**. These two columns are used to define the values and other related information to create the hole. The various options available in the **Property** column are discussed next.

Description

The **Description** option is used to display the name and the description of the standard hole.

Standard

The **Standard** option is used to specify the industrial dimensioning and hole standard. You can select the standard from the **Standard** drop-down list available in the **Parameter 1** column. By default, the **Ansi Inch** standard is selected. Various other dimensioning standards are available in this drop-down list such as **Ansi Metric**, **BSI**, **DIN**, **ISO**, **JIS**, **DME Mould Bases**, **Hasco Metric Mould Bases**, **PCS Mould Bases**, **Progressive Mould Bases**, and **Superior Mould Bases**.

Screw type

The **Screw type** option is used to define the type of fastener to be inserted in the hole. The standard holes created using the **Hole Wizard** depend on the fastener to be inserted in that hole and the size of the fastener. Therefore, creation of a hole based on the fastener to be used in that hole is a good practice and is the unique feature of SolidWorks. You can select the screw type from the **Screw Type** drop-down list available in the **Parameter 1** column. The types of screws available in the drop-down list depend on the standard selected from the **Standard** drop-down list.

Size

The **Size** option is used to define the size of the fastener that will be inserted in the hole that is created using the **Hole Wizard**. The size of the fastener is selected from the **Size** drop-down list available in the **Parameter 1** column. The sizes of the fasteners available in the **Size** drop-down list depend on the standard selected from the **Standard** drop-down list.

End Condition and Depth

The **End Condition and Depth** option is used to define the end condition of feature termination. To define the end condition for feature termination, select the end condition option from the **End Condition** drop-down list available in the **Parameter 1** area. By default, the **Blind** option is selected in this drop-down list. Therefore, if the end condition is **Blind** you have to specify the depth of feature termination. This is the reason the **Depth** area available on the right of the **End Condition** drop-down list in the **Parameter 2** column is replaced by the edit box when clicked once. You can specify the depth of feature termination in this edit box.

Note

A preview area is provided on the top right of the ***Hole Definition*** *area. You can observe the preview of the hole being created. The preview of the hole feature is changed dynamically as you change the parameters of the hole feature.*

Selected Item and Offset

The **Selected Item and Offset** option is used to define the surface for feature termination. The **Selected Item** display area in the **Parameter 1** column is active only when the **Up To Vertex**, **Up To Surface**, or **Offset From Surface** option is selected from the **End Condition** drop-down list. The **Offset** area available on the right of the **Select Item** display area is replaced by the **Offset** edit box and you can enter the value of offset in this edit box. This option is available only when you select the **Offset From Surface** option from the **End Condition** drop-down list.

Hole Fit and Diameter

The **Hole Fit and Diameter** option is used to specify the type of **Fit** to be applied in the hole to be created. The types of fits are available in the **Hole Fit** drop-down list in the **Parameter 1** column. The types of fits available in the **Hole Fit** drop-down list are **Close**, **Normal**, and **Loose**. The **Diameter** area on the right of the **Hole Fit** drop-down list in the **Parameter 2** area is replaced by the **Diameter** edit box by clicking once in this area. The value of the diameter in this area is the default value according to the standard. The value changes depending upon the type of hole fit selected from the **Hole Fit** drop-down list. You can also modify this value according to the requirement.

Angle at Bottom

The **Angle at Bottom** option is used to specify the angle at the bottom of the hole feature. The **Angle at Bottom** area available in the **Parameter 1** column is replaced by the **Angle at Bottom** edit box by clicking once. You can modify the default value for the angle at bottom by entering a new value in this edit box.

C'Bore Diameter and Depth

The **C'Bore Diameter and Depth** option is used to specify the diameter of the counterbore and the depth of the counterbore. The **C'Bore Diameter** area is available in the **Parameter 1** column and is replaced by the **C'Bore Diameter** edit box by clicking once to modify the default value. The **Depth** area available on the right of the **C'Bore Diameter** area is replaced by the **Depth** edit box by clicking once and you can modify the default depth of the counterbore.

Head Clearance

The **Head Clearance** option available in the **Property** column is used to specify the clearance distance between the head of the fastener and the placement plane of the hole feature. The **Head Clearance** area available in the **Parameter 1** column is replaced by the **Head Clearance** edit box by clicking once and you can modify the default head clearance value.

Near Side C'Sink Dia. and Angle

The **Near Side C'Sink Dia. and Angle** option is used to specify the diameter and the angle for the countersink on the upper face, which is the placement plane of the hole feature. The diameter is specified in the **Near Side C'Sink Dia.** area of the **Parameter 1** column and the angle is specified in the **Angle** area of the **Parameter 2** column.

Under Hd C'Sink Dia. and Angle

The **Under Hd C'Sink Dia. and Angle** option is used to specify the diameter and the angle for the countersink to be applied at the end of the counterbore head. The diameter is specified in the **Under Hd C'Sink Dia.** area in the **Parameter 1** column and the angle is specified in the **Angle** area in the **Parameter 2** column.

Far Side C'Sink Dia. and Angle

The **Far Side C'Sink Dia. and Angle** option is used to specify the diameter and the angle for the countersink to be applied at the end of the hole feature. This option is available only if the **Through All** or the **Up To Next** option is selected from the **End Condition** drop-down list. The diameter is specified in the **Far Side C'Sink Dia.** area in the **Parameter 1** column and the angle is specified in the **Angle** area in the **Parameter 2** column.

Favorites

The **Favorites** area available in the **Hole Definition** dialog box is used to add the frequently used holes to the favorite list. If you add a hole to the favorite list, you will not have to perform the same settings to add similar types of holes every time. To add a particular hole to the favorite list, choose the **Add** button available in the **Favorite** area of the **Hole Definition** dialog box. The **New Favorite Name** dialog box is displayed with the default name of the hole in the **Favorite Name** edit box as shown in Figure 6-4.

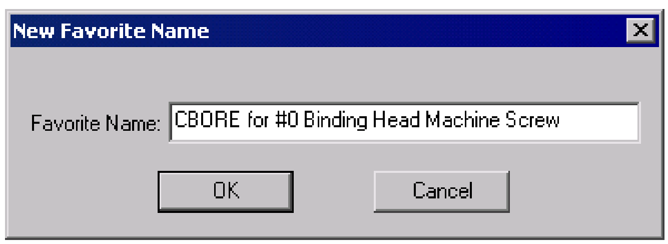


Figure 6-4 *The* ***New Favorite Name*** *dialog box*

You can enter a new name in the **Favorite Name** edit box and choose the **OK** button to add the hole to the favorite list. If you have to apply a hole feature from the favorite list, you just need to select the name of the hole feature from the **Favorites** drop-down list. The current parameters of the hole will be changed to the parameters of the hole feature that is selected from the **Favorites** drop-down list. Figure 6-5 shows the counterbore holes created using the **Hole Wizard**.

Countersink Tab

The **Countersink** tab, as shown in Figure 6-6, available in the **Hole Definition** dialog box is used to define the parameters of the countersink hole. Most of the options available in the **Property** column are the same as those in the **Property** column of the **Counterbore** tab. However, there are some additional options to define the angle and diameter of the countersink. These options of the **Property** column of the **Countersink** tab are discussed next.

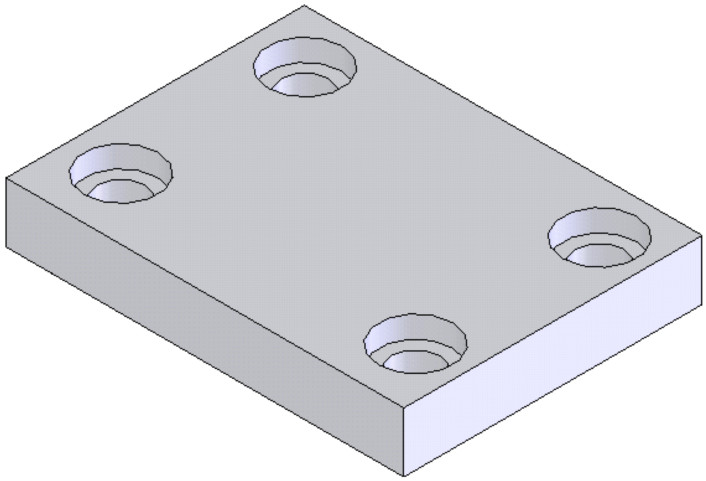


Figure 6-5 A model with counterbore holes

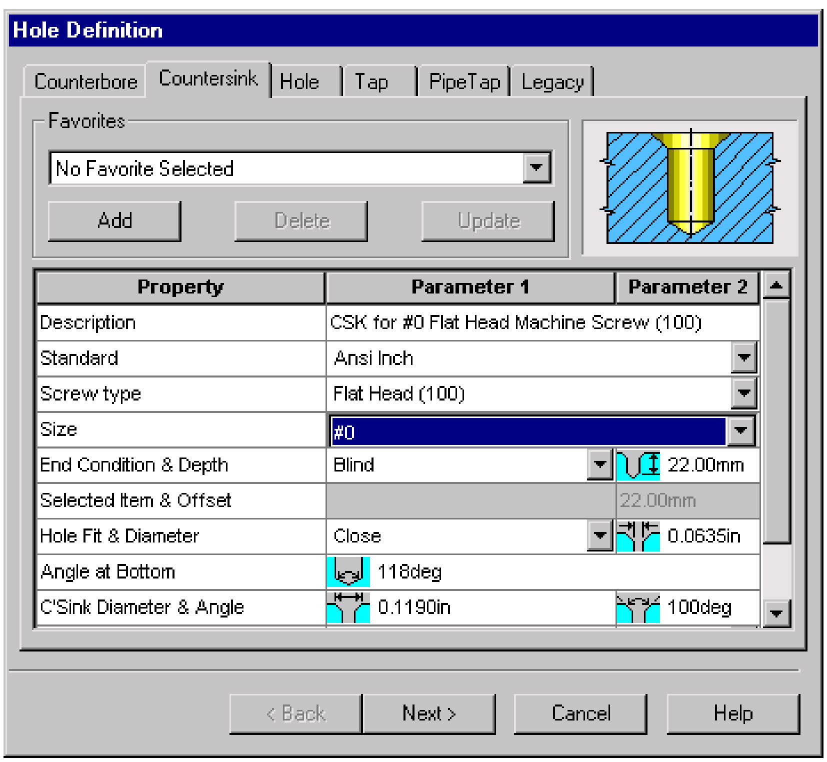


Figure 6-6 The ***Countersink*** *tab of the* ***Hole Definition*** *dialog box*

Property

The **Property** column of the **Hole Definition** dialog box is used to define the parameters of the countersink hole. As mentioned earlier, most of the options are the same as discussed in the **Property** column of the **Counterbore** tab. The options that have been discussed earlier are not included here. Some options that are not used to create only a counterbore and are not used in countersink are not available in this tab. Only the additional options available in the **Property** column of the **Countersink** tab are discussed next.

C'Sink Diameter and Angle

The **C'Sink Diameter and Angle** option is used to define the diameter and the angle of the countersink. The **C'Sink Diameter** area in the **Parameter 1** column is changed to the **C'Sink Diameter** edit box when clicked once. The value of the countersink diameter is specified in this edit box. The **Angle** area available in the **Parameter 2** area is replaced by the **Angle** edit box when clicked once. The value of the angle of countersink is specified in this edit box.

Head Clearance and Type

The **Head Clearance and Type** option is used to define the head clearance distance between the placement surface of the hole feature and the top face of the fastener and to define the type of the counter to be added. The **Head Clearance** area available in the **Parameter 1** column is replaced by the **Head Clearance** edit box when clicked once. The value of the clearance distance is specified in this edit box. The **Type** drop-down list available on the right of the **Head Clearance** area is used to specify if the clearance distance is compensated by extending the countersink or by adding a counterbore.

Figure 6-7 shows the countersink holes created using the **Hole Wizard**.

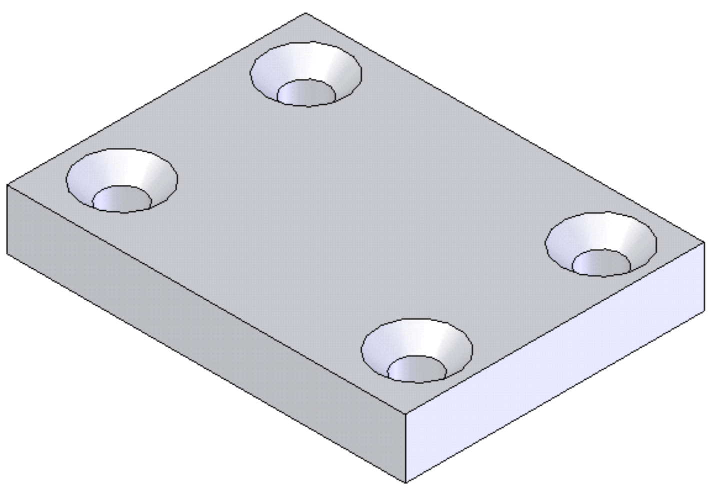


Figure 6-7 *A model with countersink holes*

Hole Tab

The **Hole** tab, shown in Figure 6-8, available in the **Hole Definition** dialog box is used to define the parameters to add a standard drilled hole to the model. Using this tab you can add simple holes that can be drilled using the standard drills available in the market. The options available in the **Property** column of this tab are used to specify the parameters for creating a standard drilled hole.

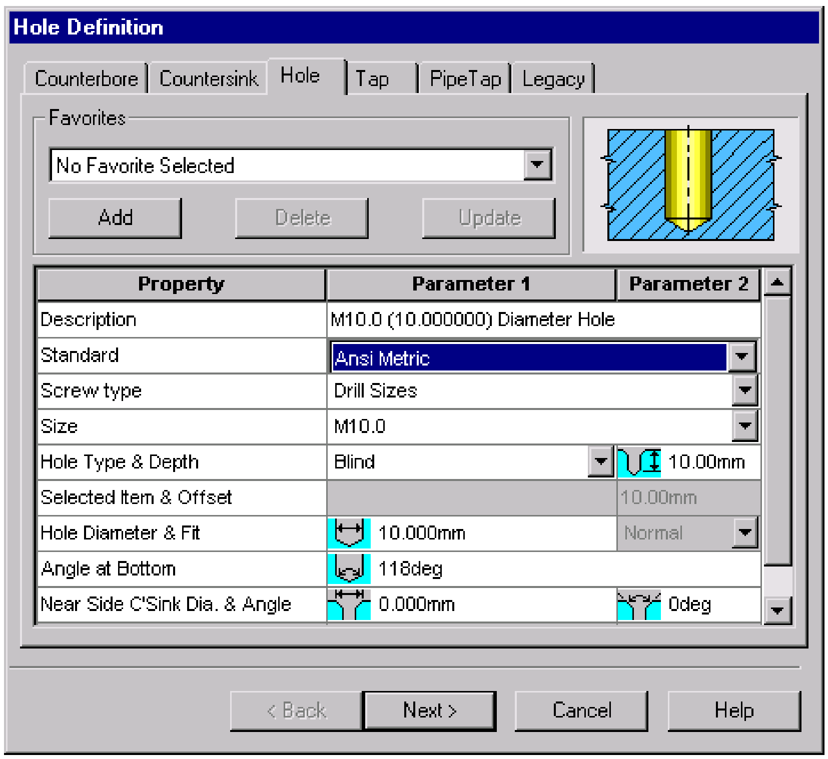


***Figure 6-8** The **Hole** tab of the **Hole Definition** dialog box*

Figure 6-9 shows the drilled holes created using the **Hole Wizard**.

Tap Tab

The **Tap** tab, shown in Figure 6-10, available in the **Hole Definition** dialog box is used to define the parameters to add a tapped hole to the model. Tap hole is basically a hole in which internal threading (also known as tapping) is shown. Therefore, using this tab you can add the standard tapped holes in your design. Almost all the options available in the **Property** column of this tab are same as discussed earlier. The additional options are discussed next.

Thread Type and Depth

The **Thread Type and Depth** option is used to define the type of depth and the value of the depth of the thread. The **Thread Type** drop-down list available in the **Parameter 2** column is used to specify the type of thread depth. The options available in the **Thread Type** drop-down list depend on the feature termination option selected from the **Tap**

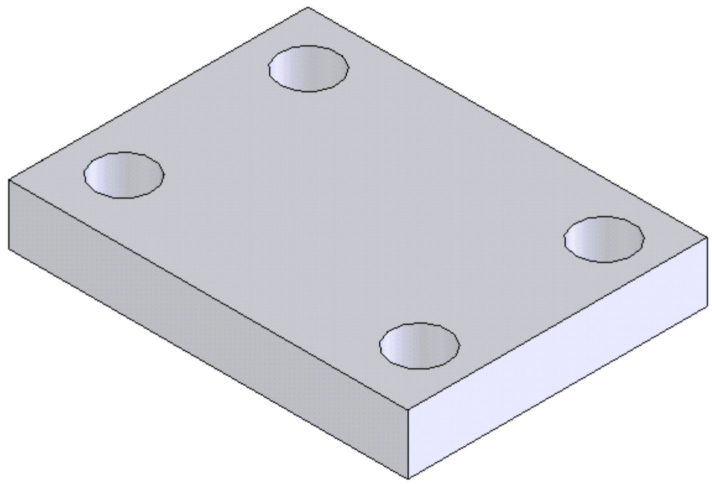


Figure 6-9 *A model with drilled holes*

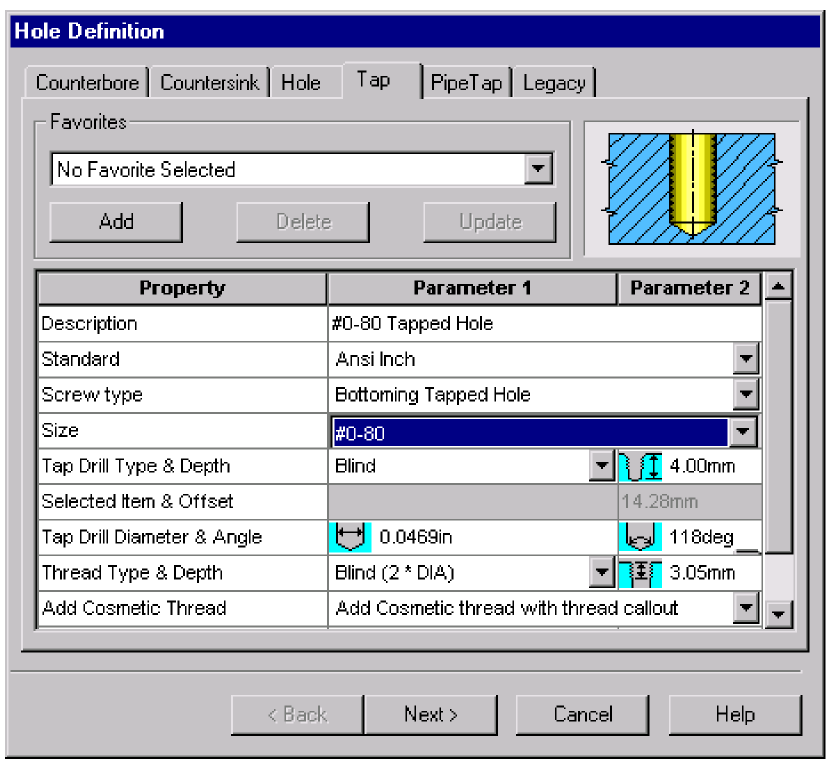


Figure 6-10 *The **Tap** tab of the **Hole Definition** dialog box*

Drill Type drop-down list. The depth of the thread is specified in the **Depth** area in the **Parameter 2** column.

Add Cosmetic Thread

The **Add Cosmetic Thread** option is used to add the cosmetic threads while creating a tapped hole. The **Add Cosmetic Thread** drop-down list is available in the **Parameter 1** column. If you select the **No Cosmetic thread** option from this drop-down list, the cosmetic thread will not be added to the tapped hole. If you select the **Add Cosmetic thread with thread callout** option from this drop-down list, the cosmetic thread and the thread callout will be added to the tapped hole feature. The **Add Cosmetic Thread without thread callout** option is used to add only the cosmetic thread to the hole feature.

> **Tip**. *In the modern modeling practice, the creation of threads is avoided in models because it results in the creation of complex geometry. When you generate the views from that model, the views generated contain the complex geometry, which is difficult to understand. Therefore, it is a better practice to avoid the creation of threads in the model and add the cosmetic threads. Using the cosmetic threads you will get the thread convention in the drawing views, which is recommended, instead of creating complete thread.*

Figure 6-11 shows the tapped holes created using the **Hole Wizard**.

