



Chapter 6

Creating Dress-Up and Hole Features

Learning Objectives

After completing this chapter, you will be able to:

- *Create holes using the Hole tool.*
- *Create fillet features.*
- *Create chamfer features.*
- *Add a draft to the faces of the models.*
- *Create a shell feature.*

ADVANCED MODELING TOOLS

In this chapter, you will learn to create some of the placed features that aid in constructing a model. For example, in the previous chapter, you learned to create holes by extruding a circular sketch using the **Pocket** tool. In this chapter, you will learn to create holes using the **Hole** tool. You will also learn some other advanced modeling tools such as fillets, chamfer, draft, shell, and so on.

Creating Hole Features

Menu: Insert > Sketch-Based Features > Hole
Toolbar: Sketch-Based Features > Hole



You can create a simple hole, tapered hole, counterbored hole, countersunk hole, and a counterdrilled hole using the **Hole** tool. You can also provide threads in the holes. However, you can create only one hole feature at a time using this tool. To create a hole, choose the **Hole** button from the **Sketch-Based Features** toolbar; you are prompted to select a face or plane. Select the face or plane from the geometry area on which you need to place the hole; the preview of the hole feature is displayed, along with the **Hole Definition** dialog box. The **Hole Definition** dialog box is shown in Figure 6-1.

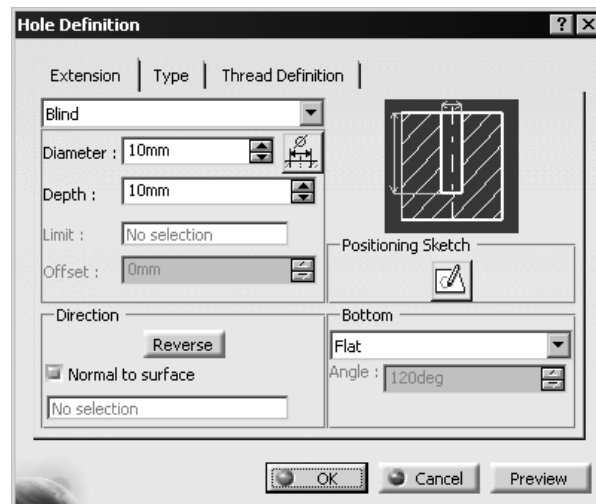


Figure 6-1 The **Hole Definition** dialog box

Creating a Simple Hole

Choose the **Type** tab; the **Simple** option is selected in the drop-down list. Therefore, a simple hole will be created using the current option. Now, choose the **Extension** tab. Next, you need to position the center point of the sketch. Choose the **Sketcher** button in the **Positioning Sketch** area; the **Sketcher** workbench is invoked. The center point of the hole is displayed as a sketched point. Locate the point using the **Constraint** tool and exit the **Sketcher** workbench. Next, set the feature termination condition and the diameter of the hole using the options in the **Extension** tab. You can also reverse the direction of the feature creation using the **Reverse**

button in the **Direction** area. By default, the **Normal to surface** check box is selected. You can also create a hole along a specified direction by clearing the **Normal to surface** check box and selecting the direction along which you need to create it.

The drop-down list in the **Bottom** area is used to specify the shape at the end of the hole. It will not be available if you select the **Up To Next** or **Up To Last** termination types. For other termination types, if you select the **Flat** option, the resulting hole will be flat at the bottom. If you select the **V-Bottom** option, the bottom of the resulting hole will be of V shape. You can set the angle of the V-shape using the **Angle** spinner. For the **Up To Next** or **Up To Last** termination types, the **Trimmed** bottom is created. Figure 6-2 shows all the three types of bottom termination options.

After setting the hole parameters, choose the **OK** button from the **Hole Definition** dialog box to create a simple hole. Figure 6-3 shows a base plate after creating the simple holes using the **Hole** tool.

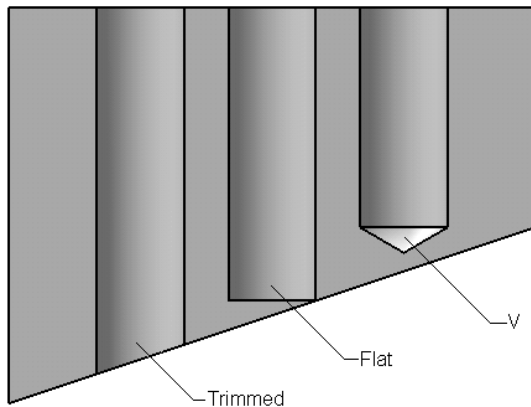


Figure 6-2 Types of bottom termination options for a hole feature

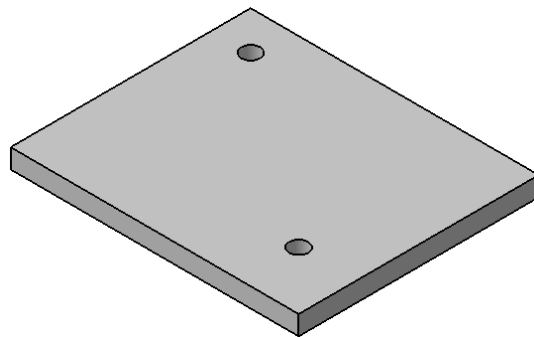


Figure 6-3 Base plate with holes created using the **Hole** tool

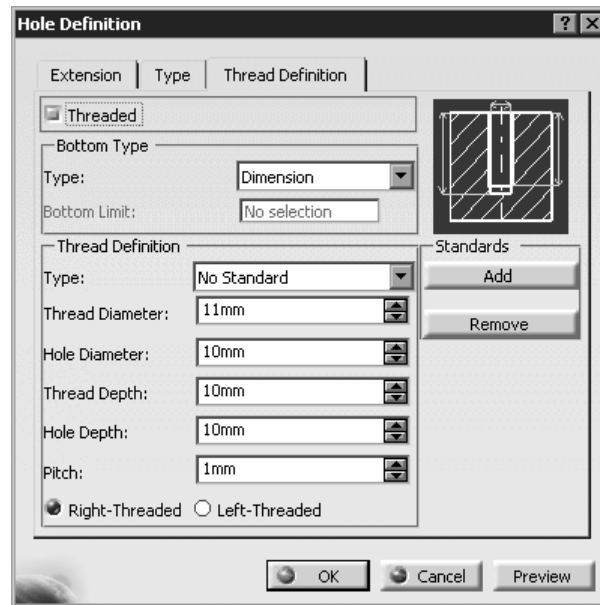


Tip. While creating a hole using the **Hole** tool, you can also apply a hole callout to display the hole tolerance. Choose the **Limit of size Definition** button from the **Extension** tab of the **Hole Definition** dialog box; the **Limit of Size Definition** dialog box is displayed. The preview of the hole tolerance callout is also displayed on the hole feature in the geometry area. Set the value of the hole tolerance using the options in the **Limit of Size Definition** dialog box and choose the **OK** button. Now, set the parameters of the hole and exit the **Hole Definition** dialog box to complete the feature creation. The annotation set is displayed in the specification tree. The information about the hole tolerance callout is displayed in it.



Creating a Threaded Hole

To create the threaded hole, choose the **Thread Definition** tab from the **Hole Definition** dialog box. By default, the **Threaded** check box is cleared. Select the **Threaded** check box to invoke the options in the **Thread Definition** area, as shown in Figure 6-4.



*Figure 6-4 The **Hole Definition** dialog box after selecting the **Threaded** radio button*

The **Bottom Type** area in this dialog box is used to select the options to specify the thread depth. By default, the **Dimension** option is selected in the **Type** drop-down list of the **Bottom Type** area. This option allows you to specify the desired thread depth in numerical values. To specify the thread depth, use the **Thread Depth** spinner in the **Thread Definition** area. The **Support Depth** option in the **Type** drop-down list of the **Bottom Type** area sets the thread depth equal to the hole depth. The **Up-To-Plane** option in the **Type** drop-down list allows you to specify the thread depth by selecting a termination plane or surface up to which the thread will be created. As soon as you select the **Up-To-Plane** option, the **Bottom Limit** display box will become active and you will be prompted to select the termination plane or the surface. Select the plane or the surface from the drawing area; the thread in the hole will terminate at the specified plane.

By default, the **No Standard** option is selected in the **Type** drop-down list in the **Thread Definition** area. Therefore, you need to manually specify the parameters to define the thread. Set the value of the thread diameter in the **Thread Diameter** spinner and the value of the hole diameter in the **Hole Diameter** spinner. By default, these values are based on the diameter value specified in the extension tab. Set the thread depth and the hole depth in the **Thread Depth** and the **Hole Depth** spinners, respectively. Also, set the pitch value in the **Pitch** spinner. By default, the **Right-Threaded** radio button is selected. To create a left hand thread, select the **Left-Threaded** radio button. After setting the parameters, choose the **OK** button from the **Hole Definition** dialog box; a threaded hole will be created. Note that the thread will not be displayed in the hole because only a cosmetic thread is added to the hole feature. When you generate the drawing view, the thread convention will be displayed in it. You will learn more about generating drawing views in the later chapters.