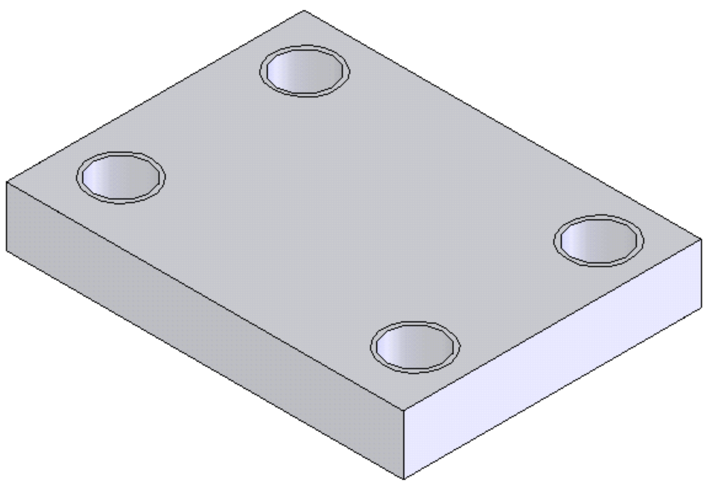


Figure 6-11 *Model with tapped holes*

Pipe Tap Tab

The **Pipe Tap** tab, shown in Figure 6-12, is used to specify the parameters for the tapered pipe tap hole. The options available in this tab are the same as discussed in earlier tabs.

Legacy Tab

The **Legacy** tab of the **Hole Definition** dialog box, shown in Figure 6-13, is used to add a user-defined hole feature. The options available in this tab are discussed next.

Hole type

The **Hole type** drop-down list is used to specify the type of hole you want to create. The types of holes available in this drop-down list are **C-Bored Drilled**, **C-Drilled Drilled**,

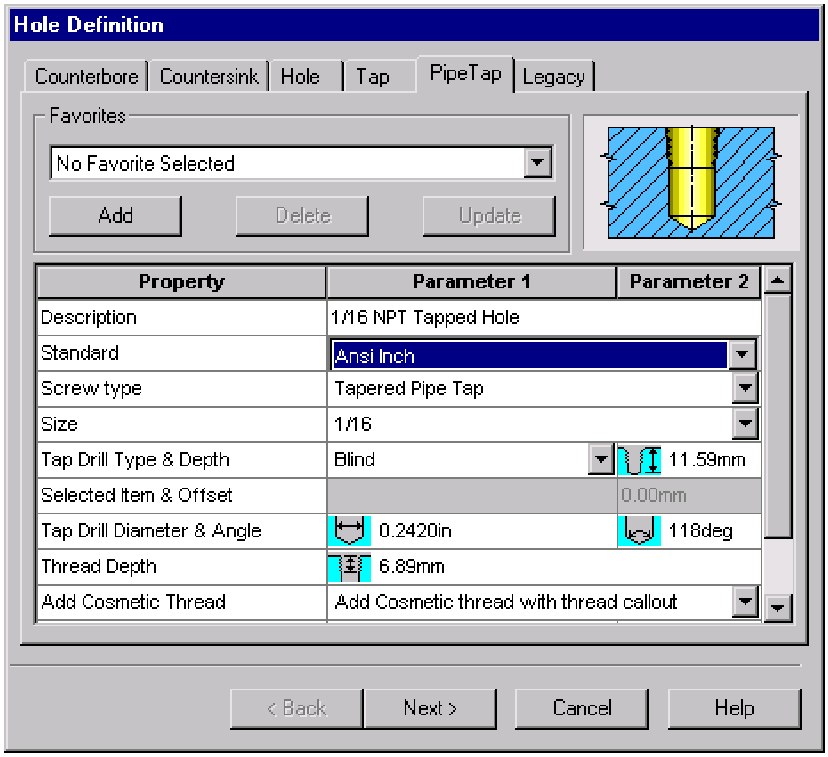


Figure 6-12 *The **Pipe Tap** tab of the **Hole Definition** dialog box*

Counterbored, **Counterdrilled**, **Countersunk**, **C-Sunk Drilled**, **Simple, Drilled**, **Tapered**, and **Tapered Drilled**. The preview of the selected hole type is displayed in the preview area provided at the right of the **Hole Definition** dialog box.

Section dimensions

The **Section dimensions** area is used to display the various dimensions of the selected hole and their default values. You can also modify the values by replacing the **Value** area by the **Value** edit box.

End condition

The **Type** drop-down list available in the **End condition** area is used to specify the end condition of feature termination.

As discussed earlier, the preview of the hole feature is dynamically updated in the drawing area while you are setting the parameters of the hole feature. This is because you had already selected the placement plane for the placement of the hole feature. If you do not select a placement plane for the placement of the hole feature, the preview of the hole feature is not displayed in the drawing area. After setting all the parameters of the hole feature, chooses the **Next** button. The **Hole Placement** dialog box is displayed as shown in Figure 6-14.

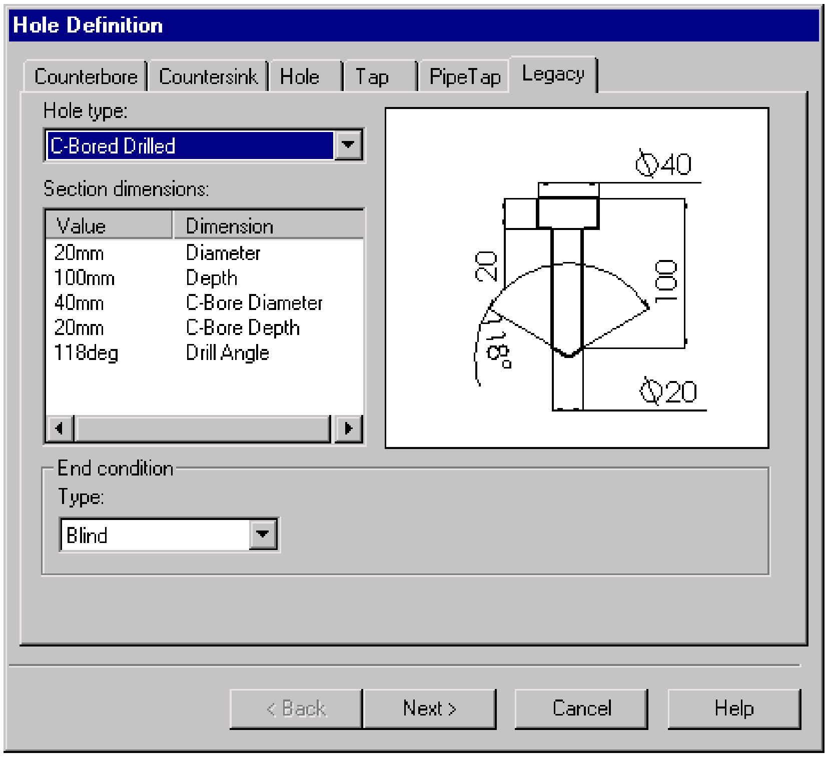


*Figure 6-13 The **Legacy** tab of the **Hole Definition** dialog box*

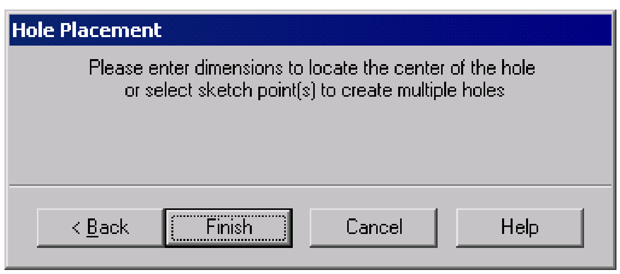


*Figure 6-14 The **Hole Placement** dialog box*

The select cursor is replaced by the placement cursor. Using the placement cursor you can place more holes in the current hole features. As discussed earlier, if the placement plane is selected earlier, the hole is already placed on the selected placement plane. If the placement plane is not selected earlier then you can specify a point to place the hole feature. You can also constrain the placement of the placement point of the hole feature using the relations and dimensions. Choose the **Finish** button to complete feature creation. Figure 6-15 shows a base plate with holes created using the hole wizard.

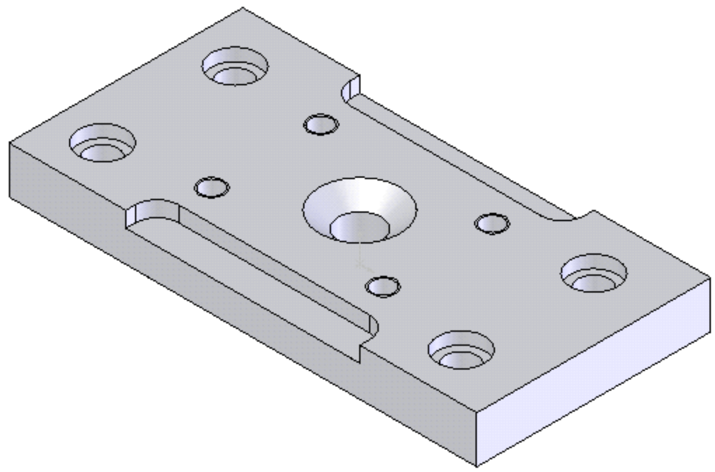
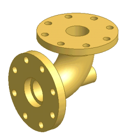


Figure 6-15 *Base Plate with holes created using the **Hole Wizard** option*



Tip. *The hole feature created using the hole wizard consists of two sketches. The first sketch is the sketch of the placement point and the second sketch is the sketch of the profile of hole feature. If you preselect the placement plane before invoking the **Hole Definition** dialog box, the resulting placement sketch will be a 2D sketch. Instead of preselecting the placement plane if you select the placement point after invoking the **Placement Point** dialog box, the resulting placement sketch is a 3D sketch. You will learn more about 3D sketches in the later chapters.*

*If a cosmetic thread is added in a tapped hole, the cosmetic thread is also displayed along with the placement and hole profile sketches. You can edit the cosmetic threads by selecting it from the **FeatureManager Design Tree** and right-clicking to display the shortcut menu. Choose the **Edit Definition** option to display the **Cosmetic Thread** dialog box. The **Cosmetic Thread** dialog box and the cosmetic threads are discussed in later chapters.*

You can also view the convention of the thread if the cosmetic thread is added to a tapped hole feature. Orient the model to the top view to observe the thread convention from the top view. Orient the model in the front view, back view, or any side view to observe the thread convention from the side views.

Creating Fillets

Toolbar:	Features > Fillet
Menu:	Insert > Features > Fillet/Round

In SolidWorks, you can add fillets as a feature in the model using the **Fillet** tool. As discussed earlier you can also add fillets within the sketch. But adding the fillets in a sketch is not a good practice according to the design point of view. This is because you have to keep the sketch as simple as possible. Using the fillet tool you can round an internal or external face or edge of a model. You can also use the advanced fillet options to add advanced fillets to the model. You can preselect the face, edge, or feature on which the fillet has to be applied. You can also select the entity to be filleted after invoking the fillet tool. Choose the **Fillet** button from the **Features** toolbar or choose **Insert** > **Features** > **Fillet/Round** from the menu bar to invoke the **Fillet PropertyManager**. The **Fillet PropertyManager** is shown in Figure 6-16. The preview of the fillet feature is also displayed

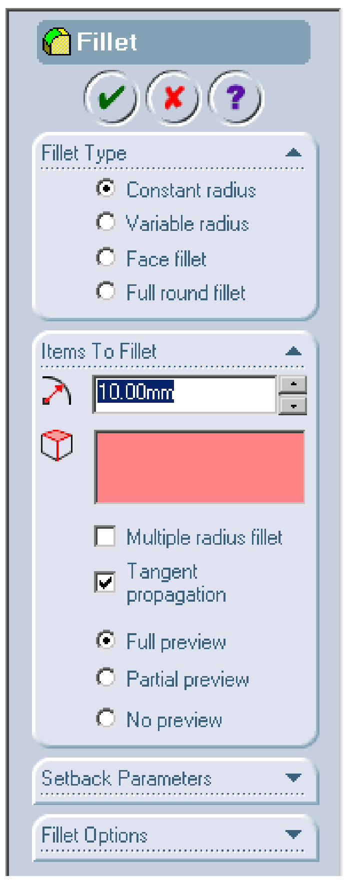


*Figure 6-16 The **Fillet PropertyManager***

in the drawing area if the entities to be filleted are selected. If preselection is not done, you are prompted to select the edges, faces, features, or loops to add the fillet feature. Using the select cursor, select the entity to be filleted. A fillet callout is also displayed along the preview of the fillet. Figure 6-17 shows the preview of the fillet feature with fillet callout.

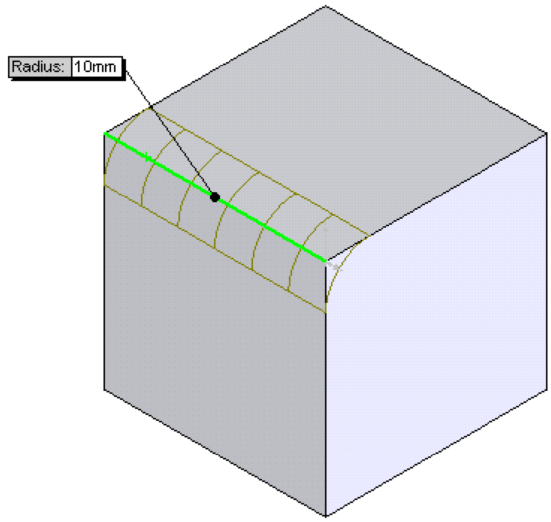


Figure 6-17 *Preview of the fillet feature*

Using the fillet tool you can create various types of fillets. Types of fillets that can be created using the fillet tool are given below.

1. Constant radius fillet
2. Variable radius fillet
3. Face Fillet
4. Full round fillet

All the above mentioned fillets are discussed next.

Constant Radius Fillet

The constant radius fillet option creates a fillet of constant radius along the selected entity. This is the default option selected in the **Fillet Type** rollout. You can set the value of the fillet radius using the **Fillet** spinner provided in the **Items To Fillet** rollout or click the value area of the fillet callout. The value area of the callout will be changed to the radius edit box. Enter the value of the radius and choose the ENTER key from the keyboard. The preview of the fillet will be changed dynamically when the value of the radius of the fillet is changed. The names of the selected entities are displayed in the **Edges**, **Faces**, **Features**, and **Loops** display area. The entities that you can select to add the fillet feature are faces, edges, features, and loops. Now, choose the **OK** button from the **Fillet PropertyManager**. Figures 6-18 through 6-23 show the selection of different entities and the resultant fillet creation from the selected entities.

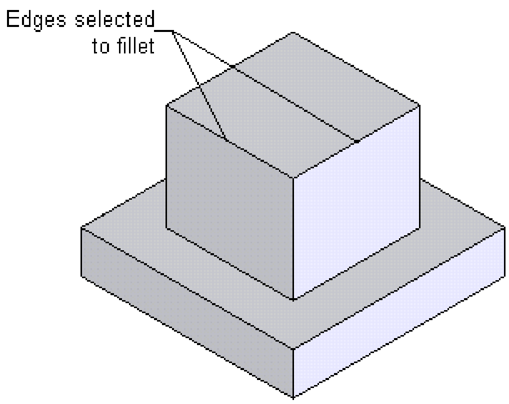


Figure 6-18 *Selecting the edges*

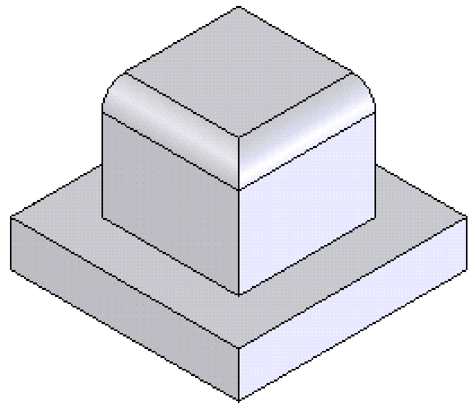


Figure 6-19 *Resultant fillet*

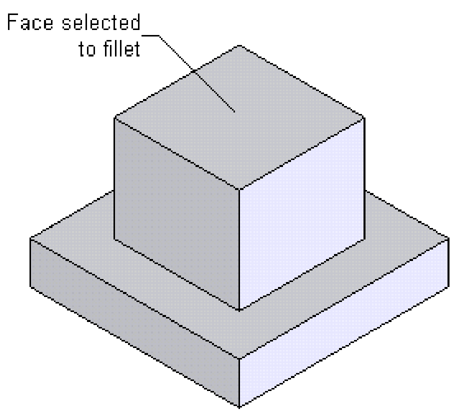


Figure 6-20 *Selecting the face*

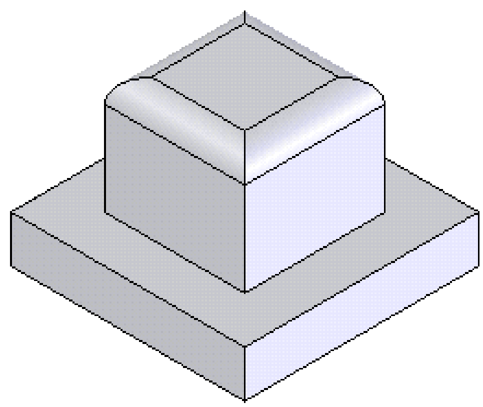


Figure 6-21 *Resultant fillet*

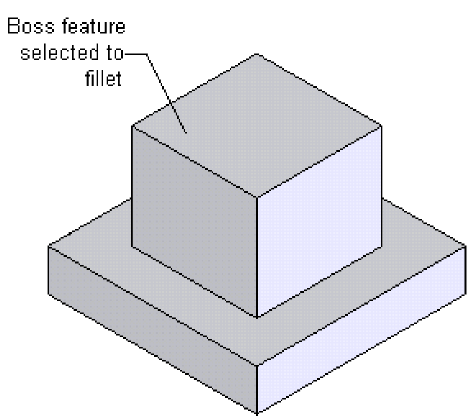


Figure 6-22 *Selecting the feature*

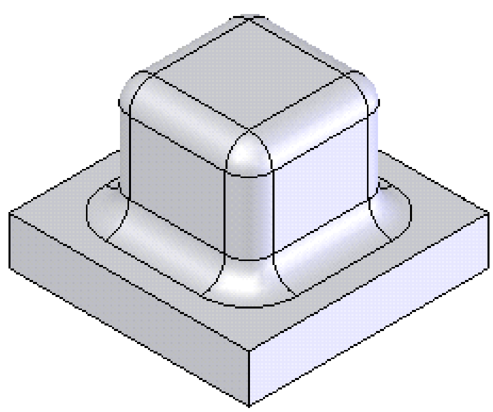
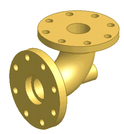


Figure 6-23 *Resultant fillet*



Tip. *You can preselect a feature for adding a fillet feature on that feature or select the feature after invoking the fillet tool. For postselection of the feature to fillet, invoke the* ***FeatureManager Design Tree*** *flyout and select the feature from the* ***FeatureManager Design Tree****. You can also select the feature from the drawing area. For selecting the feature from the drawing area, select any face of the feature from the drawing area and move the cursor away from the feature. Now, move the cursor back to the selected face and right-click to invoke the shortcut menu. Choose the* ***Select Feature*** *option from the shortcut menu to select the feature. The preview of the filleted feature is displayed in the drawing area.*

Multiple Radius Fillet

Using the **Multiple radius fillet** option provided in the **Fillet PropertyManager** you can specify a fillet with different radii to all the selected edges. For creating a fillet feature using the multiple radius option, preselect the edges, faces, or features or select them after invoking the **Fillet PropertyManager**. After invoking the **Fillet** tool, select the **Multiple radius fillet** check box. The preview of the fillet feature with the default value is displayed in the drawing area. You will notice that you are provided with different callouts for each selected entity. Figure 6-24 shows the preview of the fillet feature with **Multiple radius fillet** check box selected. The names of the selected entities are displayed in the **Edges**, **Faces**, **Features and Loops** display box. The boundaries of the currently selected entity in the display box are highlighted in yellow color. You can set the value of each selected entity by using the **Radius** spinner or specify the value of fillet radius in the radius callout as shown in Figure 6-25. As you modify the value of the radius, the preview of the fillet feature modifies dynamically in the drawing area. Figure 6-26 shows the fillet created using the multiple radius fillet.

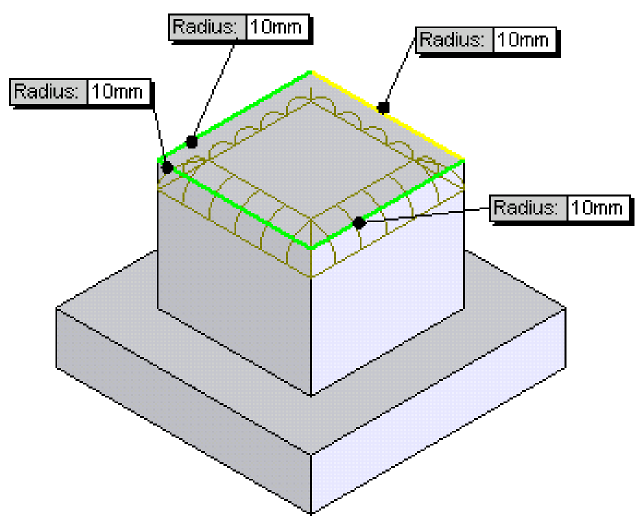


Figure 6-24 *Preview of the fillet feature with* ***Multiple radius fillet*** *check box selected*

Fillet With and Without Tangent Propagation

In SolidWorks you can add a fillet feature to a model with or without the tangent propagation.

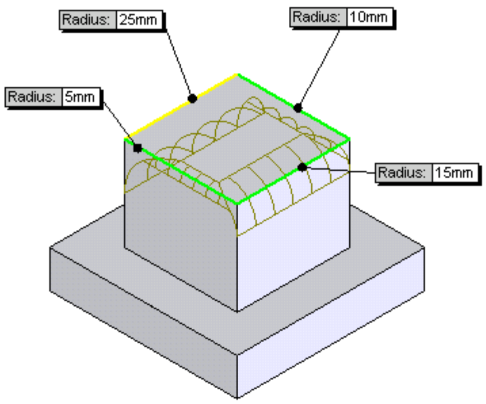


Figure 6-25 *Different radii specified in each radius callout*

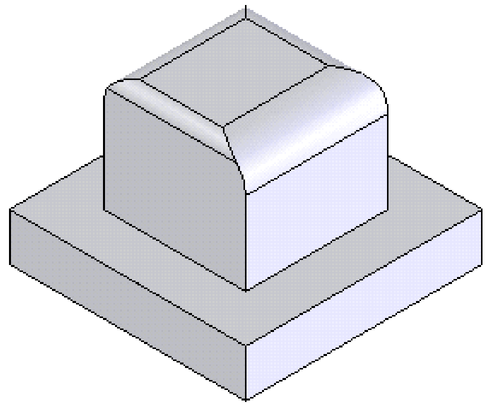
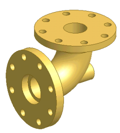


Figure 6-26 *Resultant fillet*



Tip. *The **Full preview** radio button available in the **Fillet PropertyManager** is used to preview the fillet feature before actually creating the feature. The **Full preview** radio button is selected by default. If you select the **Partial preview** radio button, then you can view only the partial preview of the fillet feature. If you select a face to add a fillet feature and select the **Partial preview** radio button then you cannot preview the fillet feature created to all the edges adjacent to the selected face. You can preview only the fillet on the single edge of the selected face. Use the A key from the keyboard to cycle the preview of the fillet feature on other edges of the selected face. If you select the **No preview** radio button, you cannot see the preview of the fillet feature.*

When you invoke the **Fillet PropertyManager**, you can observe that by default the **Tangent propagation** check box is selected. Therefore, if you select an edge, face, feature, or loop to fillet, then it will automatically select other entities that are tangential to the selected entity. Thus, it will apply the fillet feature to all the entities that are tangential to the selected one. If you clear the **Tangential propagation** check box, the fillet is applied only to the selected entity. Figure 6-27 shows the entity to be selected to add a fillet feature. Figures 6-28 and 6-29 show the fillet feature created with the **Tangent propagation** check box cleared and selected respectively.

Tip. *You can also drag and drop the fillet features created on one edge to the other edge. Using the left mouse button select the fillet feature from the* ***FeatureManager Design Tree*** *or from the drawing area and hold down the left mouse button and drag the cursor and release the left mouse button to drop the feature on the required edge or face. You can also copy the fillet feature and paste it on the selected entity.*

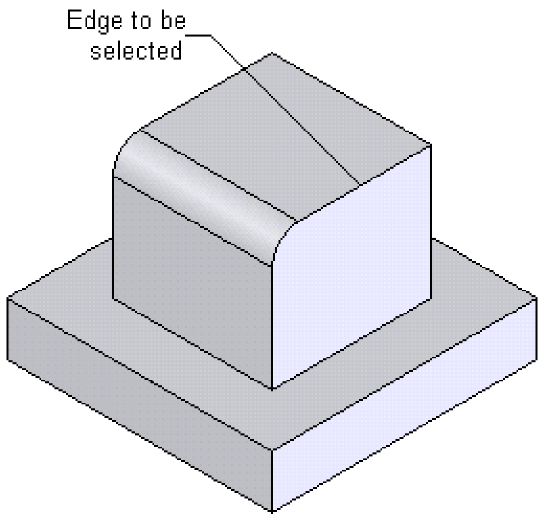


Figure 6-27 *Edge to be selected to apply the fillet feature*

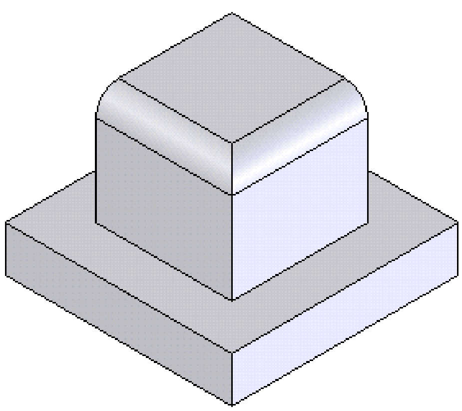


Figure 6-28 *Fillet feature created with the* ***Tangent propagation*** *check box cleared*

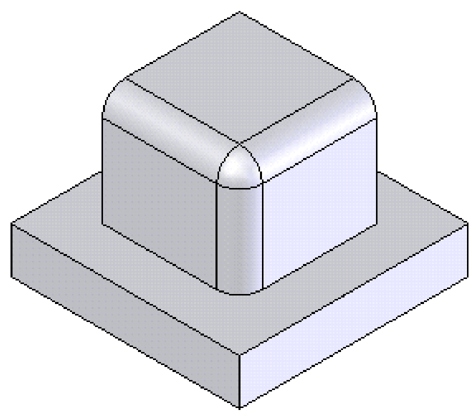


Figure 6-29 *Fillet feature created with the* ***Tangent propagation*** *check box selected*

Setback Fillets

The setback fillet is created where three or more than three edges are merged into a vertex. This type of fillet is used to smoothly blend transition surfaces generated from the edges to the fillet vertex. This smooth transition is created between all the selected edges and the vertex selected for the setback type of fillet. To create a setback fillet, invoke the **Fillet PropertyManager** and select three or more than three edges to apply the fillet. Note that the edges should share the same vertex. The preview of the fillet will be displayed in the drawing area. Now, use the left mouse button at the black arrow on the **Setback Parameters** rollout to expand the rollout. The **SetBack Parameters** rollout is displayed as shown in Figure 6-30. This rollout will be used to specify the setback parameters.

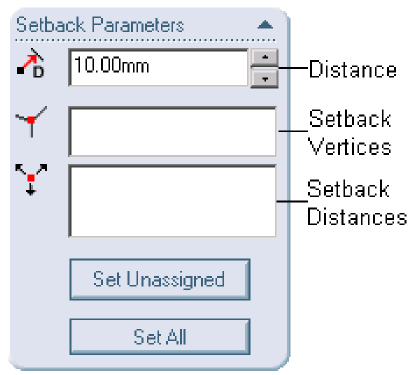


***Figure 6-30** The **Setback Parameters** rollout*

Using the left mouse button click once in the **Setback Vertices** display area to invoke the setback vertex selection command. Now, select the vertex where the edges meet. Figure 6-31 shows the selected edges and the vertex assigned the setback parameters.

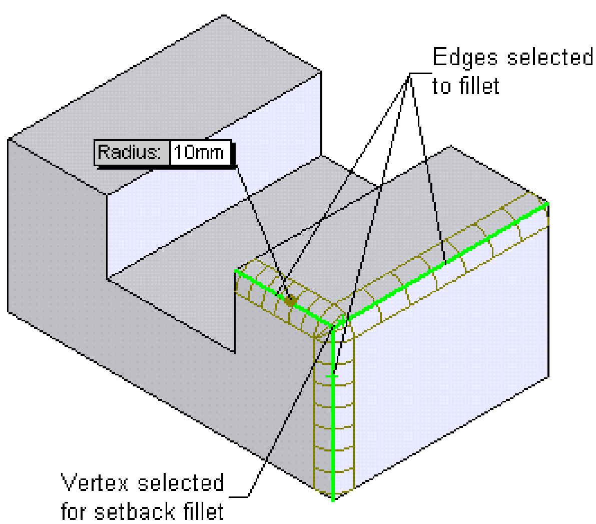


***Figure 6-31** Edges and vertex to be selected to apply the set back fillet feature*

When you select the vertex for the setback fillet, you will observe that callouts with unassigned setback distances are displayed in the drawing area. The name of the selected vertex is displayed in the **Setback Vertices** display area. The names of the selected edges are displayed in the **Setback Distances** edit box. Select the name of the edge in the **Setback Distances** edit box to assign a setback distance to that edge. A red color arrow is displayed along that edge.

Using the **Distance** spinner provided in the **Setback Parameters** rollout, assign a setback distance. Similarly, assign the setback distance to all the edges. You can also assign the setback distance directly by specifying the value in the setback callouts displayed in the drawing area. As discussed earlier, the preview of the fillet is updated automatically when you assign any value. The **Set Unassigned** button available in the **Setback Parameters** rollout is used to assign the setback distance displayed in the **Distance** spinner to all the unassigned edges. The **Set All** button is used to assign the setback distance displayed in the **Distance** spinner to all the edges. Figure 6-32 shows the preview of the setback fillet and Figure 6-33 shows a setback fillet on the right of the model and a normal fillet on the other side of the model.

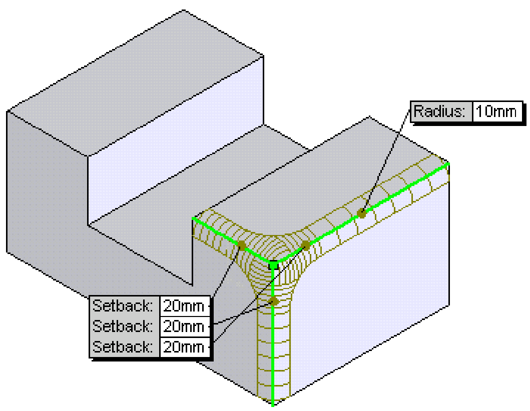


Figure 6-32 *Preview of the setback fillet*

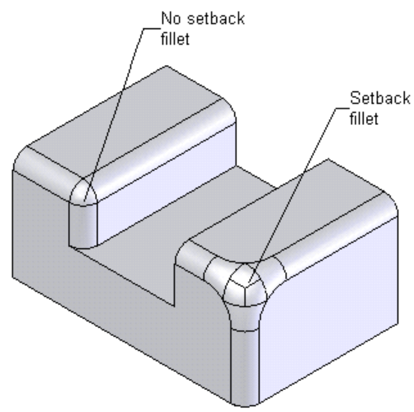


Figure 6-33 *Simple and setback fillet features*

Other Fillet Options

You are also provided with various other fillet options using which you can create an accurate and aesthetic design. The other fillet options available in the **Fillet PropertyManager** are **Keep features**, **Round corners**, **Controlling the Overflow type**, and so on. There options are discussed next.

Keep feature

If you have boss or cut features in a model and the fillet created is large enough to consume those features, it is recommended that you select the **Keep features** check box available in the **Fillet Options** rollout. This check box is selected by default, but you should confirm before creating any fillet feature. If you clear this check box, the fillet feature will consume the features that will obstruct its path. Note that the features that are consumed by the fillet feature are not deleted from the model. They disappear from the model because of some geometric inconsistency. When you rollback, suppress, or delete the fillet, the consumed features will reappear. You will learn more about rollback and suppress in later chapters. Figure 6-34 shows the model and the edge to be selected for fillet. Figure 6-35 shows the fillet feature with **Keep features** check box cleared. Figure 6-36 shows the fillet feature with **Keep features** check box selected.

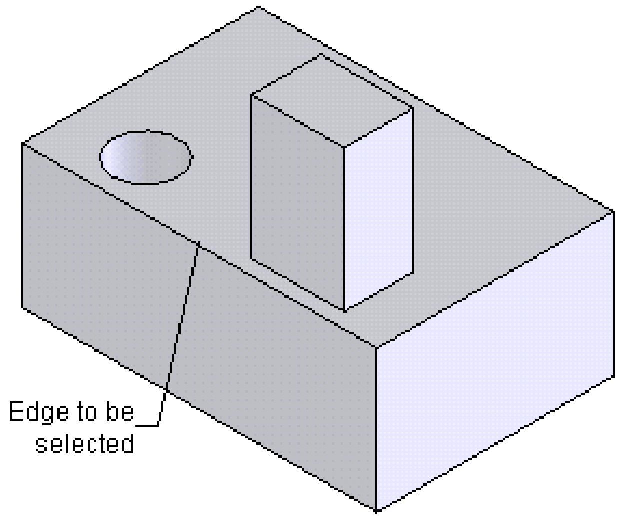


Figure 6-34 Edge to be selected to apply the fillet feature

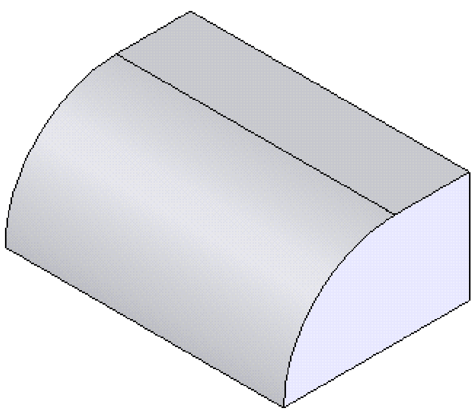


*Figure 6-35 Fillet feature created with the **Keep features** check box cleared*

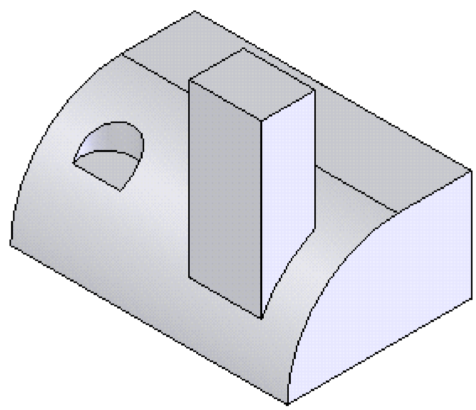


*Figure 6-36 Fillet feature created with the **Keep features** check box selected*

Round corners

The **Round corners** option is used to round the edges at the corners of the fillet feature. To create a fillet feature with round corners, select the **Round corners** check box from the **Fillet Options** rollout after specifying all the parameters of the fillet feature. Figure 6-37 shows a fillet feature created with the **Round corners** check box cleared and Figure 6-38 shows a fillet feature created with the **Round corners** check box selected.

Overflow type

The **Overflow type** area is used to specify the physical condition that the fillet feature should adopt when it extends beyond an area. By default, SolidWorks automatically adopts the best possible flow type to accommodate the fillet, depending on the geometric conditions. This is because the **Default** option is selected by default in the **Overflow type** area. The other two options available in this area are discussed next.

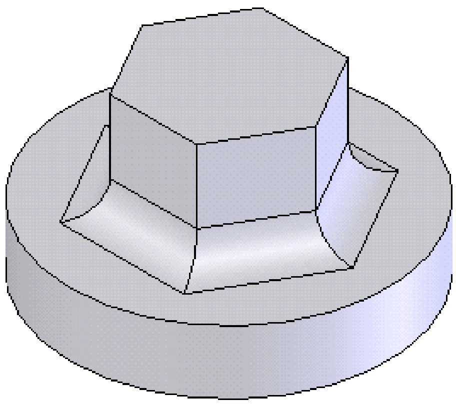


*Figure 6-37 Fillet feature created with the **Round corners** check box cleared*

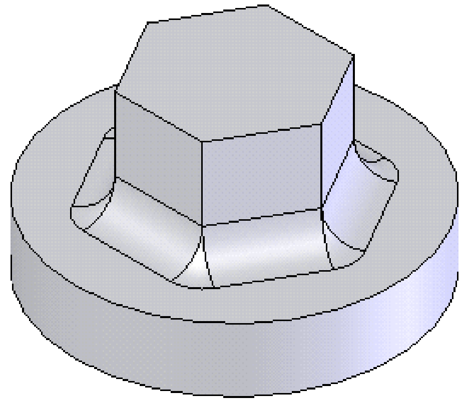


*Figure 6-38 Fillet feature created with the **Round corners** check box selected*

Keep edge

The **Keep edge** radio button is selected when the fillet feature extends beyond a specified area. Therefore, to accommodate the fillet feature this option will divide the fillet into multiple surfaces and the adjacent edges are not disturbed. A dip is created at the top of the fillet feature. Figure 6-39 shows a fillet feature created with the **Keep edge** radio button selected from the **Overflow type** area.

Keep surface

The **Keep surface** radio button available in this area is selected to accommodated the fillet feature by trimming the fillet feature. This will maintain the smooth rounded fillet surface but it disturbs the adjacent edges. As this option maintains the smooth fillet surface, it extends the adjacent surface. Figure 6-40 shows a fillet feature created with the **Keep surface** radio button selected from the **Overflow type** area.

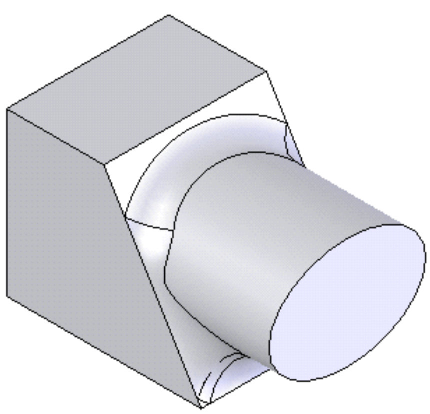


*Figure 6-39 Fillet feature created with the overflow type as **Keep edge***

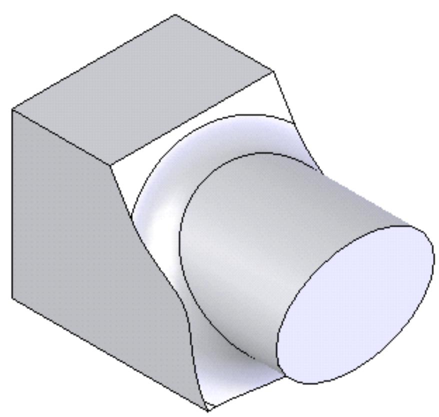


*Figure 6-40 Fillet feature created with the overflow type as **Keep surface***

Variable Radius Fillet

The variable radius fillet is created by specifying different radii along the length of selected edge at specified intervals. Depending on the options, you can create a smooth transition or a straight transition between the vertices to which the radii are applied. To create a variable radius fillet invoke the **Fillet PropertyManager**. Select the **Variable radius** radio button from the **Fillet Type** rollout. The **Variable Radius Parameters** rollout is automatically displayed in the **Fillet PropertyManager** as shown in Figure 6-41.

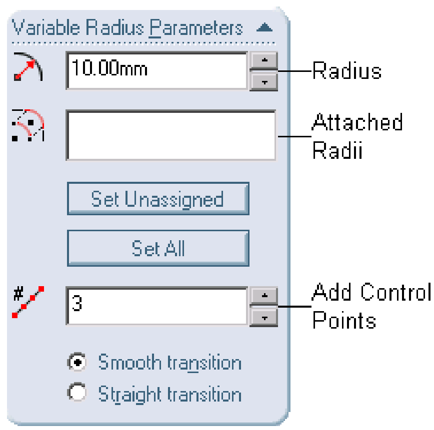


*Figure 6-41 The **Variable Radius Parameters** rollout*

You are prompted to select the edge to fillet. Using the left mouse button select the edge or edges that you want to fillet. The name of the selected edge is displayed in the **Edges, Faces, Features and Loops** display area. By default, the radius is applied at the startpoint and the endpoint. The variable radius callouts are displayed at the ends of the selected edge as shown in Figure 6-42.