To create standard threaded holes, select the **Metric Thin Pitch** or **Metric Thick Pitch** option from the **Type** drop-down list in the **Thread Definition** area. You can select the thread standard from the **Thread Description** drop-down list. In this case, you need to specify the thread and the hole depth. The hole diameter, thread diameter, and thread pitch are automatically defined on the basis of the selected standard.



Tip. You can also add user-defined thread standards for creating a threaded hole by choosing the **Add** button from the **Standards** area. The **File Selection** dialog box is displayed. Select the text file, in which the thread standards are saved and choose the **Open** button from the **File Selection** dialog box. Now, select the name of the text file from the **Type** drop-down list in the **Thread Definition** area. To remove a user defined standard, choose the **Remove** button from the **Standards** area; the **Standard Threads** dialog box is displayed. Select the standard to be removed and choose the **OK** button.

Creating a Tapered Hole

To create a tapered hole, choose the **Type** tab of the **Hole Definition** dialog box and select the **Tapered** option from the drop-down list, as shown in Figure 6-5. The preview of the tapered hole is displayed in the geometry area with the default values. Specify the taper angle in the **Angle** spinner in the **Parameters** area, as shown in Figure 6-5.

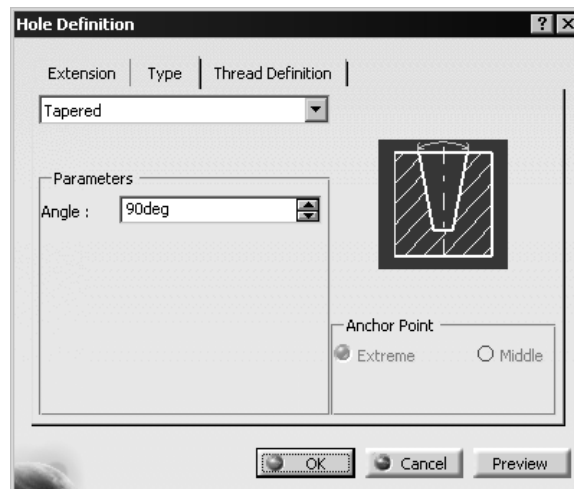


Figure 6-5 The **Hole Definition** dialog box after selecting the **Tapered** option from the drop-down list in the **Type** tab

Note that you cannot define the thread parameters for a tapered hole. After setting all parameters, choose the **OK** button from the **Hole Definition** dialog box to create the tapered hole.

Creating a Counterbored Hole

A counterbore hole is a stepped hole and has two diameters, a bigger diameter and a smaller diameter. The bigger diameter is called the counterbore diameter and the smaller diameter is called the hole diameter. This hole type requires you to specify two depths, counterbore

depth and hole depth. The counterbore depth is the depth up to which the bigger diameter will be defined. The hole depth is the total depth of the hole, including the counterbore depth. Figure 6-6 shows the sectional view of a counterbore hole. Figure 6-7 shows a base plate with the counterbored holes.

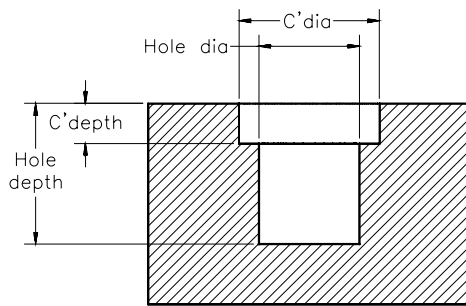


Figure 6-6 The sectional view of a counterbored hole

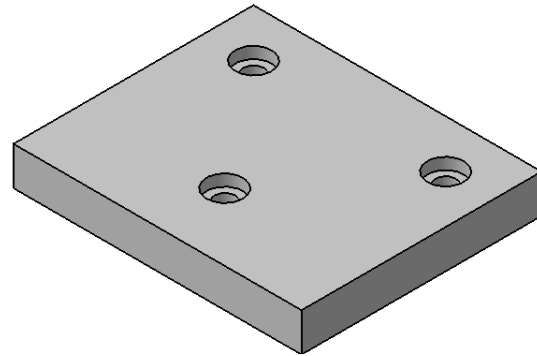


Figure 6-7 The base plate after adding the counterbored holes

To create a counterbored hole, select the **Counterbored** option from the drop-down list in the **Type** tab of the **Hole Definition** dialog box, as shown in Figure 6-8. The preview of the counterbored hole is displayed in the geometry area. You can set the value of the counter diameter using the **Diameter** spinner in the **Parameters** area. Set the value of the counter depth using the **Depth** spinner. You can set the diameter and depth of the hole using the options in the **Extension** tab.

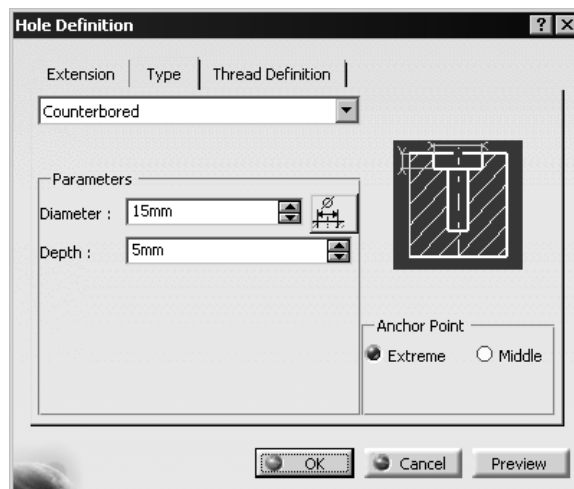


Figure 6-8 The **Hole Definition** dialog box after selecting the **Counterbored** option from the drop-down list in the **Type** tab

You will notice that the **Extreme** radio button is selected in the **Anchor Point** area of the **Type** tab. If you select the **Middle** radio button, the bottom face of the counter will be placed

on the selected placement plane. This is also the top face of the bore of the hole. You can also define the thread parameters for a counterbored hole. After setting the parameters, choose the **OK** button from the **Hole Definition** dialog box.

Creating a Countersunk Hole

A countersunk hole also has two diameters, but the transition between the bigger diameter and the smaller diameter is in the form of a tapered cone. In this hole type, you need to define the countersunk diameter, hole diameter, depth of the hole, and the countersink angle. Figure 6-9 shows the sectional view of a countersunk hole. Figure 6-10 shows the spacer plate after adding the countersunk holes.

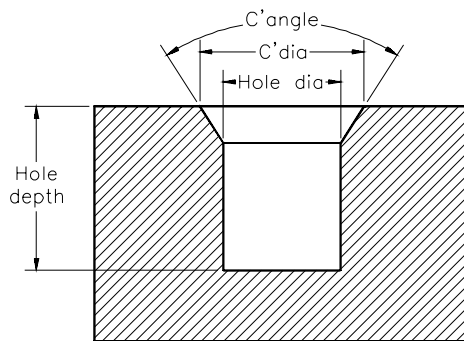


Figure 6-9 The sectional view of the countersunk hole

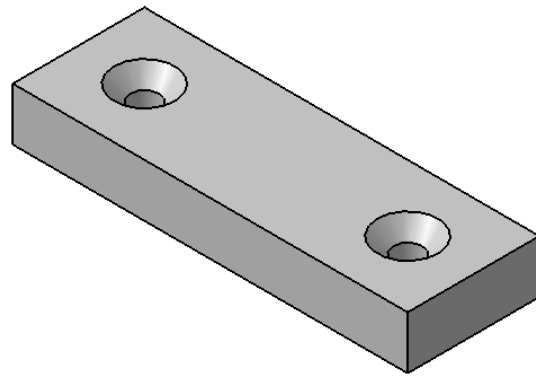


Figure 6-10 Spacer plate after adding the countersunk holes

To create a countersunk hole, select the **Countersunk** option from the drop-down list in the **Type** tab, as shown in Figure 6-11. Its preview is displayed in the geometry area.