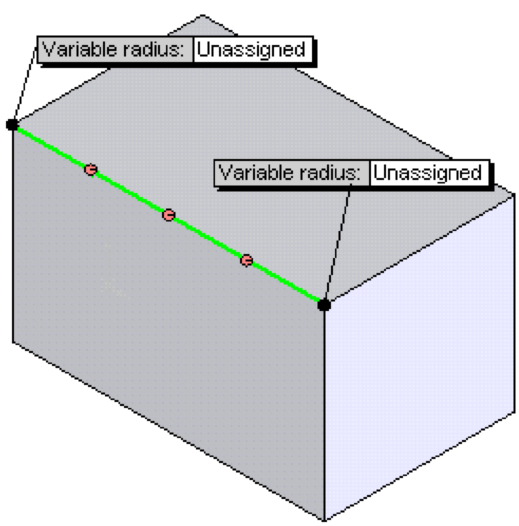


Figure 6-42 Variable radius callouts displayed at the vertices of the selected edge

The names of the vertices on which the callouts are added are displayed in the **Attached Radii** display box. You will find three red points on the selected edge because the value of the control point in the **Add Control Points** spinner is set to **3**. You can create the number of control points using the **Add Control Points** spinner. These control points are also called movable points because you can change the position of these control points. The additional radii are specified on these points on the selected edge.

Using the left mouse button select the three control points available on the selected edge. As you select the control point the **Radius and Position** callouts are displayed for each control point as shown in Figure 6-43. The name of the selected point is also displayed in the **Attached Radii** display box.

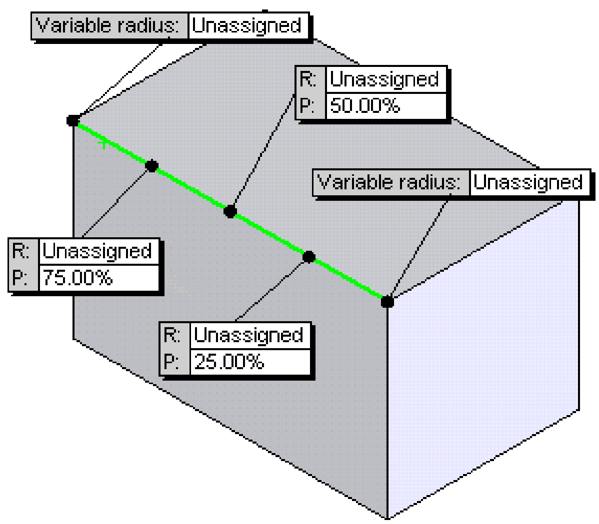


Figure 6-43 Radius and Position *callouts are displayed after selecting the control points*

You will observe that the position of three points is described in terms of percentage. You can modify the position of the points by modifying the value of percentage in the **Position** area of the **Radius and Position** callout. By following this procedure you can also modify the placement of other points. You will observe that radius value is not assigned to any of the callouts. Therefore, you need to specify the value of the radius in the callouts. Using the left mouse button select the name of the vertex or point; the name of the selected item is highlighted in yellow in its respective callout. Using the **Radius** spinner set the value of the radius for the selected item. You can also specify the value of the radius in the radius area of the callout. Set the value of each unassigned radius. You can also use the **Set Unassigned** button to assign the value displayed in the **Radius** spinner to all the unassigned radii. The **Set All** button is used to assign the same value that is displayed in the **Radius** spinner to all the radii. Figure 6-44 shows the preview of the fillet feature with modified positions of the control points with radius values specified to all the points and vertices. Figure 6-45 shows the resultant fillet feature.

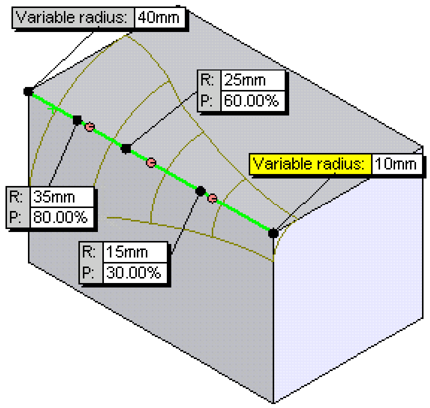


Figure 6-44 *Preview of the variable radius fillet*

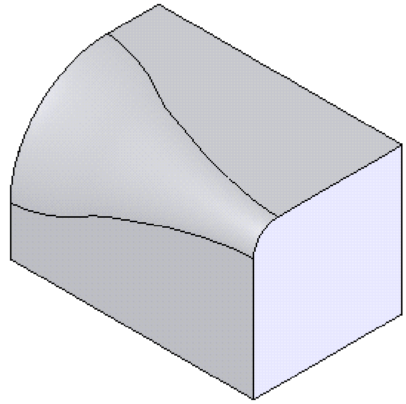


Figure 6-45 *Resultant fillet*

Smooth transition

The **Smooth transition** radio button when selected creates a smooth transition by smoothly blending the points and vertices on which you have defined the radius on the selected edge.

Straight transition

The **Straight transition** radio button when selected creates a linear transition by blending the points and vertices on which you have defined the radius on the selected edge. In this case the edge tangency is not maintained between one fillet radius and the adjacent face.

Figures 6-46 and 6-47 show the fillets created with smooth transition and straight transition options respectively.

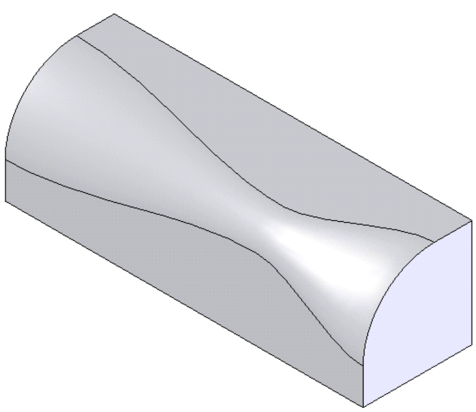


Figure 6-46 *Variable radius fillet with smooth transition*

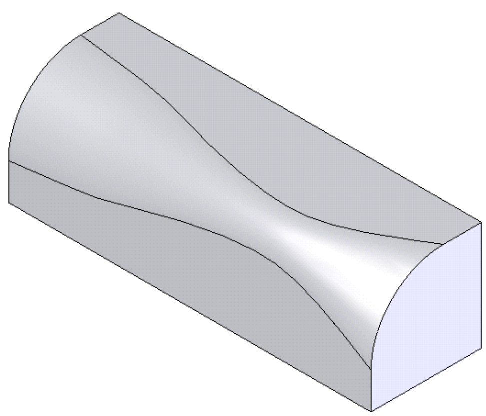


Figure 6-47 *Variable radius fillet with straight transition*

Face Fillet

Using the **Face Fillet** you can add a fillet between two sets of faces. It blends the first set of face with the second set of face. It adds or removes the material according to the geometric conditions. It can also completely or partially remove the faces to accommodate the fillet feature. To create a face fillet feature, invoke the **Fillet PropertyManager** and select the **Face fillet** radio button from the **Fillet Type** rollout. The **Item To Fillet** rollout is modified and provides the **Face Set 1** and **Face Set 2** display areas. The **Fillet PropertyManager** is shown in Figure 6-48 with the **Face fillet** radio button selected.

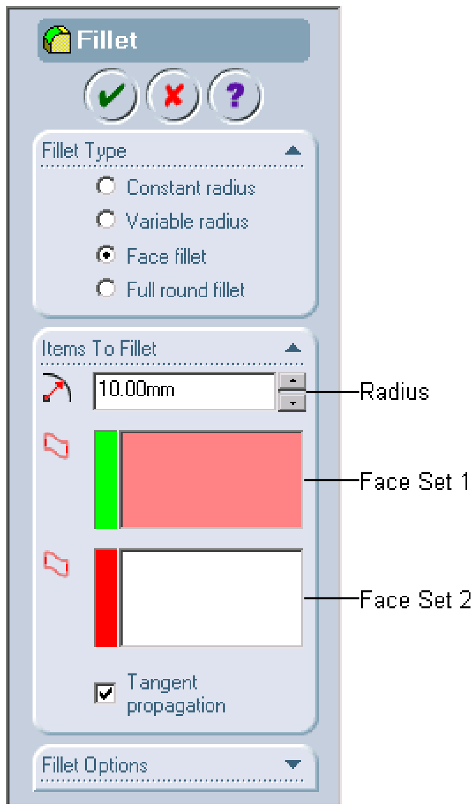


*Figure 6-48 The **Fillet PropertyManager** with **Face fillet** radio button selected*

You are prompted to select the faces to fillet for face set 1 and face set 2. Using the left mouse button select the first set of faces. You can select even more than one face in a set. The name of the selected faces is displayed in the **Face Set 1** display box and the selected faces are displayed in green color. The **Face Set 1** callout with radius is displayed in the drawing area. Click in the **Face Set 2** display area to invoke the selection tool and select the second set of faces. The second set of selected faces is displayed in red color and the **Face Set 2** callout is displayed in the drawing area. Now, set the value of the radius in the **Radius** spinner. The **Tangent propagation** check box is used to create the face fillet tangent to the adjacent faces. This check box is selected by default. If you clear this check box, the fillet is not forced to be tangent to the adjacent faces. Figure 6-49 shows the faces to be selected to apply the face fillet. Figure 6-50 displays the resultant fillet with three faces of the slot completely eliminated after applying the fillet.

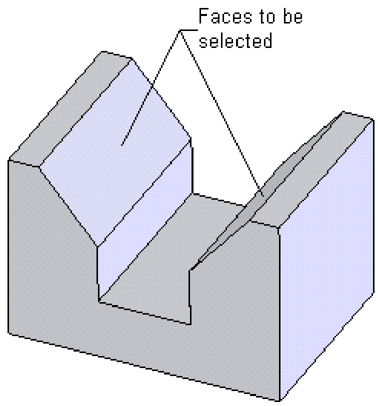


Figure 6-49 Faces to be selected

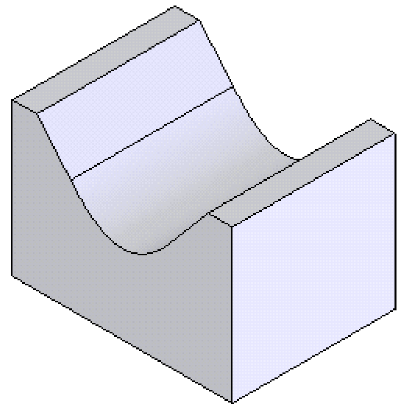


Figure 6-50 Resultant fillet

Face Fillet Using the Hold Line

Using the face fillet created with the hold line you can specify the radius and the shape of the fillet by determining a hold line. A hold line can be a set of edges, or a split line projected on a face. You will learn more about split lines in the later chapters. The radius of the fillet is determined by the distance between the hold line and the edges or faces selected to be filleted. To create a face fillet using the hold line invoke the **Fillet PropertyManager**. Using the left mouse button click on the black arrow available at the top right of the **Fillet Options** rollout to display the rollout as shown in Figure 6-51.

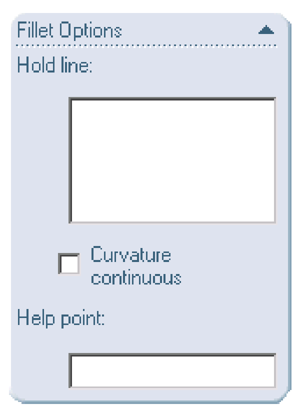


*Figure 6-51 The **Fillet Options** rollout*

The selection mode is active in the **Face Set 1** display area and you are prompted to select the faces to fillet for face set 1 and face set 2. Using the left mouse button select the faces for the face set 1. The names of the selected faces are displayed in the **Face Set 1** display area and the selected face is displayed in green. Now, click in the **Face Set 2** display area to activate the selection mode and select the faces to add in the face set 2. The selected faces are displayed in red color. By default, the **Tangent propagation** check box is selected. Therefore, you do not need to select the tangent faces in both the face sets. This is because it will automatically select the faces tangent to the selected face. Now, using the left mouse button click in the **Hold line** display box and select the hold line or lines. Now, choose the **OK** button from the **Fillet PropertyManager**. Figure 6-52 shows an example in which the faces and the hold line are selected. Figure 6-53 shows the resultant face fillet using the hold line.

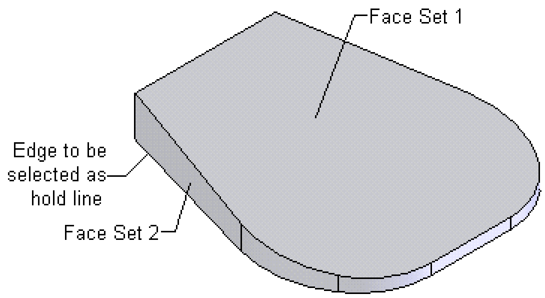


Figure 6-52 *Faces and hold line to be selected*

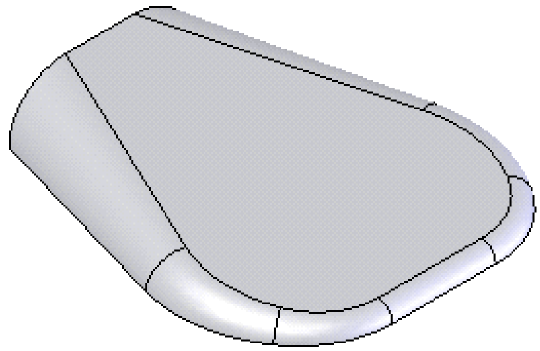


Figure 6-53 *Resultant fillet*

Curvature Continuous in the Face Fillet with Hold Line

The **Curvature continuous** check box is selected to apply the face fillet feature with continuous curvature throughout the fillet feature. Note that fillet with continuous curvature is possible only by creating a face fillet feature with hold line. You have to specify the hold lines on both set of faces. Figure 6-54 shows a model in which face fillet using the hold line is created on both the pillars. On the right pillar the face fillet is created with the **Curvature continuous** check box cleared and on the left pillar the face fillet is created with the **Curvature continuous** check box selected.

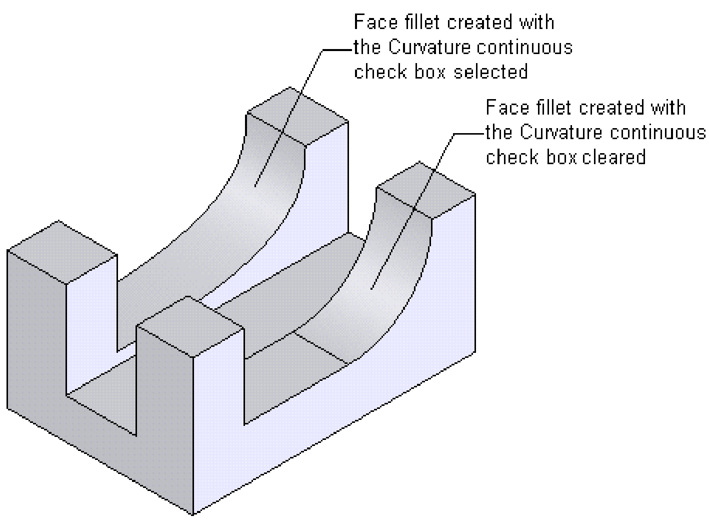


Figure 6-54 *Face fillet created with the* ***Curvature continuous*** *check box selected and cleared*

Full Round Fillet

The full round fillet is used to add a semicircular fillet feature. To create a full round fillet invoke the **Fillet PropertyManager** and select the **Full round fillet** radio button. The **Item To Fillet** rollout is modified as shown in Figure 6-55. The selection mode is active in the **Side Face Set 1** display area and you are prompted to select faces for the center and side face sets. Select the first face for the side face set 1. The selected face is displayed in green. Now, click in the **Center Face** display area and select the center face. The selected face is displayed in brown color. Next, click in the **Side Face Set 2** display area and select the face for the side face set 2. Choose the **OK** button from the **Fillet PropertyManager**. Figure 6-56 shows the two faces being selected; the third face to be selected is the left face parallel to the first selected face. Figure 6-57 shows the resultant full round fillet.

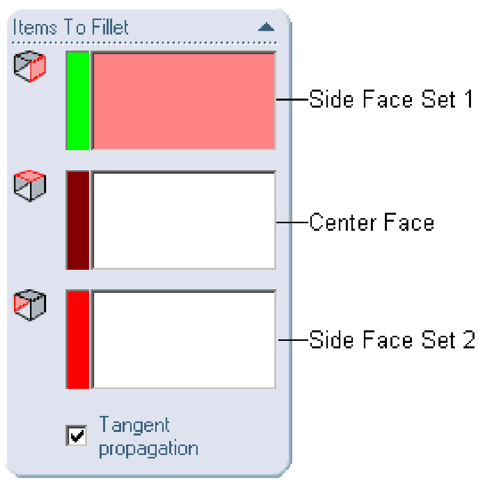


*Figure 6-55 The **Item to Fillet** rollout when the **Full round fillet** radio button is selected from the **Fillet Type** rollout*

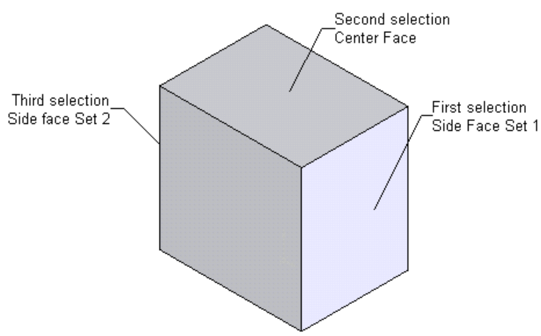


Figure 6-56 Faces selected to create full round fillet.

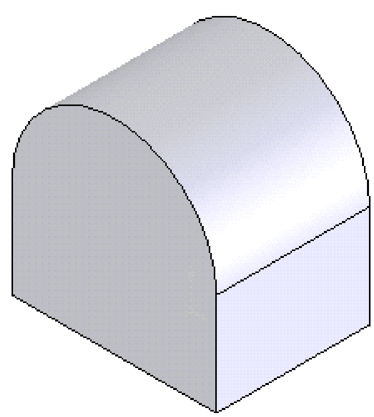


Figure 6-57 Resultant full round fillet

Selection Methods

As you have learned about basic and advanced modeling tools, it is necessary for you to learn about some selection methods using which you can increase your productivity and speed of modeling. The selection methods that can increase your speed of creating fillets and chamfers are discussed next.

Select Other

The **Select Other** option is the most common tool to cycle through the entities for selection. This option is used when the selection is difficult in a multi-featured complex model. Before invoking any other tool, select any entity and right-click to invoke the shortcut menu. Choose the **Select Other** option from the shortcut menu. The select cursor is replaced by the cycle through cursor. Also, any one face of the model is selected and the boundary of the face is highlighted in green. You can cycle through the other faces using the right-click and when the required face is selected, use the left mouse button to confirm the selection.

Select Loop

The **Select Loop** option is used to select the loops and using this tool you can also cycle through various loops before confirming the selection. This option is very useful when you are working with a complex model and you have to select a loop from that model. Using the select cursor, select any of the edge of the loop and right-click to invoke the shortcut menu. Choose the **Select Loop** option from the short cut menu. The loop that is possible by selecting that edge will be highlighted in green and an arrow will be displayed in yellow color. Move the cursor on that arrow and when the arrow is displayed in red, use the left mouse button to cycle through the loops. Repeat this until you select the required loop. Figure 6-58 shows a loop selected using the **Select Loop** option. Figure 6-59 shows the second loop selected when the left mouse button is used on the arrow to cycle through the loops.

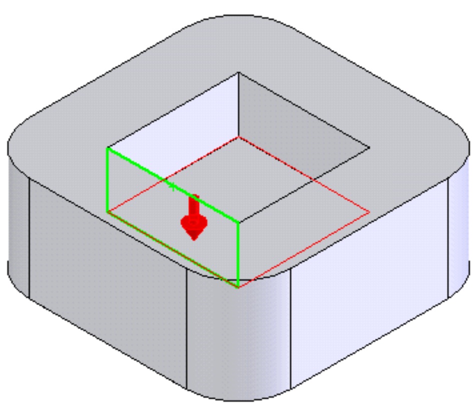


Figure 6-58 *Loop selected using the* ***Select Loop*** *tool*

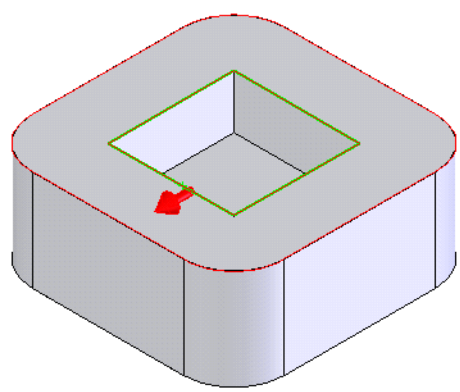
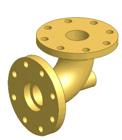


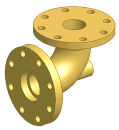
Figure 6-59 *Second loop is selected while cycling through the loops*



Tip. *The* ***Select Midpoint*** *option available in the shortcut menu is generally used in the sketching environment or while creating 3D sketches. You will learn more about 3D sketches in the later chapters.*

Select Tangency

The **Select Tangency** option is used to automatically select the edges or faces that are tangent to the selected face. This option is available in the shortcut menu only when a face or edge is tangent to the selected face or edge. For using this option select any face or edge using the select tool and right-click to invoke the shortcut menu and choose the **Select Tangency** option from the shortcut menu.



Tip. *For creating a fillet or a chamfer feature if you select a face, the fillet or chamfer is applied to all the edges of that face. Consider a case in which you have a blind cut feature on the top face of a block. You want to fillet only the upper edges of the cut feature. You can use the* ***Select Loop*** *option to execute feature creation. If you select the top face of the model, then all the edges of the top of the block and the edges of the cut will be filleted. Therefore, after selecting the top face of the block press and hold down the CTRL key and select any one upper edge of the cut. Now, apply the fillet feature to this combination of selection. You will observe that only the upper loop of edges of the cut will be filleted instead of the whole upper face. In the same way you can also fillet only the edges of the block when you are also having a slot on the top face of the model.*

Creating Chamfer

Toolbar:	Features > Chamfer
Menu:	Insert > Features > Chamfer

Chamfering is defined as a process in which the sharp edges are bevelled in order to reduce the area of stress concentration. This process also eliminates the sharp edges and corners that are not desirable. In SolidWorks chamfer is created using the **Chamfer** tool. This tool is invoked by using the **Chamfer** button available in the **Features** tool bar. You can also choose **Insert > Features > Chamfer** from the menu bar to invoke this tool. When you invoke the chamfer tool, the **Chamfer PropertyManager** is displayed as shown in Figure 6-60. Various types of chamfers created using the **Chamfer PropertyManager** are discussed next.

Edge Chamfer

The chamfers that are applied to the edges are known as edge chamfer. To create an edge chamfer invoke the **Chamfer PropertyManager** and then select the edges to chamfer. When you select an edge to chamfer, the preview of the chamfer feature with a distance and angle callout is displayed in the drawing area. The name of the selected edge is displayed in the **Edges and Faces or Vertex** display area. Also, the selected entity is highlighted in yellow color and a yellow arrow is also displayed in the preview. Figure 6-61 shows the edge to be selected for chamfering and Figure 6-62 shows the preview of the chamfer feature.

By default, the **Angle distance** radio button is selected. Therefore, the distance and angle callout is displayed in the drawing area. You can set the value of the distance and angle using the **Distance** spinner and the **Angle** spinner or you can insert the value of distance and angle directly in the distance and angle callout. The **Flip direction** check box is used to specify the direction of distance measurement. If you select the **Flip direction** check box, the arrow is also flipped in the preview in the drawing area. You can also flip the direction by clicking the arrow in the drawing area.

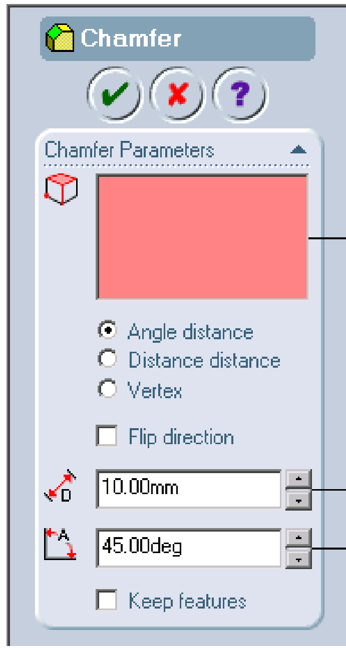


Figure 6-60 *The **Chamfer PropertyManager***

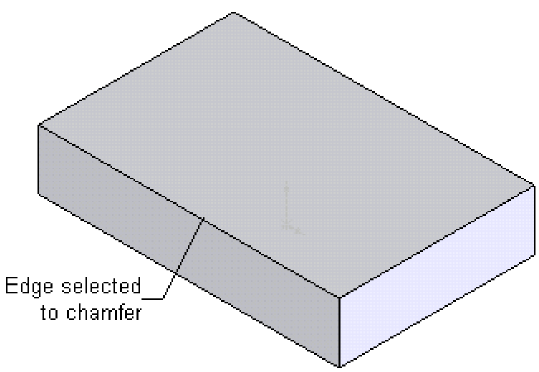


Figure 6-61 *Edge selected to chamfer*

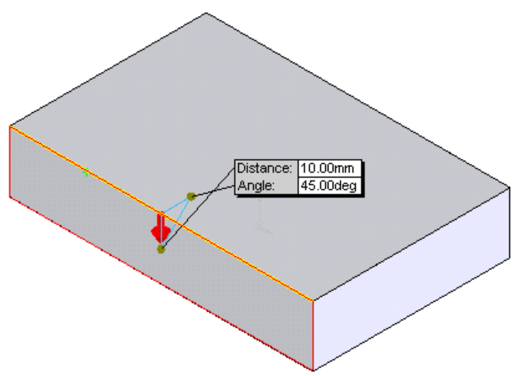


Figure 6-62 *Preview of the chamfer feature*

If you select the **Distance distance** radio button from the **Chamfers Parameters** callout, the **Flip direction** check box is replaced by the **Equal distance** check box. The angle and distance callout is replaced by distance 1 and distance 2 callouts. By default, the **Equal distance** check box is selected. Set the value of the chamfer distance in the **Distance 1** check box or specify the value in the callout. If you want to specify different distances for creating the chamfer, clear the **Equal distance** check box. The **Distance 2** spinner is displayed in the **Chamfer Parameters** rollout.

After specifying all the parameters choose the **OK** button from the **Chamfer PropertyManager**. Figure 6-63 shows the chamfer created on a base plate.

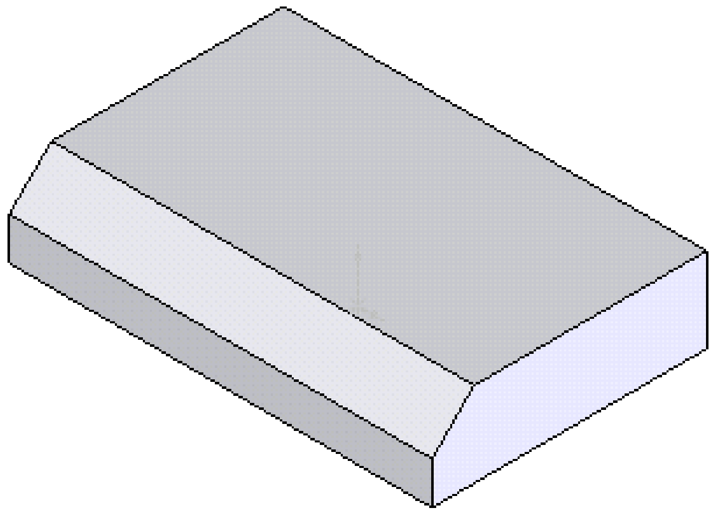
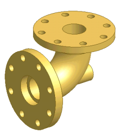


Figure 6-63 *Chamfer created on a base plate*



Tip. *You can also select the face for applying the chamfer feature. If you select the face for applying the chamfer, then the chamfer is applied to all the edges of the selected face.*

Vertex Chamfer

Using the chamfer tool you can also add chamfer to a selected vertex. It will chop the selected vertex to the specified distance. To create the vertex chamfer invoke the **Chamfer PropertyManager** and select the **Vertex** radio button from the **Chamfer Parameters** rollout. The preview of the chamfer is displayed in the drawing area with distance callouts. Figure 6-64 shows the vertex to be selected and Figure 6-65 shows the preview of the vertex chamfer.

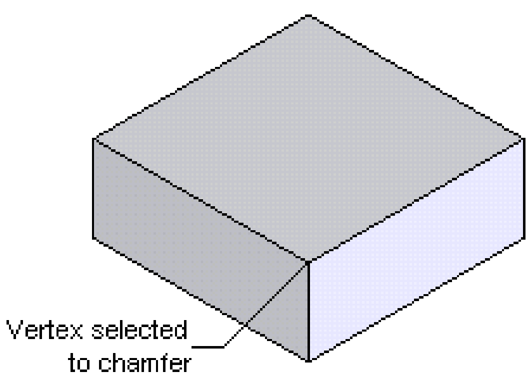


Figure 6-55 *Vertex to be selected*

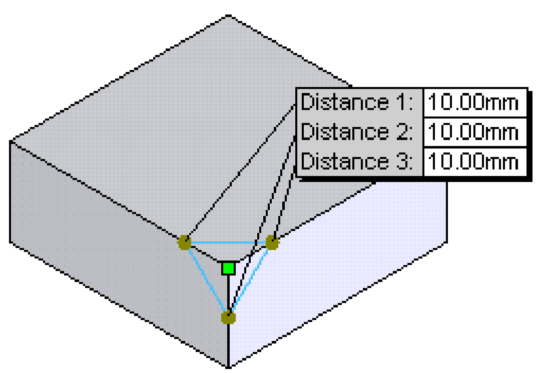


Figure 6-56 *Preview of the vertex chamfer*

Set the value of the chamfer distance along each edge in the **Distance 1**, **Distance 2**, **Distance 3** spinners. You can also specify the value of chamfer distance in the distance callouts. If you want to specify equal distance to all the edges then select the **Equal distance** check box. After specifying all the parameters choose the **OK** button from the **Chamfer PropertyManager**. Figure 6-66 shows the vertex chamfer feature applied to the base feature.

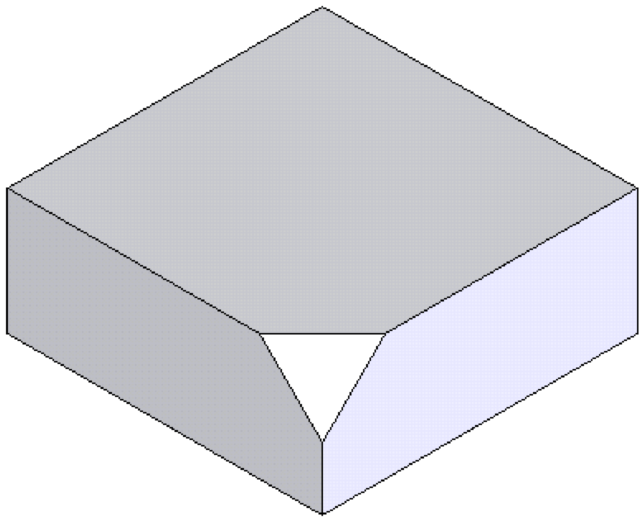


Figure 6-66 *Vertex chamfer created on a base feature*

Chamfer with and without Keep Feature option

If you have boss or cut features in a model and the chamfer created is large enough to consume those features, it is recommended that you select the **Keep features** check box. If this check box is cleared, the chamfer feature will consume the features that will obstruct its path. Note that the features that are consumed by the chamfer feature are not deleted from the model. They are removed from the model because of some geometric inconsistency. When you rollback or delete that chamfer, the consumed features will reappear. Figure 6-67 shows the chamfer feature with the **Keep features** check box cleared. Figure 6-68 shows the chamfer feature with the **Keep features** check box selected.

Creating the Shell Feature

Toolbar:	Features > Shell
Menu:	Insert > Features > Shell

Shelling is defined as the process in which the material is scooped out from a model and the resultant model is hollowed from inside. The resultant model will be a hollow model with walls of specified thickness and cavity inside. The selected face or faces of the model are also removed in this operation. If you do not select any face to remove, it will create a closed hollow model. You can also specify multiple thickness to the walls. You can create shell feature using the **Shell** tool. Use the **Shell** button from the **Features** toolbar or choose **Insert** > **Features** > **Shell** from the menu bar to invoke the **Shell** tool. The **Shell1**

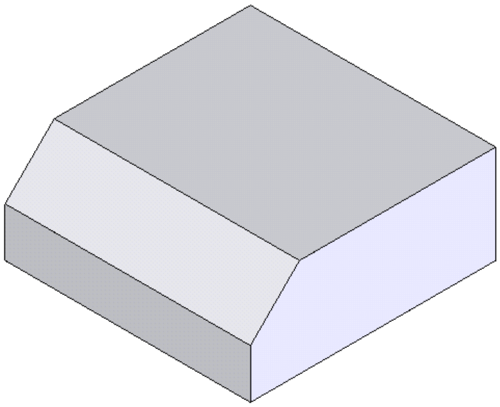


*Figure 6-67 Chamfer feature with **Keep features** check box cleared*

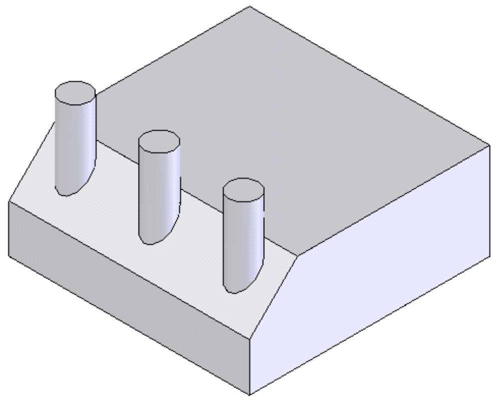


*Figure 6-68 Chamfer feature with **Keep features** check box selected*

PropertyManager is invoked and the confirmation corner is also displayed. The **Shell1 PropertyManager** is displayed as shown in Figure 6-69.

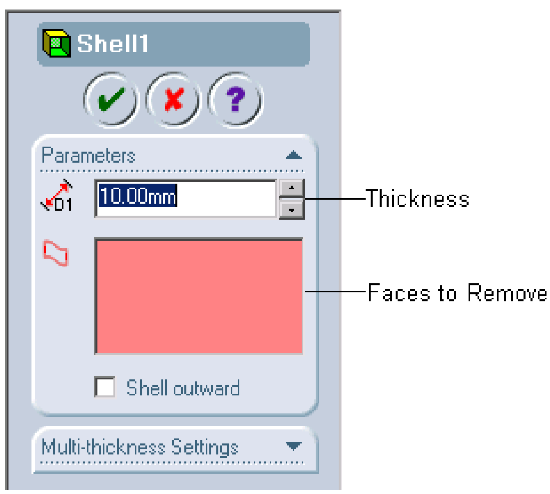


*Figure 6-69 The **Shell1 PropertyManager***

You are prompted to select the faces to remove. Select the face or faces from the model that you want to remove. The selected faces will be highlighted in green color and their names are displayed in the **Faces to Remove** display area. Set the value of the wall thickness in the **Thickness** spinner and choose the **OK** button from the **Shell1 PropertyManager**. Figure 6-70 shows the face selected to remove and Figure 6-71 shows the resultant shell feature created.

If none of the faces are selected to be removed, then the resultant model will be hollowed from inside with no face removed. Figure 6-72 shows a model in the **Hidden Line Visible** mode with a shell feature in which faces are not selected to be removed.

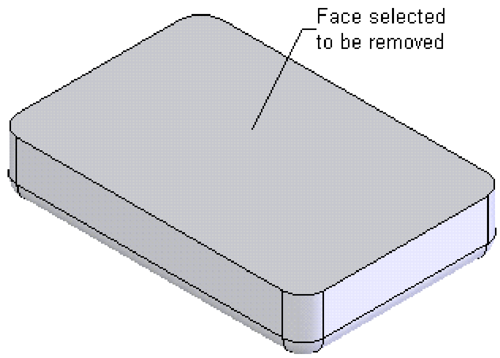


Figure 6-70 *Face selected to be removed*

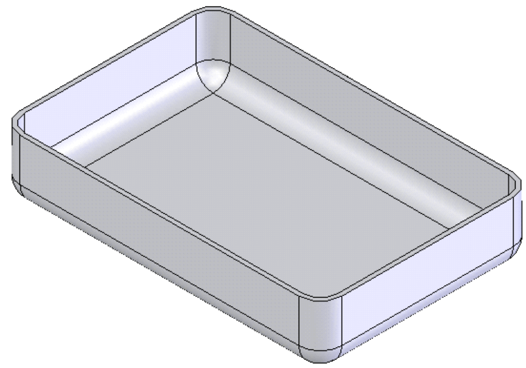


Figure 6-71 *The resultant shell feature*

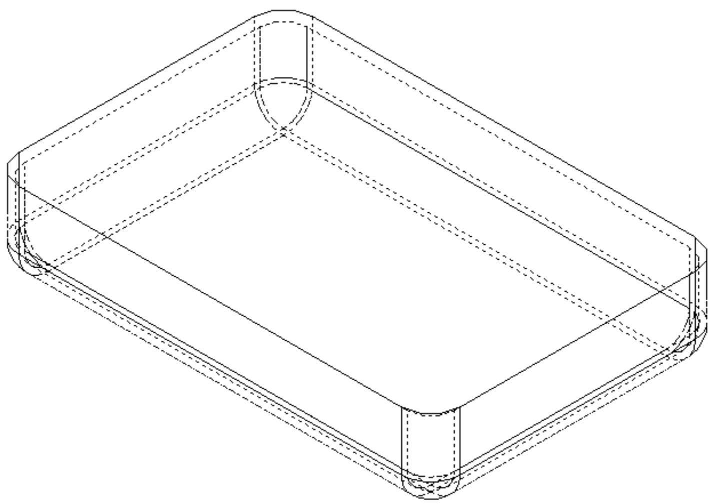
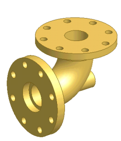


Figure 6-72 *Shell feature with faces not selected to remove*

The in-built artificial intelligence of shelling command in SolidWorks enables the shell feature to decide how much quantity of material to be removed depending upon the geometric conditions. Refer to Figure 6-73 in which the wall thickness of the shell feature is less for uniform shelling of the entire model. Refer to Figure 6-74 in which the wall thickness is more because of which it cannot accommodate the uniform shelling of the entire model. Therefore, the shell feature will not remove the material from those area where the material removal is not possible.



Tip. *If the thickness of the shell feature is more than the radius of the fillet feature then the fillet will not be included in the shell feature. Therefore, it results in sharp edges after adding the fillet. Same is with the chamfer feature.*

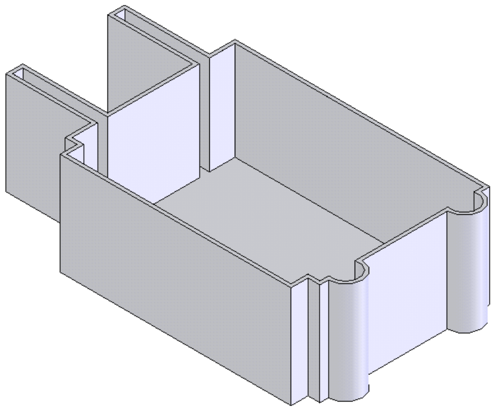


Figure 6-73 Shell feature with smaller shell thickness

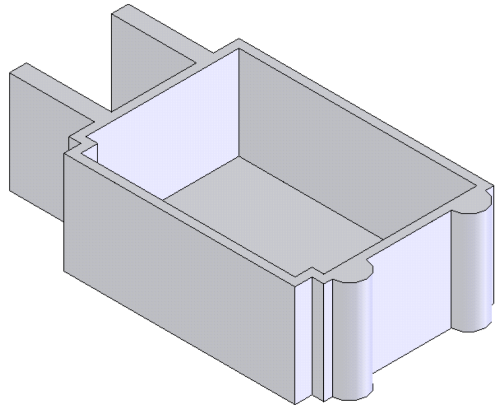
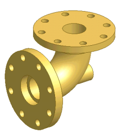


Figure 6-74 Shell feature with larger shell thickness



Tip. *The* ***Shell outward*** *check box is selected to create the shell feature on the outer side of the model.*

The faces selected to be removed in the shell feature can be a planar face or a curved face. But creating a shell by removing a curved face depends upon the geometry of the curved face to adopt the specified shell thickness and other geometric conditions.

Multi-thickness Shell

Using this option available in the shell tool you can specify different thickness values to the selected faces. To use this option, invoke the **Shell PropertyManager** and select the face to remove and then specify the uniform thickness in the **Thickness** spinner of the **Parameters** rollout. Now, select the blue arrow on the top right of the **Multi-thickness Settings** rollout to open this rollout. Click in the **Multi-thickness Faces** display area to activate the selection mode. The **Multi-thickness Settings** area with the selection mode active is displayed in Figure 6-75.