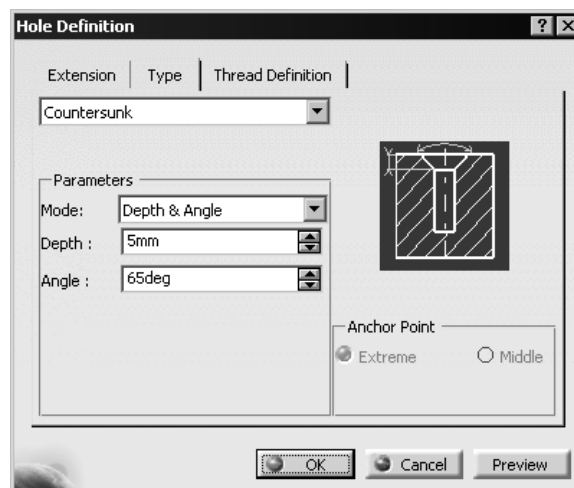


Figure 6-11 The **Hole Definition** dialog box after selecting the **Countersunk** option from the drop-down list in the **Type** tab

You can choose the option for specifying the parameters of the countersunk using the **Mode** drop-down list. By default, the **Depth & Angle** option is selected. Therefore, you need to define the depth and angle of the countersunk in the **Depth** and **Angle** spinners, respectively. If you select the **Depth & Diameter** option from the **Mode** drop-down list, you need to define the depth and diameter of the countersunk in the **Depth** and **Diameter** spinners, respectively. Similarly, if you select the **Angle & Diameter** option from the **Mode** drop-down list, you need to set the value of the angle and diameter in the respective spinners. Now, set the other parameters of the hole feature in the **Extension** tab. You can also specify the thread parameters of the countersunk hole. After setting the parameters, choose the **OK** button from the **Hole Definition** dialog box.

Creating a Counterdrilled Hole

A counterdrilled hole is a combination of a counterbored and a countersunk hole. This hole type has two diameters and the transition between the bigger diameter and the smaller diameter, after applying the counterbore depth, is in the form of a tapered cone, refer to Figure 6-12. You will have to define the counterbore diameter, hole diameter, depth of counterbore, depth of the hole, and countersink angle. Figure 6-12 shows the sectional view of a counterdrilled hole. Figure 6-13 shows the spacer plate with the counterdrilled holes.

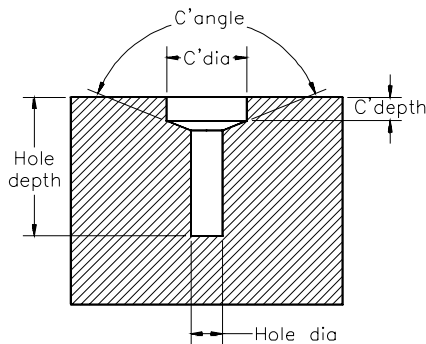


Figure 6-12 The sectional view of the counterdrilled hole

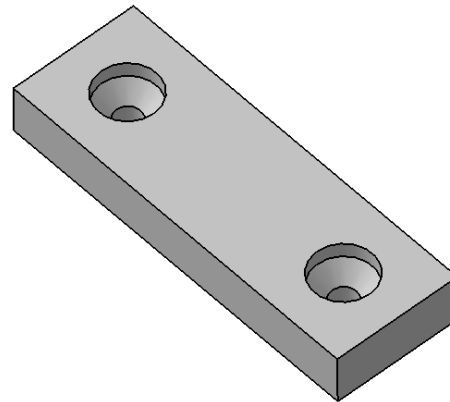


Figure 6-13 The spacer plate after adding the counterdrilled holes

To create a counterdrilled hole, select the **Counterdrilled** option from the drop-down list in the **Type** tab; its preview is displayed in the geometry area. Figure 6-14 shows the **Hole Definition** dialog box, after selecting the **Counterdrilled** option from the drop-down list. You need to set the value of the diameter of the counter using the **Diameter** spinner, and the value of its depth using the **Depth** spinner. Next, you need to set the value of the drill angle in the **Angle** spinner. You can also specify the thread parameters while creating a counterdrilled hole. After specifying all parameters, choose the **OK** button from the **Hole Definition** dialog box.

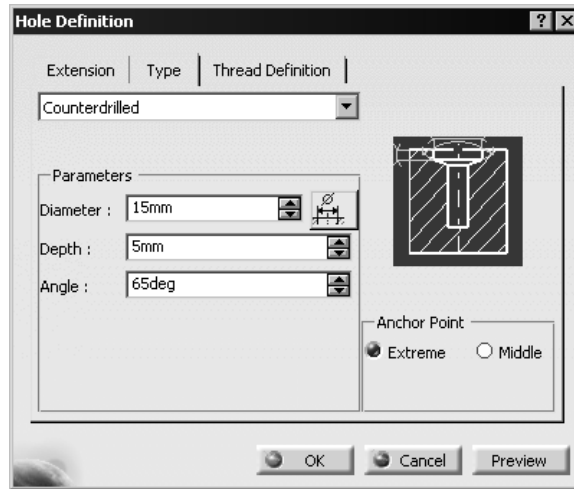


Figure 6-14 The **Hole Definition** dialog box after selecting the **Counterdrilled** option from the drop-down list in the **Type** tab



Tip. To apply cosmetic threads on holes or cylindrical shafts, choose the **Thread/Tap** button from the **Dress-Up Features** toolbar; the **Thread/Tap Definition** dialog box is displayed. You need to select the cylindrical surface on which you wish to apply the thread, then select the face from which the thread will start. Now, set the thread parameters in the **Numerical Definition** area and choose the **OK** button from the **Thread/Tap Definition** dialog box. The **Thread.1** feature is displayed in the specification tree.

You need to make sure that the **Thread/Tap** tool should not be used for applying threads to the cylindrical holes created using the **Hole** tool. If you do so, a warning message window is displayed, which prompts you to use the **Hole** command to tap a hole.

Creating Fillets

A fillet is generally provided to reduce the stress concentration in the model. The **Part** workbench of CATIA V5 provides you with the tools to fillet the sharp edges of the models. You can create simple edge fillets, variable radius fillets, face to face fillets, and tritangent fillets using the tools in the **Part** mode of CATIA V5. Choose the black arrow on the right of the **Edge Fillet** button in the **Dress-Up Features** toolbar; the **Fillets** toolbar will be invoked, as shown in Figure 6-15. The procedure of creating various types of fillets is discussed next.

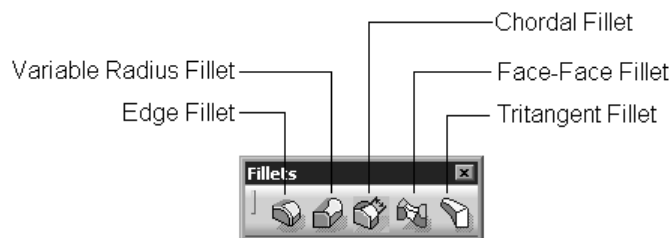


Figure 6-15 The **Fillets** toolbar

Creating an Edge Fillet

Menu: Insert > Dress-Up Features > Edge Fillet
Toolbar: Dress-Up Features > Fillets > Edge Fillet



To create an edge fillet, choose the **Edge Fillet** button from the **Fillets** toolbar; the **Edge Fillet Definition** dialog box is displayed, as shown in Figure 6-16.

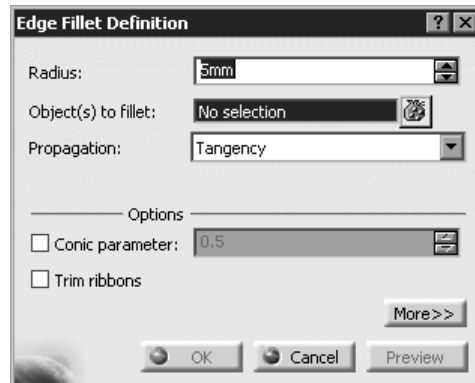


Figure 6-16 The *Edge Fillet Definition* dialog box

On invoking the **Edge Fillet Definition** dialog box, you will be prompted to select an edge or a face. Select the edge that you need to fillet; the number of the selected edges will be displayed in the **Object(s) to fillet** selection area. Note that the default radius value will be displayed only on the first selected edge. Set the value of the fillet radius using the **Radius** spinner and choose the **OK** button from the **Edge Fillet Definition** dialog box. Figure 6-17 shows the edge selected to be filleted. Figure 6-18 shows the resulting filleted edge.

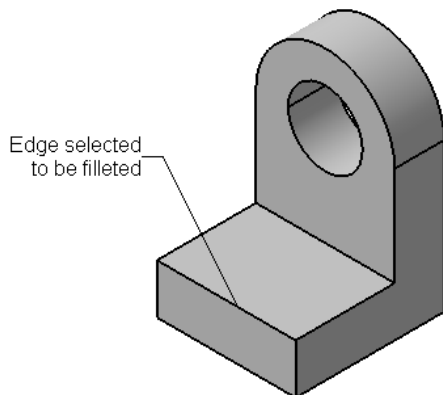


Figure 6-17 Edge selected to be filleted

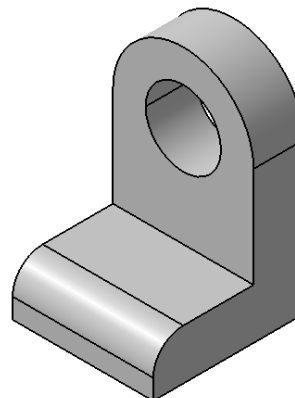


Figure 6-18 Resulting edge fillet

Figure 6-19 shows the face selected to be filleted and Figure 6-20 shows the resulting filleted face.