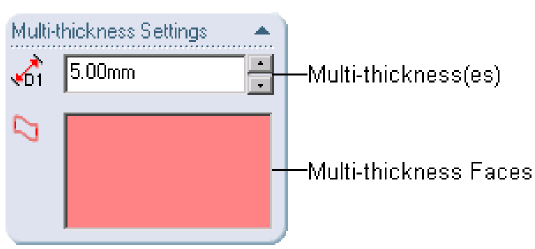


Figure 6-75 The ***Multi-thickness Settings*** *rollout*

Now, you are prompted to specify multi-thickness(es). Select the faces on which you want to specify the special thickness. Set the value of the thickness using the **Multi-thickness(es)** spinner and choose the **OK** button. Figure 6-76 shows the faces to be selected to specify the multi-thickness shell. Figure 6-77 shows the resultant shell feature.

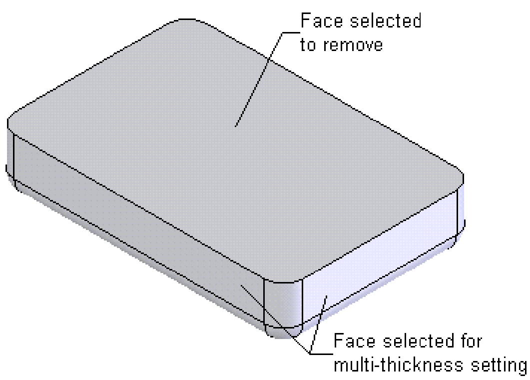


Figure 6-76 *Faces to be selected to apply multi-thickness shell*

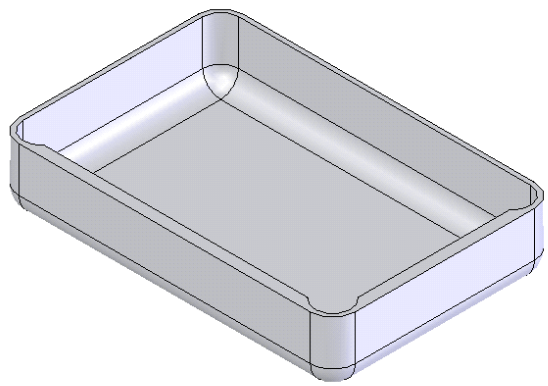


Figure 6-77 *Resultant multi-thickness shell*

TUTORIALS

Tutorial 1

In this tutorial you will create the model shown in Figure 6-78. The dimensions of the model are shown in Figure 6-79. **(Expected time: 30 min)**

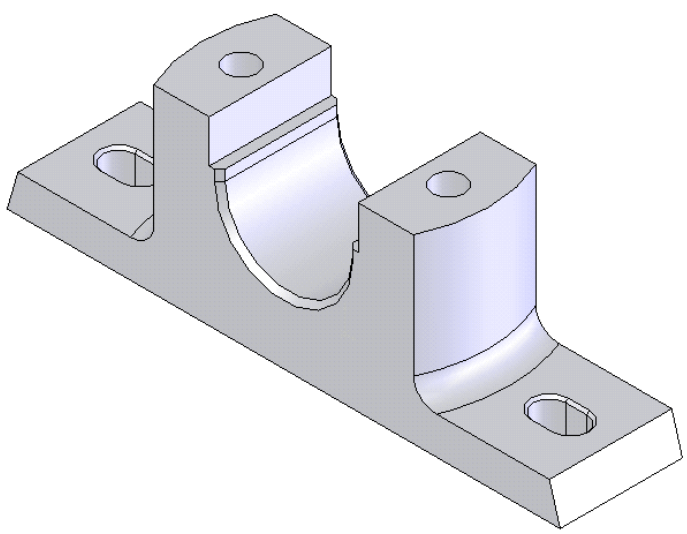


Figure 6-78 *Solid model for Tutorial 1*

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

a. Create the base feature of the model on the default plane, refer to Figures 6-80 and 6-81.
b. Create the second feature, which is a cut feature, on the top planar face of the base feature, refer to Figures 6-82 through 6-84.

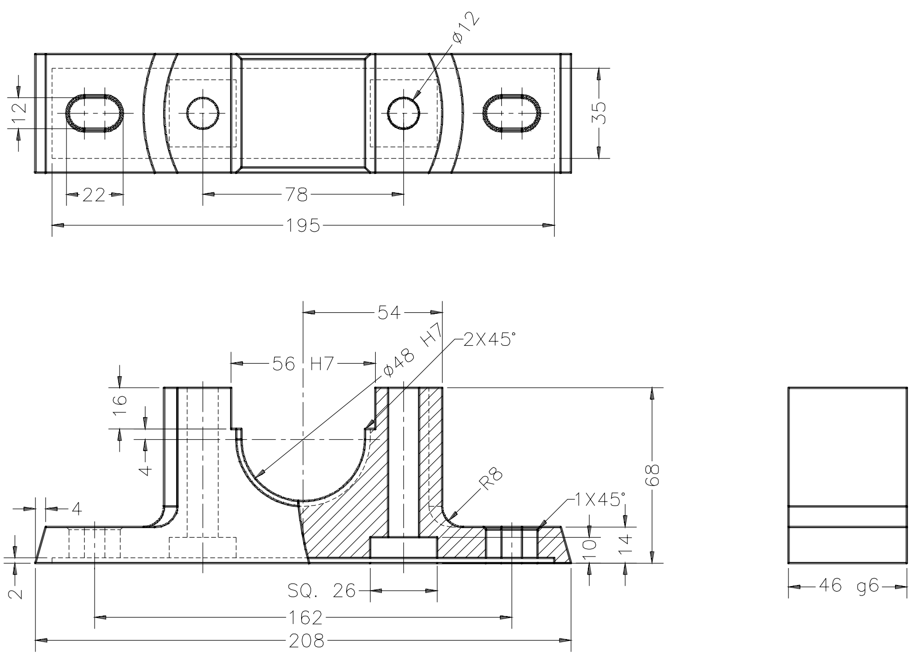


Figure 6-79 Top view, Front broken-out section view, and Side view with dimensions

c. Create the rectangular recess at the bottom of the base as a cut feature, refer to Figure 6-85.
d. Create the square cuts, which act as the recess of square head bolt, refer to Figure 6-85.
e. Create a hole feature using the **Hole PropertyManager** and modify the placement of the hole feature, refer to Figure 6-86.
f. Create the cut feature on the second top planar face of the base feature, refer to Figure 6-87.
g. Create the fillet feature on the solid model, refer to Figures 6-88 and 6-89.
h. Apply the chamfers to the solid model, refer to Figures 6-90 and 6-91.

Creating the Base Feature

1. Start SolidWorks and then open a new part file from the **Template** tab of the **New SolidWorks Document** dialog box.

2. Draw the sketch of the front view of the model using the automatic mirroring option to capture the design intent of the model.

3. Add the required relations and dimensions to the sketch as shown in Figure 6-80.

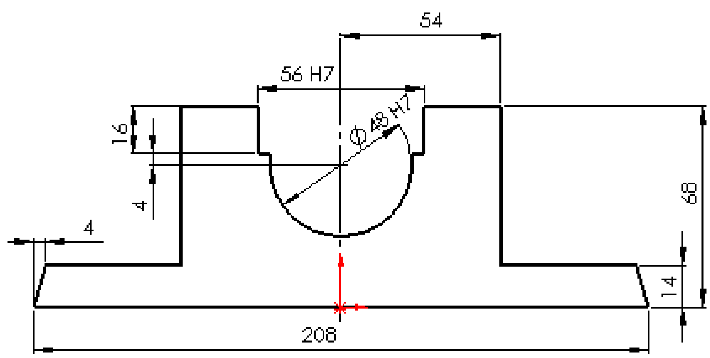
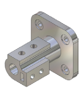


Figure 6-80 *Fully defined sketch of the base feature*



Note

*As shown in the above figure, tolerances are applied to some dimensions. After adding the dimensions to the sketch, select the dimension and add the tolerance to the dimension using the **Dimension PropertyManager** as discussed in the earlier chapters.*

*Using the **Dimension Properties** dialog box change the radial dimension of the arc to the diameter dimension.*

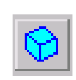
Next, you need to extrude the sketch to a distance of 46mm using the mid plane option. You will extrude the sketch using the midplane option because it is recommended that parts that are to be assembled should have default planes in the center of the model.

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

5. Right-click in the drawing area and choose the **Mid Plane** option from the shortcut menu to extrude the sketch symmetrically to both the sides of the sketching plane.

6. Set the value of the **Depth** spinner to **46** and choose the **OK** button from the **Extrude PropertyManager**.

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



The base feature created after extruding the sketch to a given depth is shown in Figure 6-81.

Creating the Second Feature

The second feature of the model is a cut feature. This feature is created by extruding a sketch

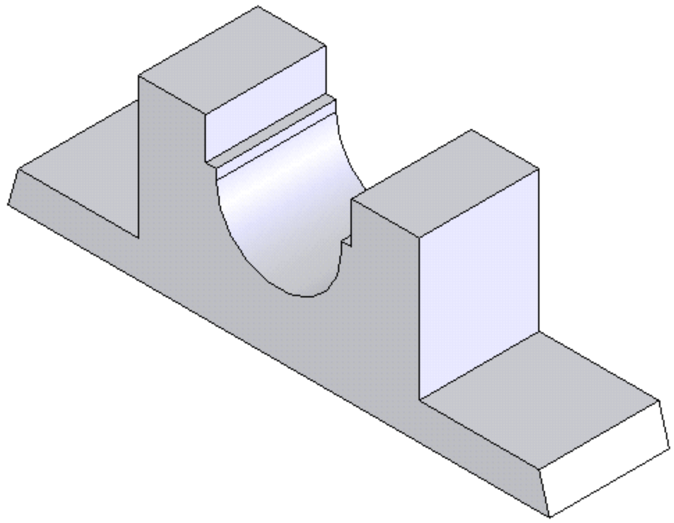
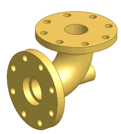


Figure 6-81 *Base feature of the model*



Tip. *Refer to Figure 6-79; you will observe that the dimension that reflects the depth of the extruded feature has a tolerance applied to it. Therefore, double-click the extruded feature in the* ***FeatureManager Design Tree*** *or in the drawing area. All the dimensions applied to the model are displayed. Select the dimension that reflects the depth of the extruded feature and apply the tolerance using the* ***Dimension PropertyManager***.

created on the top planar face of the base feature using the cut option. The cut feature will be extruded upto the selected surface.

1. Selected the top planar face of the base feature as the sketching plane and invoke the sketching environment.

2. Orient the model normal to the sketching plane and using the standard sketching tools create the sketch for the cut feature and apply the required relations to the sketch.

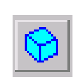
   The sketch of the second feature is shown in Figure 6-82.

3. Choose the **Extruded Cut** button from the **Features** toolbar to invoke the **Cut-Extrude PropertyManager**.

   The preview of the cut feature is displayed in the drawing area with the default values of the blind option. You need to extrude the cut feature upto the selected surface. Therefore, orient the model in the isometric view because the selection of feature termination surface is very easy in the isometric view.

4. Choose the **Isometric** button from the **Standard Views** toolbar to orient the model in the isometric view.

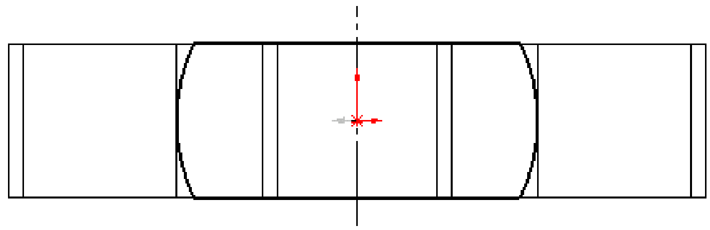


Figure 6-82 *Sketch of the second feature*

You will observe that the preview of the cut feature is inside the model, but you need to remove the outer part of the sketch profile. Therefore, you need to change the side of the cut feature.

5. Select the **Flip side to cut** check box. The preview of the cut feature is modified dynamically.

6. Right-click in the drawing area and choose the **Up To Surface** option from the shortcut menu.

   You are prompted to select a face or a surface to complete the specification of Direction 1.

7. Select the surface for feature termination shown in Figure 6-83 and right-click to end feature creation.

   The model after adding the cut feature is shown in Figure 6-84.

Creating the Rectangular Recess

The third feature to be added in the model is the rectangular recess. This rectangular recess is created as a rectangular cut feature added on the bottom face of the model.

1. Orient the model using the **Rotate** tool and select the bottom face of the base feature as the sketching plane.

2. Invoke the sketching environment and orient the model normal to the sketching plane.

3. Using the standard sketching tool create the sketch of the rectangular recess, which is in the form of a rectangle of 195 x 35 mm. Apply the required relations and dimensions to the sketch.

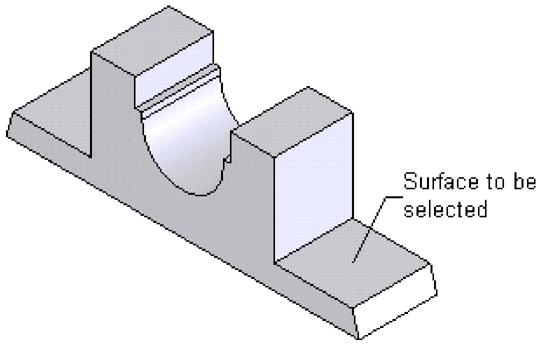


Figure 6-83 *Surface to be selected for the cut feature*

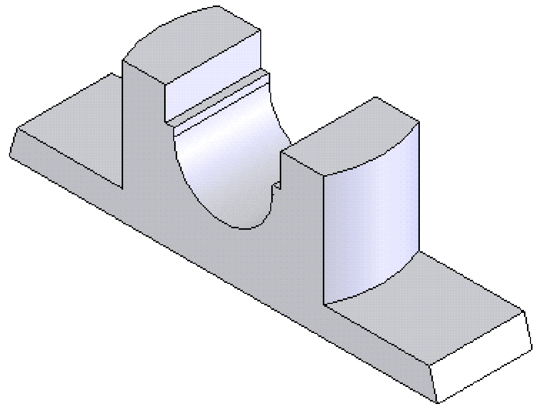


Figure 6-84 *Cut feature added to the model*

4. Choose the **Extruded Cut** button from the **Features** toolbar.

5. Set the value of the **Depth** spinner to **2** and choose **OK** from the **Cut-Extrude PropertyManager**.

Creating the Recess for the Head of Square Head Bolt

It is clear from Figure 6-79 that the bolt to be inserted in the part will be a square head bolt. Therefore, you need to create the recess for the head of the bolt. It will be created by extruding the square sketch to a given distance using the cut option.

1. Rotate the model and select the upper face of the recess as the sketching plane. Invoke the sketching environment and orient the model normal to the sketching plane.

2. Using the standard sketching tools create the sketch. The sketch includes two squares of length 26 and the distance between the centers of the squares is 78. Refer to Figure 6-79. Apply the required relations and dimensions.

3. Using the **Cut-Extrude PropertyManager** extrude the sketch to the given distance. For distance refer to Figure 6-79.

   The rotated model after adding this cut feature is shown in Figure 6-85.

Creating the Hole Feature

After creating the recess for the head of the square head bolt, you need to create the hole using the hole tool.

For creating a hole using the hole tool, you first need to create a sketched point. The sketched point will be created on the placement plane of the hole feature. After creating the sketched hole select the hole placement plane.

1. Select the right top planar surface of the base feature as the sketch plane.

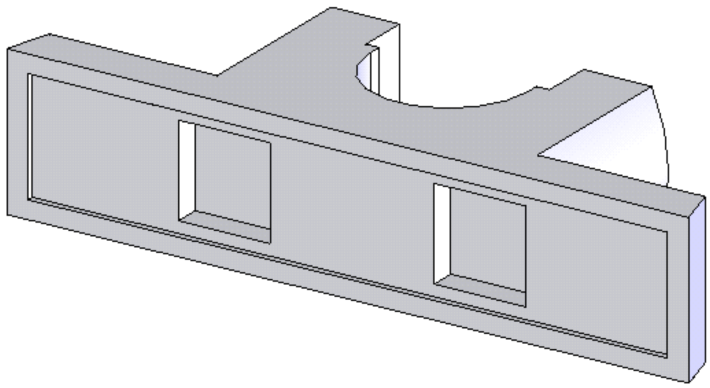
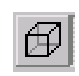


Figure 6-85 *Cut feature added to the model*

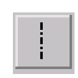
2. Create two sketched points on the right and the left top planar faces of the base feature.

   Next, you need to define the placement of the sketched point. It is clear from Figure 6-79 that the centerpoint hole is coincident with the center point of the square recess feature. Therefore, you will coincide the sketched points with the center of the squares. First you will have to change the model view display from **Shaded** to **Hidden Lines Visible** so that the square recess feature is visible.

3. Choose the **Hidden Lines Visible** button from the **View** toolbar.



4. Choose the **Centerline** button from the **Sketch Tools** toolbar and move the cursor to the lower left corner of the square. When the cursor is displayed in yellow color, specify the startpoint of the line. Move the cursor to the upper right corner of the square and when the cursor is displayed in yellow color, specify the endpoint of the line.



5. Right-click in the drawing area and choose the **Select** option from the shortcut menu. This will end line creation and also exit the **Line** tool. A centerline is created as a diagonal of the square.

6. Press and hold down the CTRL key from the keyboard and select the centerline created previously. Now select the left sketched point.

   The **Properties PropertyManager** is displayed. Using this you will add the relation between the centerpoint of the circle and the centerline. Since the centerline is created as the diagonal of the square and it is a general geometric fundamental that the midpoint of the diagonal of a square or a rectangle lies at the center, therefore, you will add a relation such that the centerpoint of the circle will be placed at the midpoint of the diagonal.

7. Choose the **Midpoint** button from the **Add Relations** rollout of the **Properties PropertyManager**. Choose the **OK** button from the **Properties PropertyManager**.

8. Similarly, create the diagonal centerline coincident to the right square. Add the midpoint relation between the centerline and the sketched point.

9. Exit the sketcher environment and change the model display mode to shaded. Select the right top planar surface of the base feature as the placement plane for the hole feature.

   As you select the placement plane, the **Simple Hole** option is enabled in the menu bar.

10. Choose **Insert** > **Features** >**Hole** > **Simple Hole** from the menu bar to invoke the **Hole PropertyManager**.

    As you invoke the **Hole PropertyManager** the preview of the hole feature with default settings is displayed in the drawing area. The confirmation corner is also displayed.

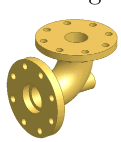
11. Right-click in the drawing area and choose the **Through All** option from the shortcut menu to define feature termination.

12. Set the value of the **Hole Diameter** spinner to **12** and choose the **OK** button from the **Hole PropertyManager**.

13. Select the centerpoint of the hole and drag the cursor to the right sketched point. Release the left mouse button when the cursor turns yellow in color.

14. Using the same procedure create the second hole feature on the left side of the model. The model after creating the hole features is displayed in Figure 6-86.

15. The next feature of the model is a cut feature. To create this feature you will have to create the sketch on the second top planar face of the base feature. The model after adding the cut feature is shown in Figure 6-87.



Tip. *Refer to Figure 6-79; you will find that the dimension of the cut feature is given from the center of the sketch. For creating this dimension create a diagonal centerline between the right endpoint of the lower horizontal line and the left endpoint of the upper horizontal line. Create a sketched point and apply the midpoint relation between the centerline and the sketched point. Using this sketched point you can dimension the sketch from the center of the dimension.*

Creating the Fillet Feature

After creating all the other features, you will now add the fillet feature to the model.