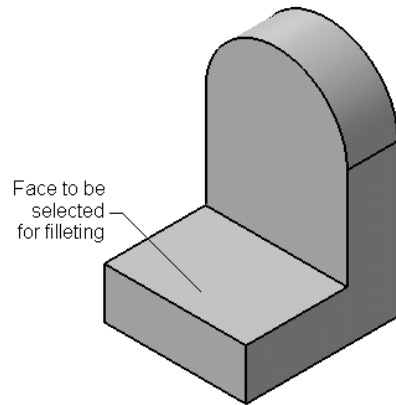


Figure 6-19 Face selected to be filleted

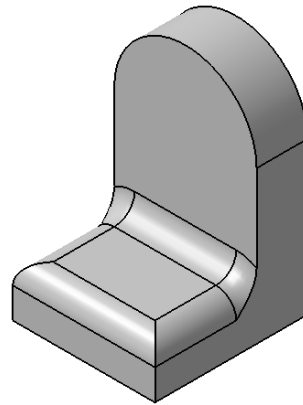



Figure 6-20 Resulting filleted face

The options in the **Edge Fillet Definition** dialog box, for creating advance edge fillets, are discussed next.

Managing Selected Entities

With the recent releases, a new option has been introduced to manage the entities in the current selection set. To do so, choose the **Selection Filter**  button on the right of the **Object(s) to fillet** selection area; the **Fillet objects** dialog box will be displayed. All the selected entities are listed in this dialog box. The **Remove** button in this dialog box is used to remove an entity from the current selection set. The **Replace** button is used to replace an entity from the current selection set with another entity from the model.

Managing the Fillet Propagation

While filleting edges, you can manage the propagation of the fillet. By default, the **Tangency** option is selected in the **Propagation** drop-down list. Therefore, the edges tangent to the selected edge will also be selected and filleted. If you select the **Minimal** option from the **Propagation** drop-down list, only the selected edge will be filleted. Figure 6-21 shows the edge to be filleted. Figure 6-22 shows the edge filleted using the **Tangency** option and Figure 6-23 shows the edge filleted using the **Minimal** option.

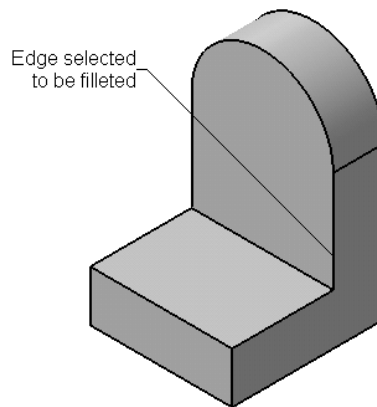


Figure 6-21 Edge selected to be filleted

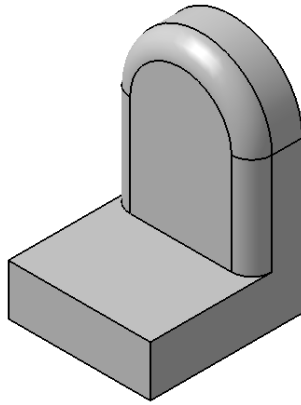


Figure 6-22 Fillet using the *Tangency* option

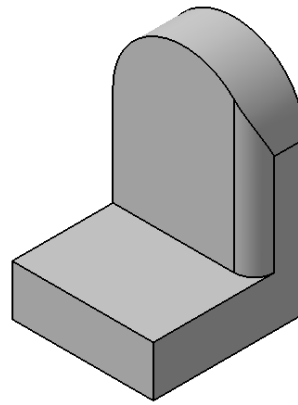


Figure 6-23 Fillet using the *Minimal* option

Creating Conical Fillets

You can create a fillet that has the geometry of a conic. Select the **Conic parameter** check box in the **Options** area; the **Conic parameter** spinner will get activated. The value in the **Conic parameter** spinner should lie between 0 and 1. A value between 0 and 0.5 will result into a fillet with elliptical geometry. A value of 0.5 will result into a fillet with parabolic geometry. Whereas, a value between 0.5 and 1 will result into a fillet with hyperbolic geometry, refer to Figure 6-24.

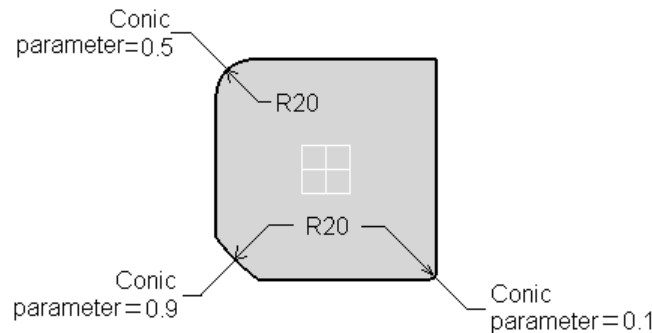


Figure 6-24 Fillets with different values of *Conic parameter*



Note

The conical fillets displayed in Figure 6-24 are created by selecting the edges of the model. You can also create conical fillets by selecting the faces of the model.

Trimming the Overlapping Fillets

You can also use the options in the **Fillet** tool to trim the intersecting surfaces. Consider the case of the model shown in Figure 6-25. Using the **Fillet** tool, three edges of the model are filleted. If you select the **Trim ribbons** check box in the **Edge Fillet Definition** dialog box, the intersecting surfaces created as a result of the fillet will be trimmed. The **Trim ribbons** check box is only enabled when the **Tangency** option is selected in the **Propagation** drop-down list. Figure 6-26 shows the resulting fillet after selecting the **Trim ribbons** check box.

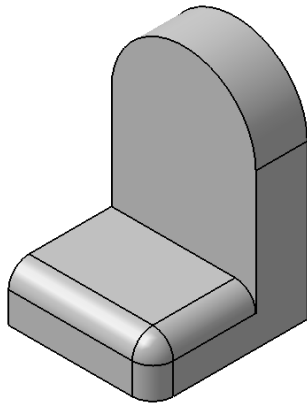


Figure 6-25 Resulting fillet with the **Trim ribbons** check box cleared

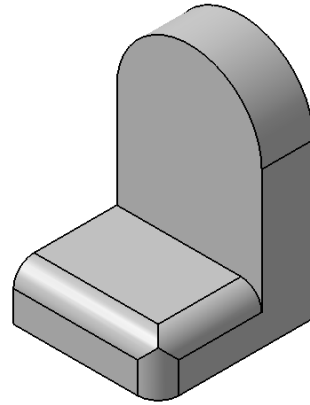


Figure 6-26 Resulting fillet with the **Trim ribbons** check box selected

Selecting the Edges to be Kept

Sometimes while filleting, some of the edges get distorted in order to accommodate the fillet radius, as shown in Figure 6-27. In this model, the bottom edge of the elliptical extruded feature is filleted. The inclined edges are distorted in order to accommodate the fillet radius. To avoid this distortion, choose the **More** button from the **Edge Fillet Definition** dialog box; the **Edge Fillet Definition** dialog box expands. Click once in the **Edge(s) to keep** selection area and select the distorted edges. Now, choose the **OK** button from the **Edge Fillet Definition** dialog box. The edges will not be distorted, as shown in Figure 6-28.

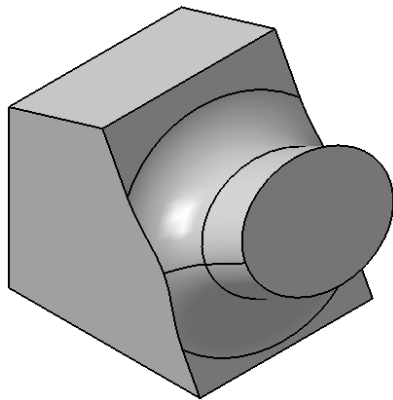


Figure 6-27 Edges distorted to accommodate the fillet radius

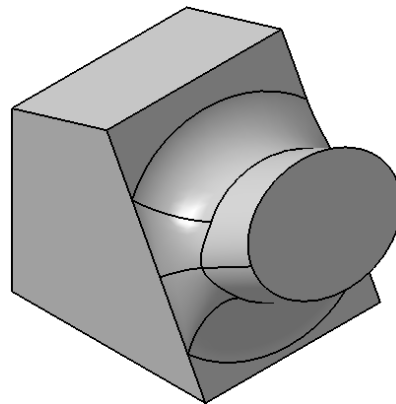


Figure 6-28 The model after selecting the edges to be kept



Note

If the fillet radius is too large to retain the edges, the **Update Diagnosis** dialog box is displayed. You need to reduce the fillet radius to create the fillet.

Setting the Limits of the Fillet

You can also set the limit of the fillet along the selected edge up to which the fillet will be created. Select the edge or edges to be filleted and set the value of the radius. Now, expand the **Edge Fillet Definition** dialog box using the **More** button. Click once in the **Limiting element(s)** selection area and select the plane up to which you need to create the fillet. An arrow that is displayed in the geometry area will define the direction of the fillet creation. You can flip the direction by clicking on the arrow in the preview of the fillet. You can also create a point or plane within the **Edge Fillet** tool to define the limit of the fillet. To create a point or plane within the **Edge Fillet** tool, right-click in the **Limiting element(s)** selection area; a contextual menu is displayed. Define the limit using the options in the contextual menu. Figure 6-29 shows the edge to be filleted and the limiting element to be selected. Figure 6-30 shows the resulting fillet.

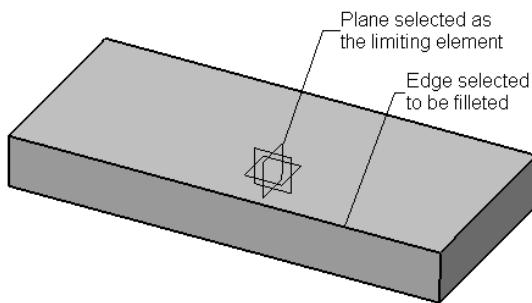


Figure 6-29 Edge to be filleted and the limiting element to be selected

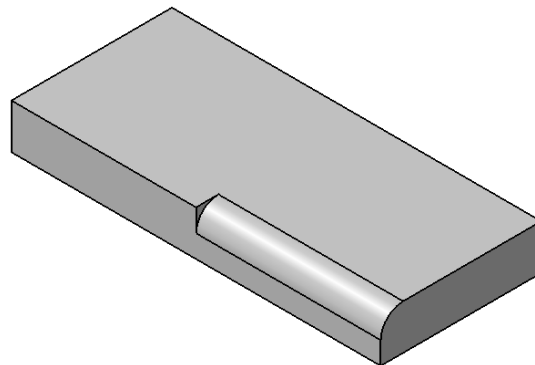


Figure 6-30 Resulting fillet



Note

Instead of selecting or creating a limiting element, you can also specify the limit of a fillet by directly selecting points on the edge to be filleted. To define the limits using this method, select the edge to fillet and define the fillet radius. Now, expand the dialog box and click once in the **Limiting element(s)** selection area. Click on the selected edge where you need to define the limit of the fillet; a blue circle is displayed on the current selection. The arrow defining the direction of the fillet creation is also displayed. If you have selected two elements to limit the fillet, you need to make sure the arrows of both the limits point in the opposite directions. You can flip the direction of arrows by clicking on them. Figure 6-31 shows the fillet after specifying two limit elements. In this figure, the arrows of both the limits point toward the midpoint of the edge.

Setback Fillet by Blending the Corners

The setback fillet is created where three or more than three edges are merged into a vertex. This fillet type is used to smoothly blend the transition surfaces generated from the edges to the fillet vertex. This smooth transition is created between all the selected edges and the selected vertex for the setback type of fillet. To create this fillet type, select the edges that you need to fillet and then set the value of the fillet radius. Now, expand the **Edge Fillet Definition** dialog box using the **More** button. Choose the **Blend corner(s)**

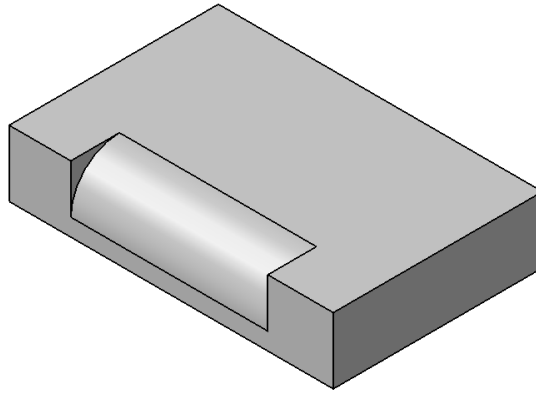


Figure 6-31 Fillet after specifying the two limit elements

button from the **Edge Fillet Definition** dialog box. The vertex formed by merging the selected edges is selected and the **Corner.1** callout is displayed attached to the vertex. You will notice that individual setback dimensions are also attached to the selected edges. Select any one of the dimensions and set its value in the **Setback distance** spinner. Similarly, set the setback distance for the other edges. Figure 6-32 shows the edges selected to be filleted. Figure 6-33 shows the preview of the setback fillet after setting the setback distance. Figure 6-34 shows the resulting setback fillet.

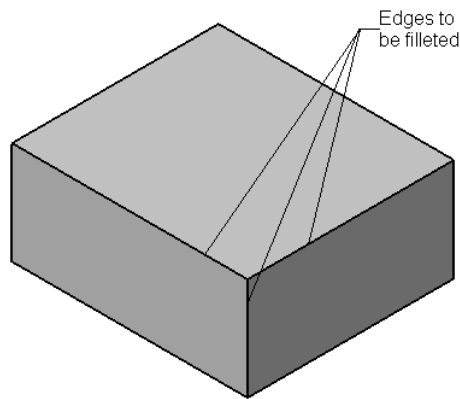


Figure 6-32 Edges selected to be filleted

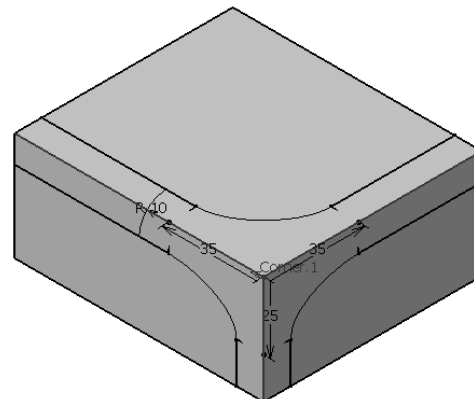


Figure 6-33 Preview of the setback fillet



Note

Make sure the setback distance is equal to or greater than the fillet radius. Else, the fillet will not be created.

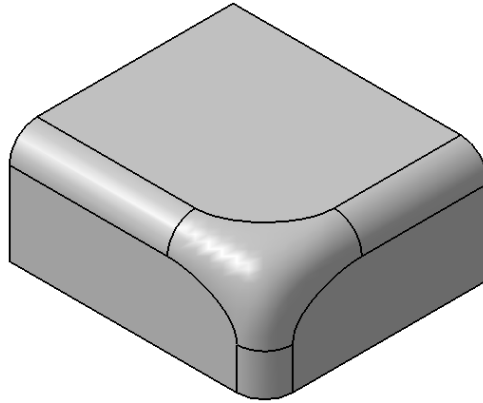


Figure 6-34 Resulting setback fillet

Creating Variable Radius Fillets

Menu: Insert > Dress-Up Features > Variable Fillet
Toolbar: Dress-Up Features > Fillets > Variable Radius Fillet



You can create a fillet by specifying different radii along the length of the selected edge using the **Variable Radius Fillet** tool. The transition of the fillet can be smooth or straight, depending upon the option you choose. To create a variable radius fillet, choose the **Variable Radius Fillet** button from the **Fillets** toolbar; the **Variable Radius Fillet Definition** dialog box will be displayed. Figure 6-35 shows the expanded form of this dialog box.

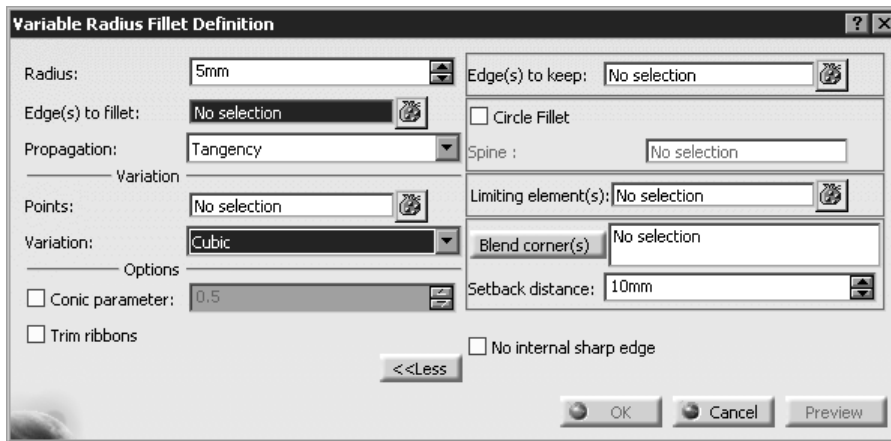


Figure 6-35 Expanded form of the **Variable Radius Fillet Definition** dialog box

Select the edge that you need to fillet; two radius callouts will be attached to the endpoints of the selected edge. You can also select multiple edges for applying the fillets and use the **Selection Filter** button on the right of the **Edge(s) to fillet** selection area to filter the selection. Next, double-click on one of the radius callouts and set the value of the radius in the **Value** spinner. Similarly, double-click on the other callout and set the value of the second radius in

the **Value** spinner. Now, choose the **OK** button from the **Variable Radius Fillet Definition** dialog box. The model, after creating the variable radius fillet, is shown in Figure 6-36.