1. Choose the **Fillet** button from the **Features** toolbar to invoke the **Fillet PropertyManager**.

   After invoking the **Fillet PropertyManager** you are prompted to selected edges, faces,

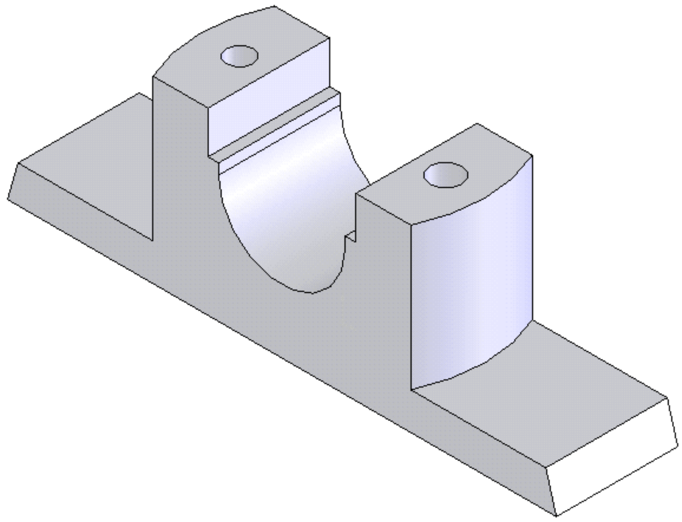


Figure 6-86 *Model after adding the hole features*

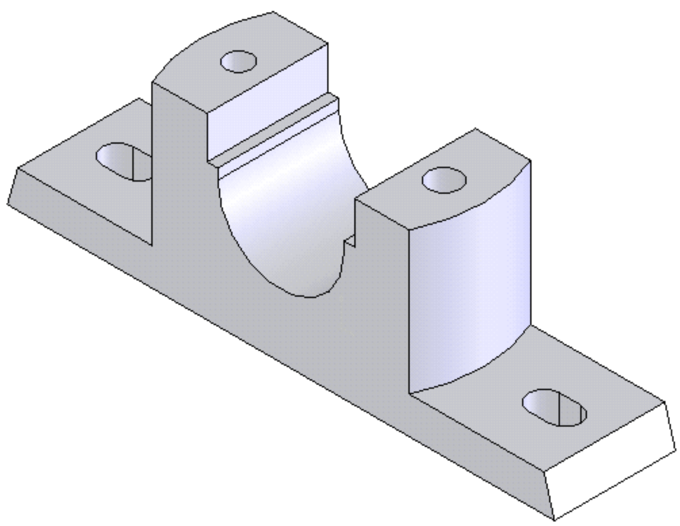


Figure 6-87 *Model after adding the cut feature*

features, or loops to fillet. As evident from the model you will select only the edges to apply the fillet feature.

2. Using the select tool select the edges shown in Figure 6-88.

As soon as you select the edges the preview of the fillet feature with the default values is shown in the drawing area. A radius callout is also displayed along the selected edge. Now, you need to modify the default value of the fillet feature.

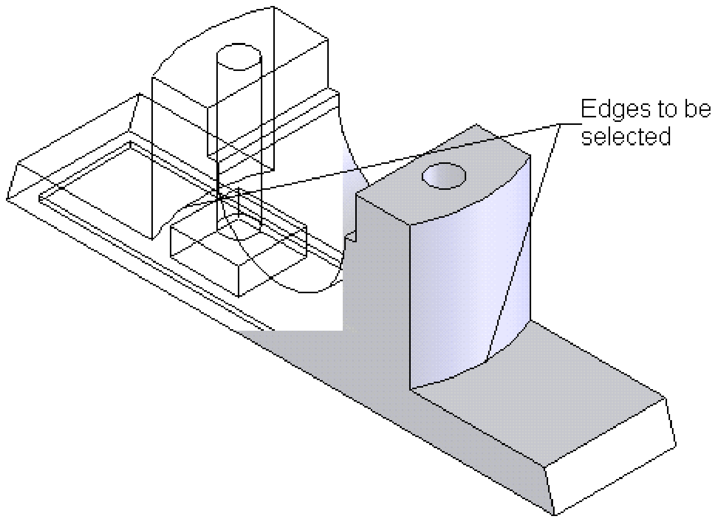
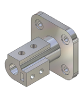


Figure 6-88 Edges to be selected for the fillet feature



Note

*If the preview of the fillet feature is not displayed in the drawing area then select the **Full Preview** radio button available in the **Item To Fillet** rollout of the **Fillet PropertyManager**.*

3. Set the value of the **Radius** spinner to **8** and choose the **OK** button from the **Fillet PropertyManager**.

   The model after adding the fillet feature is shown in Figure 6-89.

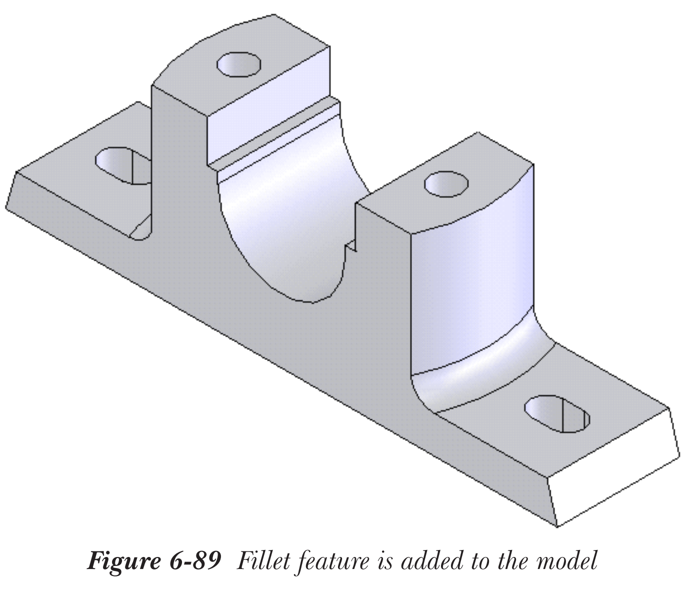
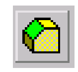


Figure 6-89 Fillet feature is added to the model

Creating the Chamfer Feature

The next feature of this model is the chamfer feature.

1. Choose the **Chamfer** button from the **Features** toolbar to invoke the **Chamfer PropertyManager**.



2. Using the select tool select the edges of the cut features as shown in Figure 6-90. The edges that are tangent to the selected edges are selected automatically because the selection mode of the chamfer feature uses tangent selection.

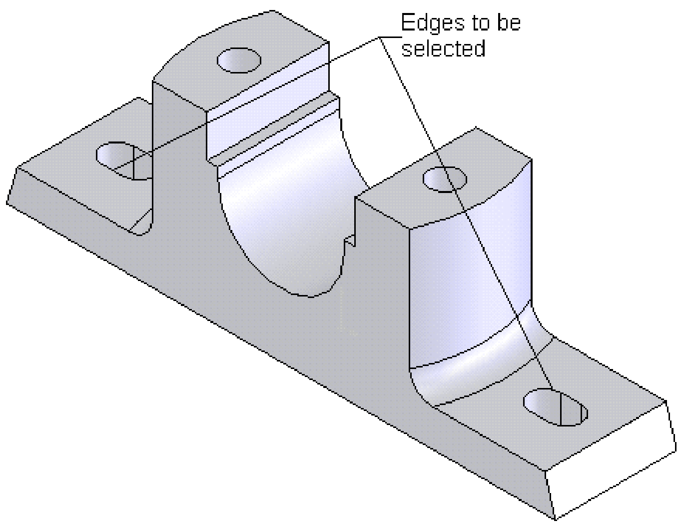


Figure 6-90 *Edges to be selected*

As soon as you select the edges the preview of the chamfer feature is displayed in the drawing area with the default values. The angle and distance callout is also displayed. Now, you will set the required value of chamfer.

The required chamfer parameters are 1mm and 45-degree. The **Angle distance** radio button is selected by default in the **Chamfer Parameters** rollout. The value of angle in the **Angle** spinner is set as 45-degree, therefore, you do not need to modify this value. You will set only the value of the distance in the **Distance** spinner.

3. Set the value of the distance in the **Distance** spinner to **1** and choose the **OK** button from the **Chamfer PropertyManager**.

Following the same procedure add the chamfer feature to the semicircular edge of the base feature. Refer to Figure 6-79 for the parameters of the chamfer feature.

The final solid model is shown in Figure 6-91. The **FeatureManager Design Tree** of the model is shown in Figure 6-92.

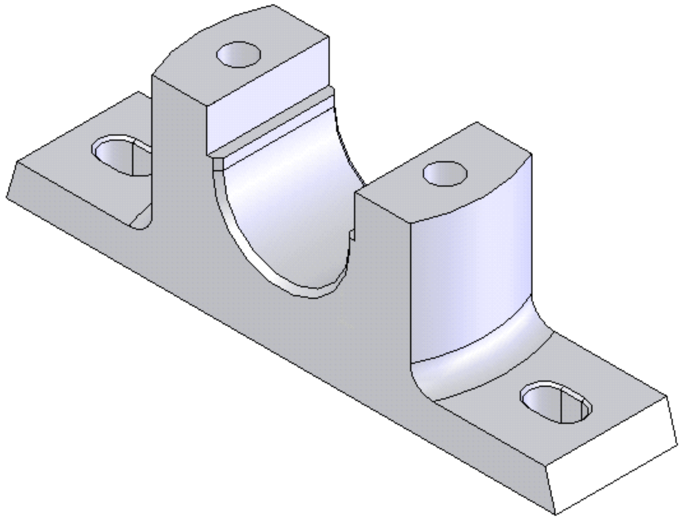


Figure 6-91 *Final solid model*

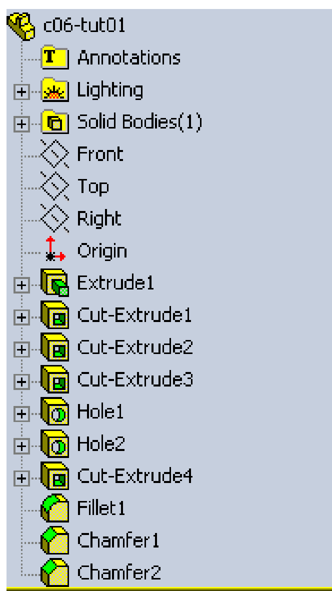
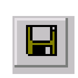


Figure 6-92 *The **FeatureManager Design Tree** of the model*

Saving the Model

Next, you need to save the model.

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



\My Documents\SolidWorks\c06\c06-tut01.SLDPRT.

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

Tutorial 2

In this tutorial you will create the model shown in Figure 6-93. For better understanding of the model the section view of the model is shown in Figure 6-94. The dimensions of model are shown in Figure 6-95. **(Expected time: 30 min)**

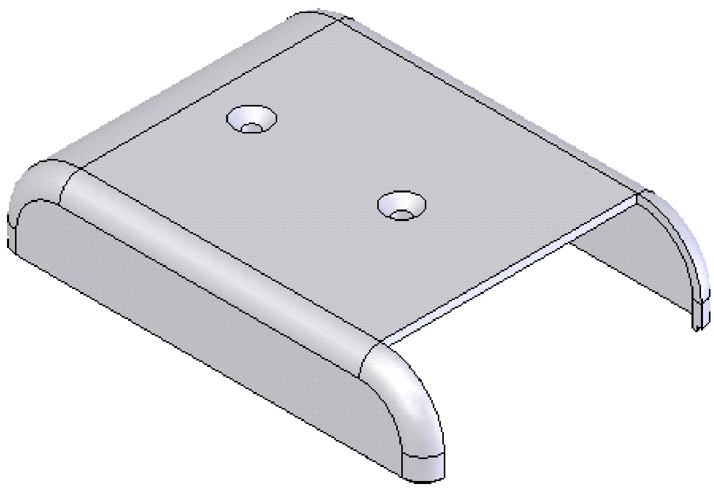


Figure 6-93 *Solid model for Tutorial 2*

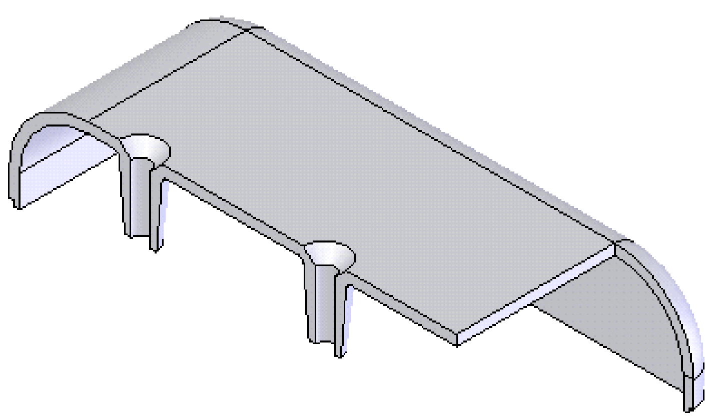


Figure 6-94 *Section view of the model*

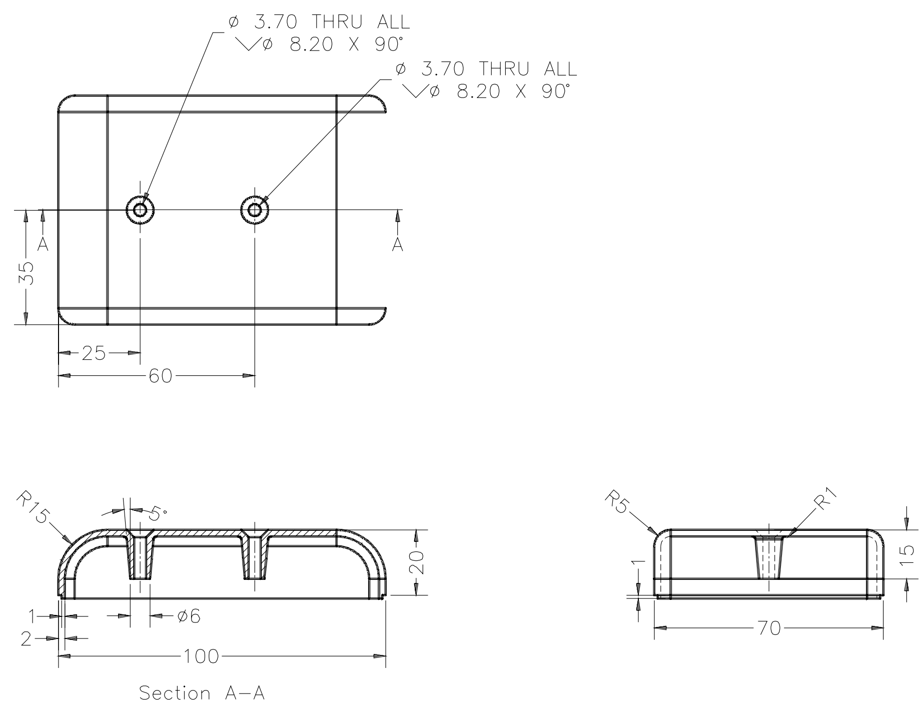


Figure 6-95 *Top view, Front section view, and right-side view with dimensions*

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

a. Create the base feature of the model by extruding a rectangle of 100x70mm to a distance of 20mm.
b. Add the fillet features to the base feature, refer to Figures 6-96 through 6-99.
c. Add the shell feature to create a thin walled part and remove some of the faces, refer to Figures 6-100 and 6-101.
d. Create a reference plane at an offset distance from the top planar face of the base feature and extrude the sketch created on the new plane, refer to Figure 6-102.
e. Using the **Hole Wizard** add the countersink hole feature to the model, refer to Figure 6-103.
f. Add the fillet feature to the extruded feature, refer to Figures 6-104 and 6-105.
g. Create the lip of the component by extruding the sketch, refer to Figure 6-106.

Creating the Base Feature

Create a new SolidWorks part document. First you need to create the base feature of the model. The base feature of the model will be created by extruding a rectangle of 100x70mm to a distance of 20mm. It is clear from the model that the sketch of the base feature will be created on the top plane. Therefore, you have to select the top plane as the sketching plane.

1. Select the **Top** plane from the **FeatureManager Design Tree** and invoke the sketching environment.

2. Orient the view normal to eye view.

3. Using the standard sketching tools create the sketch of the base feature and add the required relations and dimensions to the sketch.

4. Choose the **Extruded Boss/Base** button from the **Features** toolbar and extrude the base feature to a depth of 20mm.

Creating the Fillet Features

After creating the base feature, you need to add the fillet features to the model. In this model you will add three fillet features. Two fillet features will be added at this stage of the design process and the remaining one fillet feature will be added at a later stage of the design process.

1. Choose the **Fillet** button from the **Features** toolbar to invoke the **Fillet PropertyManager**.

2. Select the edges of the model as shown in Figure 6-96.

   As soon as you select the edges of the model, the preview of the fillet with the default values and the radius callout are displayed in the drawing area.

3. Set the value of the **Radius** spinner to **15** and choose the **OK** button from the **Fillet PropertyManager**.

   Figure 6-97 shows the model after adding the first fillet feature.

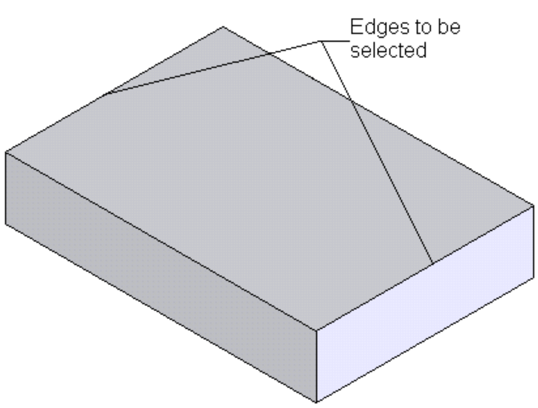


***Figure 6-96** Edges to be selected*

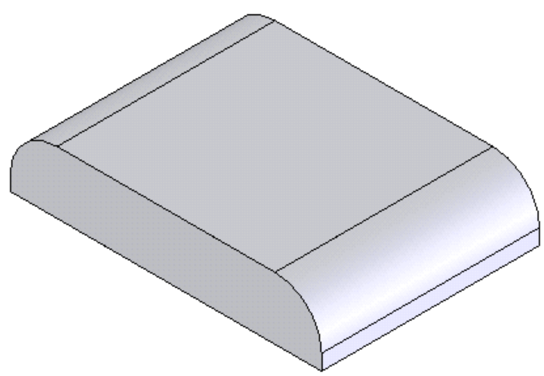


***Figure 6-97** Fillet added to the base feature*

   Now, you will add the second fillet feature to the model.

4. Invoke the **Fillet PropertyManager** and set the value of the **Radius** spinner to **5**.

5. Select the edges of the model as shown in Figure 6-98 and right-click to complete feature creation.

   The model after adding the second fillet feature is shown in Figure 6-99.

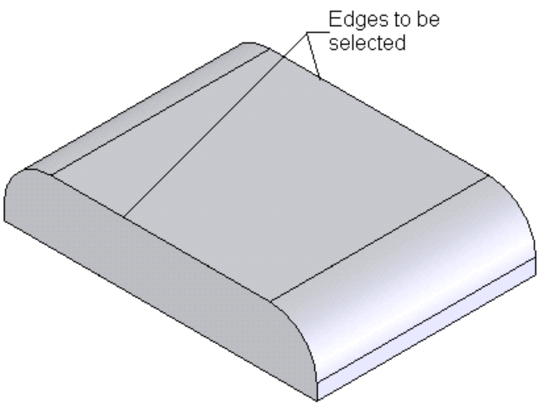


Figure 6-98 *Edges to be selected*

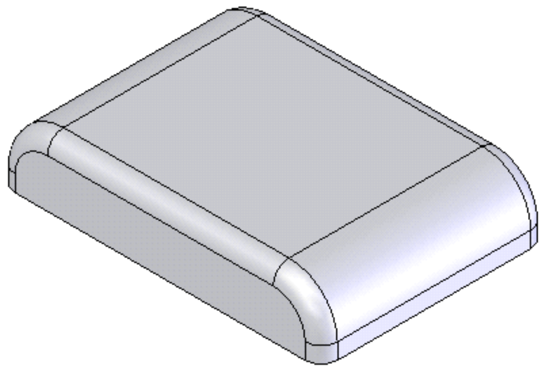
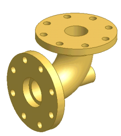


Figure 6-99 *Second fillet added to the model*



Tip. *The edges tangent to the selected edges are filleted automatically because by default, the* ***Tangent propagation*** *check box is selected in the* ***Fillet PropertyManager****.*

Creating the Shell Feature

It is clear from Figures 6-93 and 6-94 that this model will need a shell feature. Therefore, now you will add the shell feature to the model. As discussed earlier, the shell feature is used to scoop out the material from the model, leaving behind a thin-walled hollow part.

1. Choose the **Shell** button from the **Features** toolbar to invoke the **Shell PropertyManager**. The confirmation corner is also displayed.

   You are prompted to select the faces to remove.

2. Rotate the model and select the faces to remove as shown in Figure 6-100. The names of the selected faces are displayed in the **Faces to Remove** display area.

3. Set the value of the **Thickness** spinner to **2** and choose the **OK** button from the **Shell PropertyManager**.

   The model after adding the shell feature is shown in Figure 6-101.

Creating the Extruded Feature

The next feature that you are going to create is an extruded feature. But, before creating this extruded feature, you need to create a reference plane at an offset distance from the top planar face of the base feature.

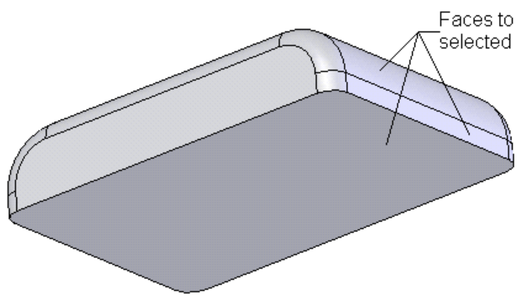


Figure 6-100 Face to be selected to remove

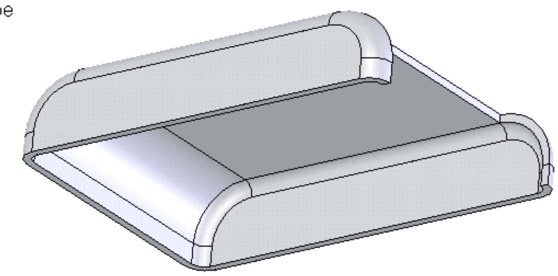


Figure 6-101 Shell feature added to the model

1. Invoke the **Plane PropertyManager** and create a plane at an offset distance of 15 from the top planar face of the base feature. You have to select the **Reverse direction** check box from the **Plane PropertyManager** to create the plane.

2. Select the newly created plane as the sketching plane and create the sketch using the standard sketching tools. The sketch consists of two circles of diameter 6. For other dimensions refer to Figure 6-85.

3. Extrude the sketch using the **Up To Next** option and add an outward draft of 5-degree. Hide the newly created plane.

   Figure 6-102 shows the rotated model after adding the extruded feature.

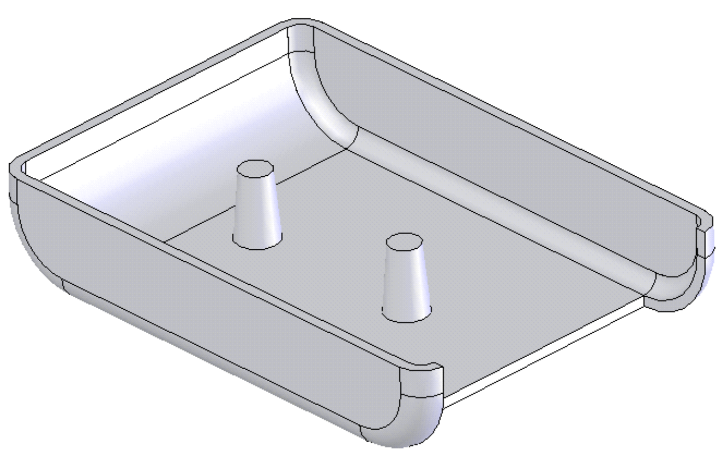
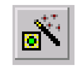


Figure 6-102 Model after adding the extruded feature

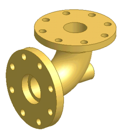
Adding the Countersink Hole using the Hole Wizard

The next feature that you are going to create is a countersink hole, refer to Figure 6-85. You will observe that the hole callout displayed for countersink is generally used to accommodate the **M3.5 Flat Head Machine Screw**. In SolidWorks you are provided with one of the largest standard hole generating technique known as **Hole Wizard**. Therefore, using the **Hole Wizard** you will add the standard holes to the model that can accommodate standard fasteners.

1. Rotate the model and select the top planar face of the base feature and then choose the **Hole Wizard** button from the **Features** toolbar.



The **Hole Definition** dialog box is displayed and the preview of the hole feature on the selected face with the default values is shown in the drawing area.



Tip. *If you select the placement plane for the placement of the hole feature before invoking the* ***Hole Definition*** *dialog box then the placement sketch will be a 2D sketch. If you select the placement plane after invoking the* ***Hole Definition*** *dialog box then the hole placement sketch will be a 3D sketch. You will learn more about 3D sketches in the later chapters.*

2. Choose the **Countersink** tab of the **Hole Definition** area.

Now, you will set the parameters to define the standard hole. The preview of the standard hole will be modified dynamically as you set the parameters for the hole feature.

3. Choose the **Ansi Metric** option from the **Standard** drop-down list.

4. Choose the **Flat Head** option from the **Screw Type** drop-down list

5. Choose the **M3.5** option from the **Size** drop-down list and choose the **Through All** option from the **End Condition** drop-down list.

6. Choose the **Next** button from the **Hole Definition** dialog box. The **Hole Placement** dialog box is displayed and the sketcher environment is invoked. The select cursor is replaced by the point cursor.

7. Using the point cursor specify another point anywhere on the top planar face of the base feature to create another hole.

Since both the placement points are not properly placed, you need to add the required relations or dimensions to define the placement of the placement points. For adding the relation to fully define the placement first change the model display from **Shaded** to **Hidden Lines Visible**.

8. Choose the **Hidden Lines Visible** button from the **View** toolbar to display the model with hidden lines visible.

9. Right-click in the drawing area and choose the **Select** option from the shortcut menu to invoke the select tool.

10. Right-click in the drawing area and choose the **Add Relation** option from the shortcut menu. The **Add Relations PropertyManager** is displayed.

11. Select the first sketch point and then select the first upper hidden circle. Choose the **Concentric** button from the **Add Relations** rollout.

12. Click anywhere in the drawing area and select the second sketch point and the second upper hidden circle and apply the concentric relation as done earlier. Choose the **OK** option from the confirmation corner.

13. Choose the **Finish** button from the **Hole Placement** dialog box and choose the **Shaded** button from the **View** toolbar.

    The isometric view of the model after adding the hole feature is shown in Figure 6-103.

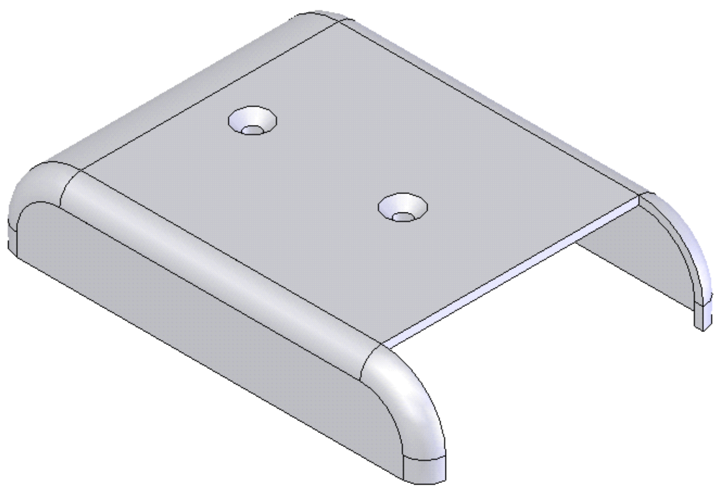


Figure 6-103 *Model after adding the hole feature using the* ***Hole Wizard***

Adding the Fillet Feature

Now, you will add a fillet feature to the edges of the extruded feature that was created earlier.

1. Rotate the model and invoke the **Fillet PropertyManager**. Select the edges of the model as shown in Figure 6-104.

2. Set the value of the **Radius** spinner to **1** and choose the **OK** button to end feature creation.

   The model after adding the fillet feature is shown in Figure 6-105.

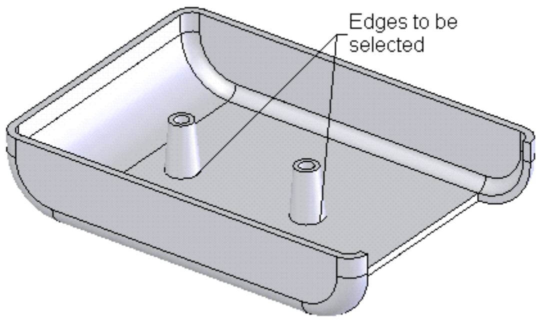


Figure 6-104 Edges to be selected

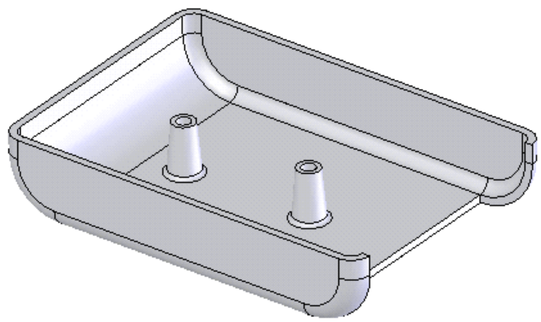


Figure 6-105 Fillet feature added to the model

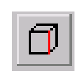
Adding a Lip to the Model

The last feature that you will add to the model is a lip. It will be created by extruding an open sketch using the thin option.

1. Select the bottom face of the base feature as the sketching plane and invoke the sketching environment.

2. Using the select tool select any one of the lower inner edge of the model and right-click to display the shortcut menu. Now, choose the **Select Tangency** option from the shortcut menu.

3. Choose the **Convert Entities** button from the **Sketch Tools** toolbar.



   The selected edges are converted into sketched entities.

4. Choose the **Extruded Boss/Base** button from the **Features** toolbar. The **Thin Feature** rollout is invoked automatically because you are extruding an open profile.

5. Set the value of **Distance** spinner in the **Direction 1** rollout to **1** and set the value of the **Thickness** spinner in the **Thin Feature** rollout to **1**.

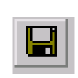
6. Choose the **OK** button from the **Extrude PropertyManager**.

The rotated view of the final model is shown in Figure 6-106. The **FeatureManager Design Tree** of the model is shown in Figure 6-107.

Saving the Model

Next, you need to save the model.

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



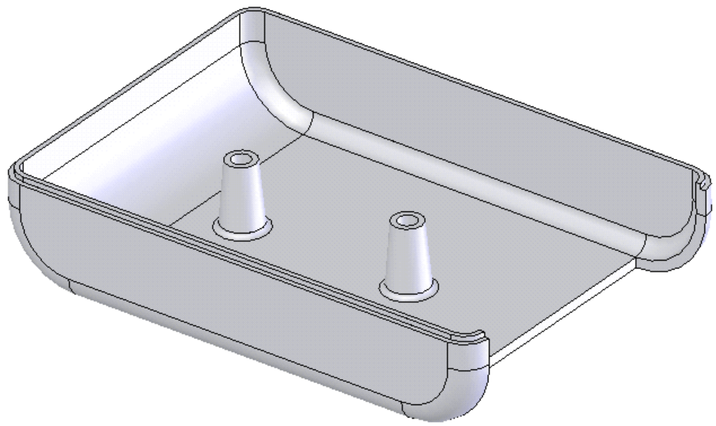


Figure 6-106 *Final model rotated to display the maximum features*

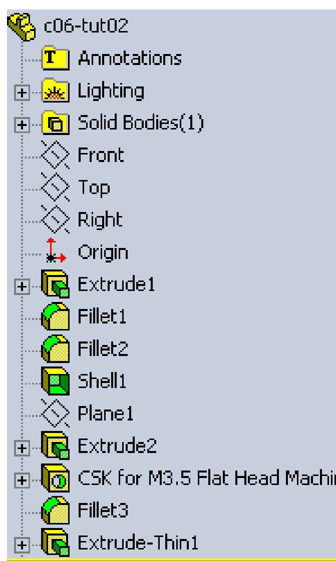


Figure 6-107 *The **FeatureManager Design Tree** of the model*

\My Documents\SolidWorks\c06\c06-tut02.SLDPRT.

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

SELF-EVALUATION TEST

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

1. Using the **Hole PropertyManager** you can create counterbore, countersink, and tapped holes. (T/F)

2. The hole features created using the **Hole Wizard** and the **Hole PropertyManager** are not parametric. (T/F)

3. You cannot define a user-defined hole using the **Hole Wizard**. (T/F)

4. You cannot preselect the edges or faces for creating a fillet feature. (T/F)

5. In SolidWorks you can create a multi-thickness shell feature. (T/F)

6. The __________ check box is selected to create the shell feature on the outer side of the model.

7. The __________ is created by specifying different radii along the length of selected edge at specified intervals.

8. The names of the faces to be removed in the shell features are displayed in the __________ display area.

9. If you want to specify different distances while creating the chamfer then clear the __________ check box.

10. The __________ check box is selected to apply the face fillet feature with continuous curvature throughout the fillet feature.

REVIEW QUESTIONS

Answer the following questions:

1. The __________ option is used to add the standard holes to the model.

2. After specifying all the parameters to a hole feature using the **Hole Definition** dialog box, the __________ dialog box is displayed to specify the placement of the hole feature.

3. The __________ radio button is selected to create a smooth transition by smoothly blending the points and vertices on which you have defined the radius on the selected edge.

4. The __________ tab available in the **Hole Definition** dialog box is used to define the parameters to add a standard drilled hole.

5. By default, the __________ radio button is selected in the **Chamfer PropertyManager**.

6. If you preselect the placement surface for creating a hole feature using the **Hole Wizard** then the resultant placement sketch will be a

   (a) 2D sketch (b) Planar sketch
   (c) Basier spline (d) 3D sketch

7. Using which options you do not use radius while creating a fillet feature?

   (a) Face fillet with hold line (b) Constant radius fillet
   (c) Variable radius fillet (d) Full round fillet

8. Which radio button from the **Variable Radius Parameters** rollout is used to create smooth transition while creating a variable radius fillet?

   (a) **Straight transition** (b) **Parametric transition**
   (c) **Smooth transition** (d) **Surface transition**

9. If you do not select any face to be removed while creating a shell feature, then what will be the resultant model?

   (a) Remains complete solid model (b) Thin walled hollow model
   (c) Automatically removed one face (d) None of these

10. Which dialog box is displayed when you choose the **Hole Wizard** button from the **Features** toolbar?

    (a) **Hole** (b) **Hole Definition**
    (c) **Hole Wizard** (d) **Hole Parameters**

EXERCISES

Exercise 1

Create the model shown in Figure 6-108. The dimensions of the model are shown in Figure 6-109. **(Expected time: 30 min)**

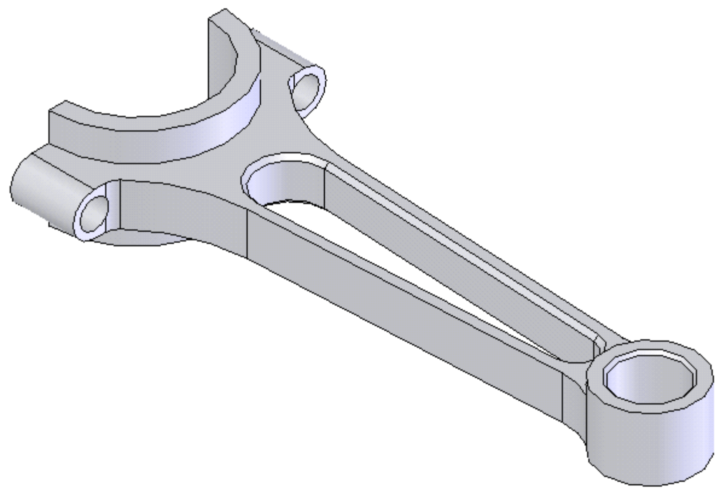


Figure 6-108 *Solid model for Exercise 1*

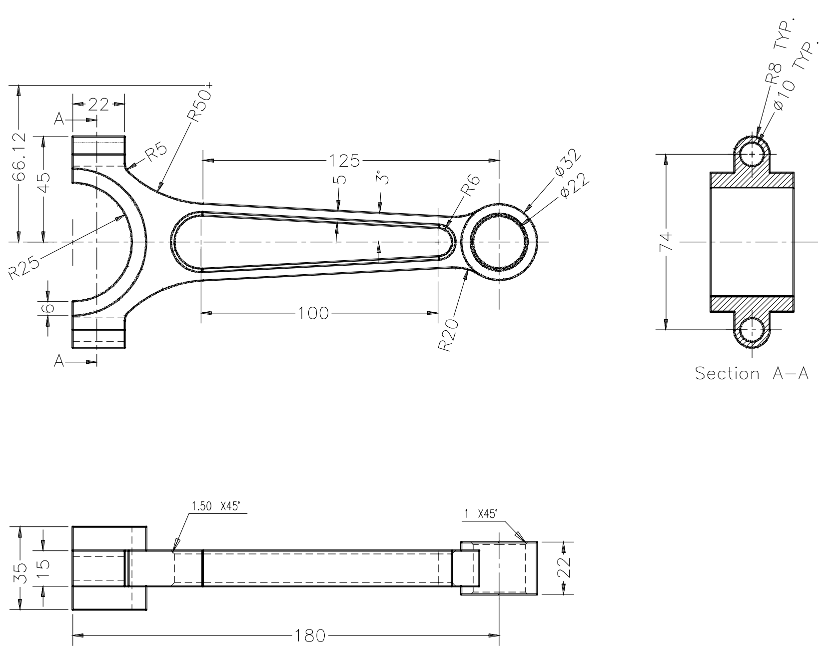


Figure 6-109 *Views and dimensions of the model for Exercise 1*

Exercise 2

Create the model shown in Figure 6-110. The dimensions of the model are shown in Figure 6-111. **(Expected time: 30 min)**

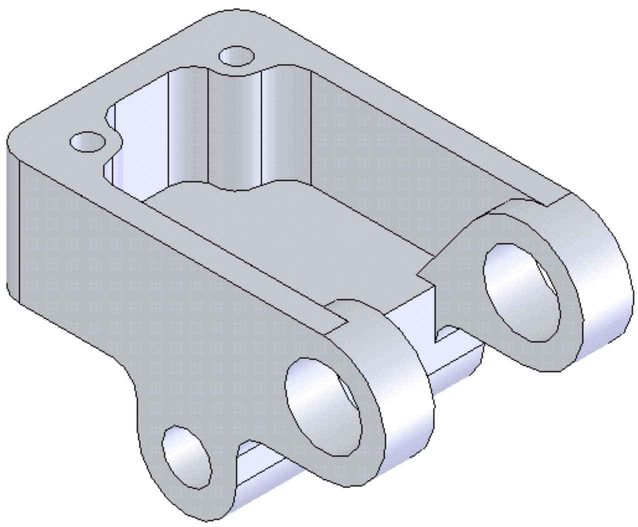


Figure 6-110 *Solid model for Exercise 2*

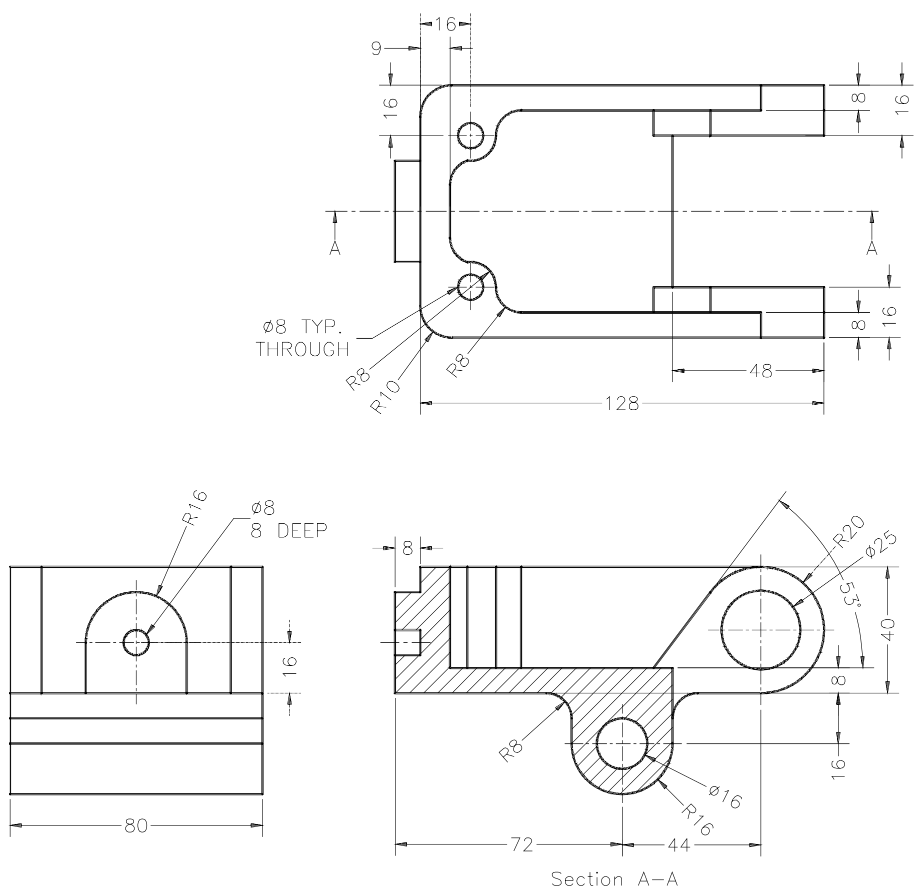


Figure 6-111 Views and dimension of the model for Exercise 2

Answers to Self-Evaluation Test

1. T, **2.** F, **3.** F, **4.** F, **5.** T, **6. Shell outward**, **7.** Variable radius fillet, **8. Faces to Remove**, **9. Equal distance**, **10. Curvature continuous**