You can also define additional control points on the selected edge. To do so, click once in the **Points** selection area of the **Variation** area of the **Variable Radius Fillet Definition** dialog box and then click anywhere on the edge; a callout will be attached to the specified point. You can double-click on the callout value to modify the fillet radius. You can also create as many control points as you need by repeating this procedure. Figure 6-37 shows a variable radius fillet after specifying the radii at additional control points.

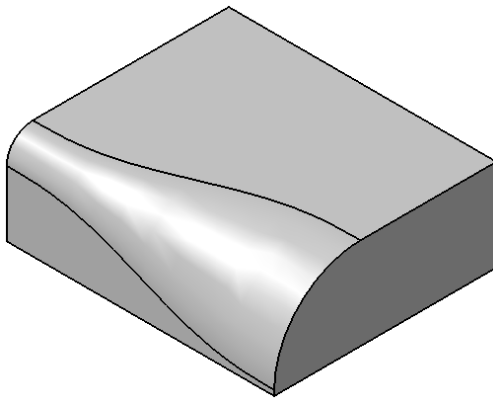


Figure 6-36 Variable radius fillet created by specifying radii at the two endpoints of the edge

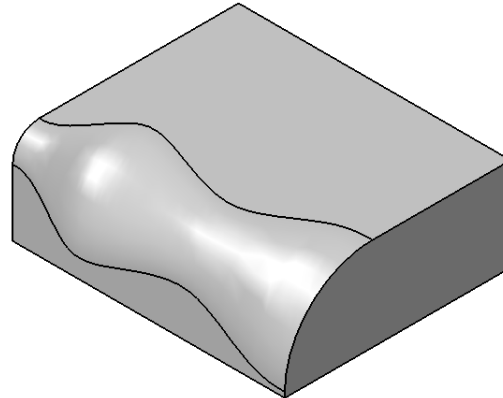


Figure 6-37 Variable radius fillet after defining additional control points

You can also manage the transition of the variable radius fillet. By default, the **Cubic** option is selected in the **Variation** drop-down list of the **Variation** area of the **Variable Radius Fillet Definition** dialog box. This option will result in smooth transition of the fillet surface. If you select the **Linear** option from the **Variation** drop-down list, it will result in straight transition of the fillet surface. Figure 6-38 shows the variable radius fillet with the **Cubic** option selected and Figure 6-39 shows the variable radius fillet with the **Linear** option selected.

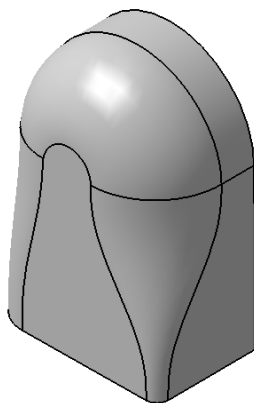


Figure 6-38 Variable radius fillet with the **Cubic** option selected

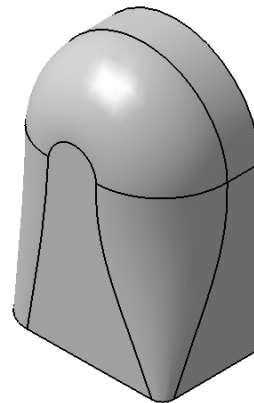


Figure 6-39 Variable radius fillet with the **Linear** option selected

**Note**

The other options in the **Variable Radius Fillet Definition** dialog box are similar to those discussed for the **Edge Fillet Definition** dialog box.

Creating Chordal Fillets

Menu: Insert > Dress-Up Features > Chordal Fillet
Toolbar: Dress-Up Features > Fillets > Chordal Fillet



The **Chordal Fillet** tool is used to fillet the selected edge by specifying the chord length between the two side edges of the fillet, refer to Figure 6-40. Note that the radius value is not required to create the chordal fillet. You can also create variable chord length fillet along the length of the edge. To create a chordal fillet, choose the **Chordal Fillet** button from the **Fillets** toolbar; the **Chordal Fillet Definition** dialog box will be displayed. Figure 6-41 shows the expanded form of this dialog box.

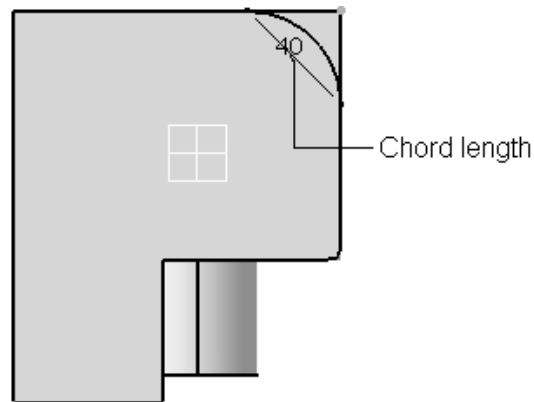


Figure 6-40 Model specifying the chord length

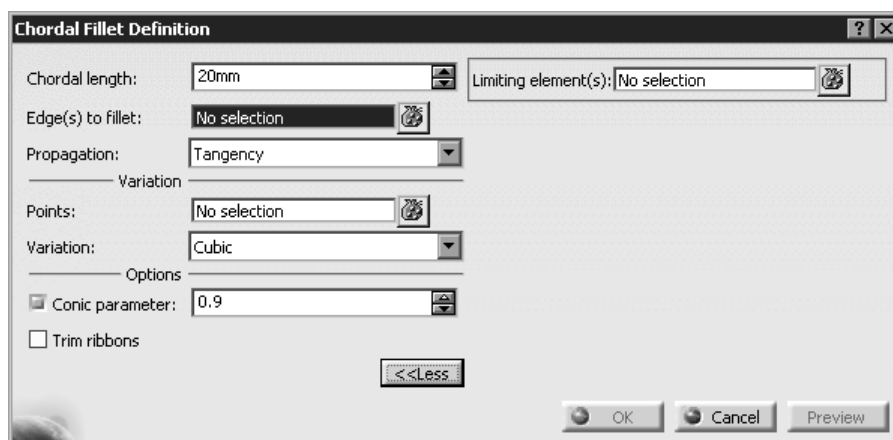


Figure 6-41 Expanded form of the **Chordal Fillet Definition** dialog box

Select the edge that you need to fillet; two chord length callouts will be attached to the endpoints of the selected edge. You can also select multiple edges for applying the fillet and use the **Selection Filter** button on the right of the **Edge(s) to fillet** selection area to filter the selection. Next, double-click on one of the chord length callouts and set the value of the chord length in the **Value** spinner. Similarly, double-click on the other callout and set the value of the second chord length in the **Value** spinner. Now, choose the **OK** button from the **Chordal Fillet Definition** dialog box.

Creating Face-Face Fillets

Menu: Insert > Dress-Up Features > Face-Face Fillet
Toolbar: Dress-Up Features > Fillets > Face-Face Fillet



The **Face-Face Fillet** tool is used to fillet the selected faces of the model. To create a face fillet, choose the **Face-Face Fillet** button from the **Fillets** toolbar; the **Face-Face Fillet Definition** dialog box will be displayed, as shown in Figure 6-42.

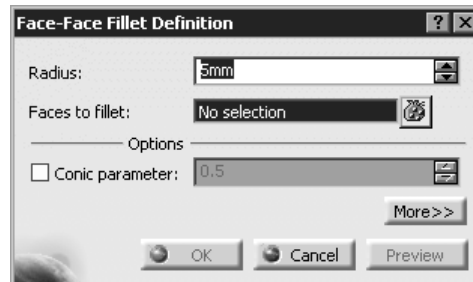


Figure 6-42 The **Face-Face Fillet Definition** dialog box

Select the first and second faces from the geometry area and then set the value of the radius of the fillet using the **Radius** spinner. Choose the **Preview** button from the **Face-Face Fillet Definition** dialog box. If the **Feature Definition Error** window is displayed, you need to modify the value of the fillet radius after exiting this window. Figure 6-43 shows the faces to be selected to create the face-face fillet and Figure 6-44 shows the resulting face-face fillet.

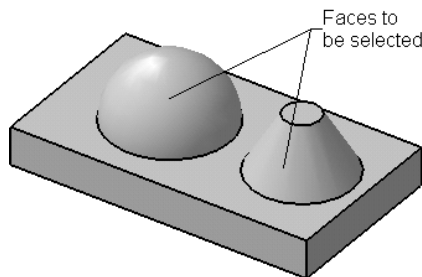


Figure 6-43 Faces to be selected

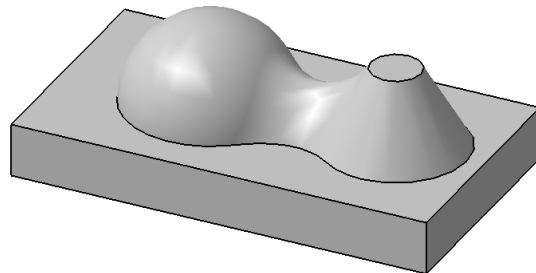


Figure 6-44 Resulting face-face fillet

Creating Tritangent Fillets

Menu: Insert > Dress-Up Features > Tritangent Fillet
Toolbar: Dress-Up Features > Fillets > Tritangent Fillet



The **Tritangent Fillet** tool is used to create the fillet feature that is tangent to three selected faces. To create a tritangent fillet, choose the **Tritangent Fillet** button from the **Fillets** toolbar; the **Tritangent Fillet Definition** dialog box will be displayed, as shown in Figure 6-45.

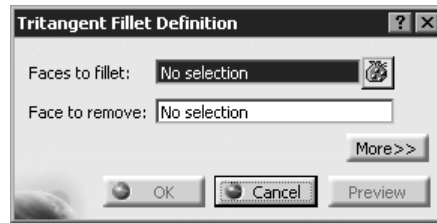


Figure 6-45 The **Tritangent Fillet Definition** dialog box

You are prompted to select the first face. On doing so, you will be prompted to select the second face. Next, you are prompted to select the face to be removed. Select the face from the geometry area, refer to Figure 6-46. Choose the **Preview** button from the **Tritangent Fillet Definition** dialog box to preview the tritangent fillet. Figure 6-46 shows the faces to be selected and Figure 6-47 shows the resulting tritangent fillet.

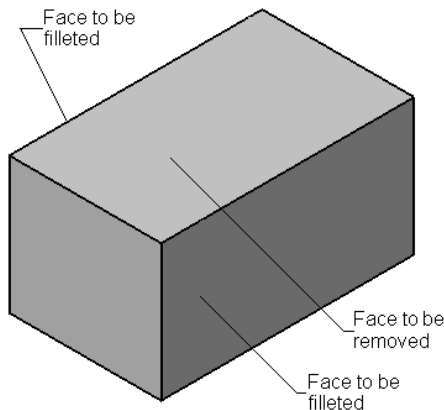


Figure 6-46 Faces to be selected

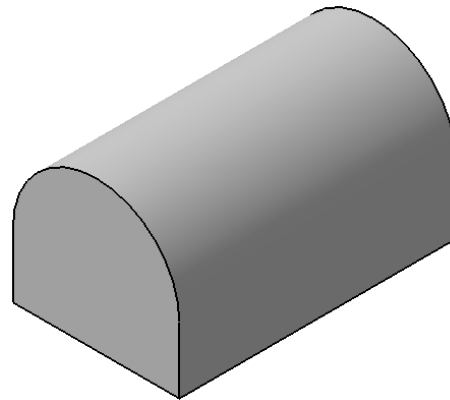


Figure 6-47 Resulting tritangent fillet

Creating Chamfers

Menu: Insert > Dress-Up Features > Chamfer
Toolbar: Dress-Up Features > Chamfer



Chamfering is defined as a process by which the sharp edges are beveled in order to reduce the stress concentration in the model. This process also eliminates the sharp edges that are not desirable. To chamfer the edges of a model, choose the **Chamfer** button from the **Dress-Up Features** toolbar; the **Chamfer Definition** dialog box will be displayed, as shown in Figure 6-48.

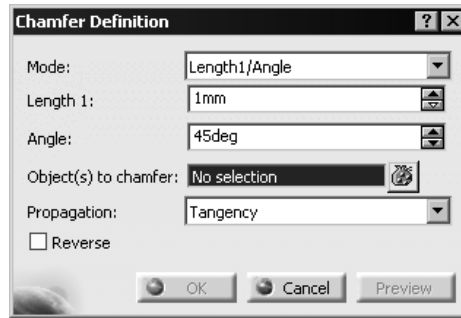


Figure 6-48 The Chamfer Definition dialog box

You are prompted to specify the required data to define the chamfer. First, you need to select the edges or faces that are to be chamfered. If you select a face to chamfer, all edges of that face are chamfered. The numbers of the selected elements are displayed in the **Object(s) to chamfer** selection area. You can also use the **Selection Filter** button on the right of the **Object(s) to chamfer** selection area to filter the selection.

You will notice that the **Length1/Angle** option is selected by default in the **Mode** drop-down list. Therefore, you need to define the values of the length of the chamfer and its angle in the **Length 1** and **Angle** spinners, respectively. On selecting the **Length1/Length2** option from the **Mode** drop-down list, you will be prompted to define the value of the first and second lengths of the chamfer in the **Length 1** and **Length 2** spinners, respectively.

To chamfer all edges tangent to the selected edges, select the **Tangency** option from the **Propagation** drop-down list. To chamfer only the selected edge, select the **Minimal** option from the **Propagation** drop-down list.

The **Reverse** check box is selected to flip the direction of the first length. Figure 6-49 shows the edge selected to be chamfered and Figure 6-50 shows the resulting chamfer.

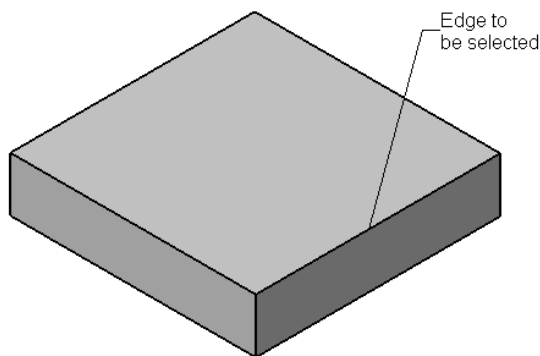


Figure 6-49 Edge selected to chamfer

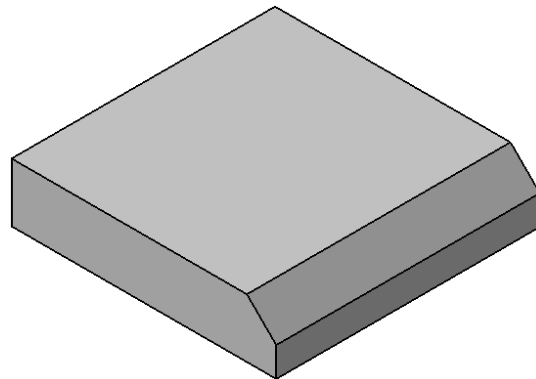


Figure 6-50 Resulting chamfer

Adding a Draft to the Faces of the Model

A draft is defined as the process of adding a taper angle to the faces of the model. Adding a draft to the faces of the model is one of the most important operations, especially while creating the components that need to be cast, molded, or formed. Draft angles enable components to be easily ejected from the die. The **Part** workbench of CATIA V5 provides you with various tools to draft faces of the model. These tools are discussed next.

Adding a Simple Draft

Menu: Insert > Dress-Up Features > Draft
Toolbar: Dress-Up Features > Drafts > Draft Angle



The **Draft Angle** tool is the most widely used tool to add a draft to the faces of the model. To add a draft, choose the **Draft Angle** button from the **Drafts** toolbar; the **Draft Definition** dialog box will be displayed, as shown in Figure 6-51. Also, an arrow will be displayed at the origin, pointing in the default pull direction.



Figure 6-51 The *Draft Definition* dialog box

Select the faces from the geometry area on which you need to add the draft angle; the selected faces will be displayed in brown. The faces tangent to the selected face are automatically selected. You can also filter the selection using the **Selection Filter** button on the right of the **Face(s) to draft** selection area. Next, you need to define a neutral plane. Click once in the **Selection** selection area in the **Neutral Element** area and then select a face or plane that will be defined as the neutral plane. By default, the **None** option is selected in the **Propagation** drop-down list. If you select the **Smooth** option, the faces tangent to the selected face are also selected automatically as the neutral element. Now, set the value of the draft angle in the **Angle** spinner and choose the **OK** button. Figure 6-52 shows the faces to be drafted and the face to be selected as the neutral plane. Figure 6-53 shows the resulting drafted faces.

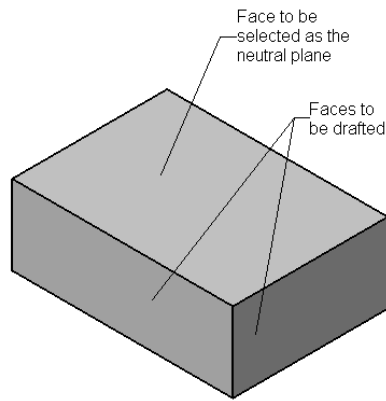


Figure 6-52 Faces and planes to be selected

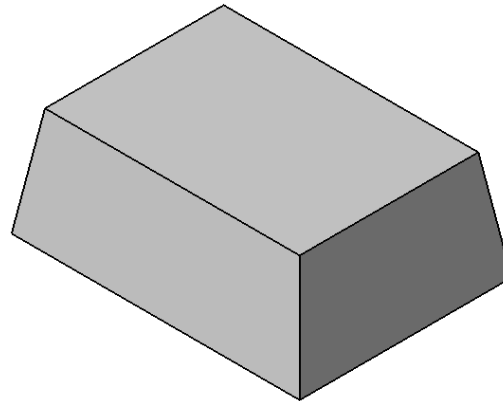


Figure 6-53 Resulting drafted faces

Figure 6-54 shows the xy plane to be selected as the neutral plane and Figure 6-55 shows the resulting drafted faces.

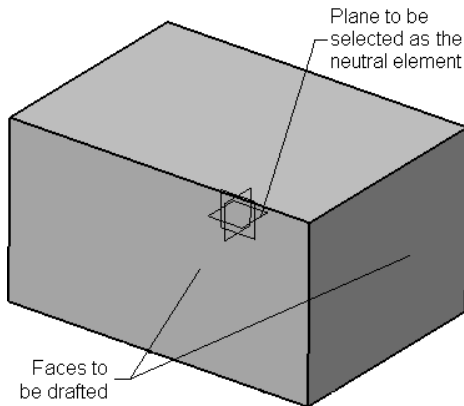


Figure 6-54 Faces and planes to be selected

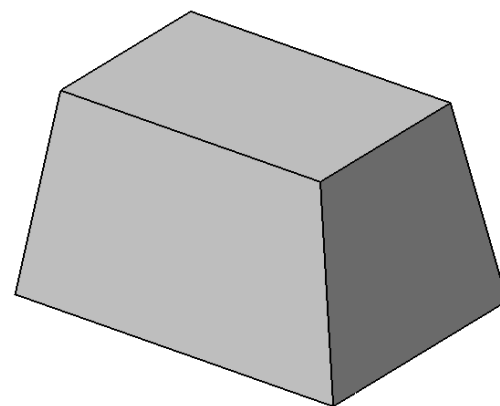


Figure 6-55 Resulting drafted faces



Tip. To add a draft to all faces that are in contact with the neutral face, instead of selecting all the faces one by one, select the **Selection by neutral face** check box and select the neutral face.

Defining the Parting Element while Adding Drafts to the Faces

You can also define the parting elements while drafting the faces of the model. To define it, choose the **More** button from the **Draft Definition** dialog box to expand the dialog box. If you choose the **Parting = Neutral** check box from the **Parting Element** area, the neutral element is selected as the parting element. Consider the case shown in Figure 6-54, in which a plane passing through the center of the model is selected as the neutral plane. Figure 6-56 shows the faces drafted with the **Parting = Neutral** check box selected. When you select the **Parting = Neutral** check box, the **Draft both sides** check box is enabled. If you select this check box, the draft is added to both sides of the parting element, refer to Figure 6-57.

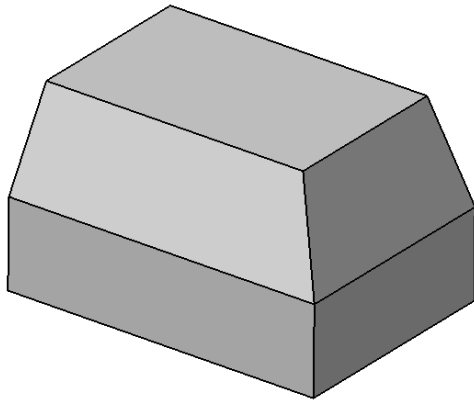


Figure 6-56 Faces drafted with the **Parting = Neutral** check box selected

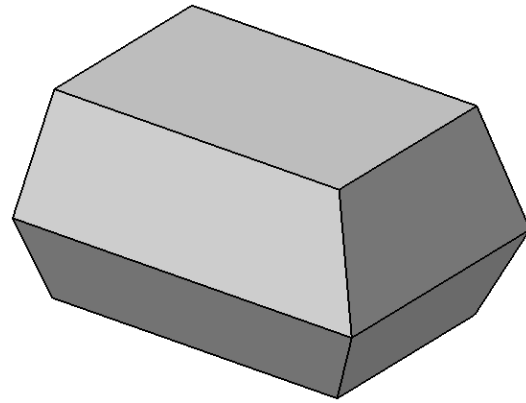


Figure 6-57 Faces drafted with the **Draft both sides** check box selected

You can also select a user-defined parting element other than the neutral plane. To select a user-defined parting element, select the **Define parting element** check box from the **Parting Element** area and select the parting element from the geometry area. Now, set the other parameters of the draft and choose the **OK** button from the **Draft Definition** dialog box. Figure 6-58 shows the faces to be drafted, neutral plane, and the parting plane and Figure 6-59 shows the resulting drafted faces.

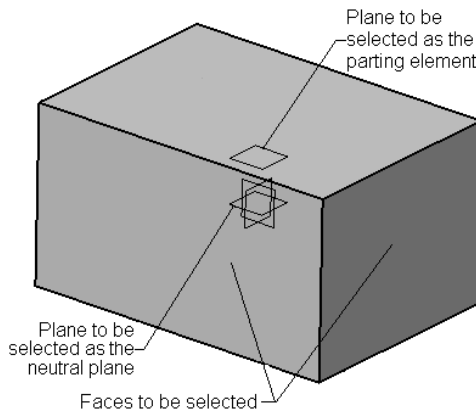


Figure 6-58 References to be selected

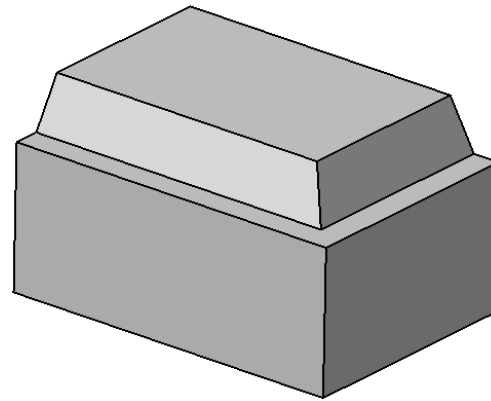


Figure 6-59 Resulting drafted faces

You can also define the limit elements while adding a draft to the faces of the model. To do so, click once in the **Limiting Element(s)** selection area and select the limiting elements from the geometry area. You need to make sure that if you specify two limiting elements, the feature is created in the opposite directions. Figure 6-60 shows the limiting elements to be selected and Figure 6-61 shows the resulting draft feature.



Tip. By default, the pulling direction is selected along the Z axis direction. You can also specify a user-defined pulling direction by clicking once in the **Pulling Direction** selection area and then selecting the pulling direction from the geometry area.

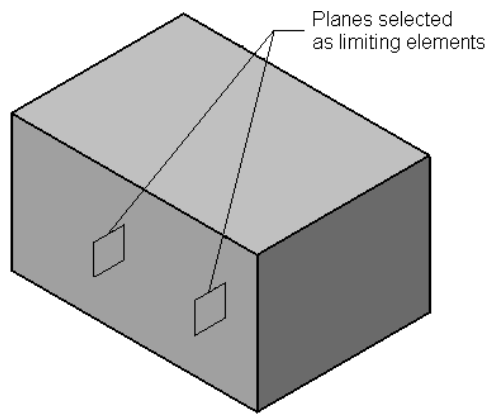


Figure 6-60 Limiting elements to be selected

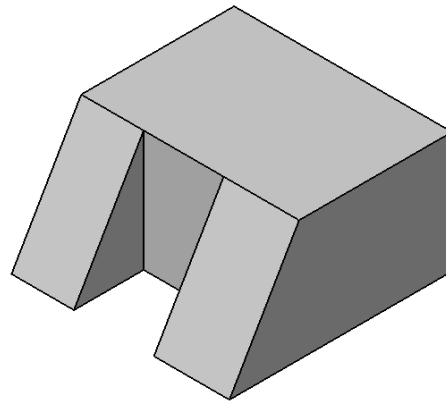


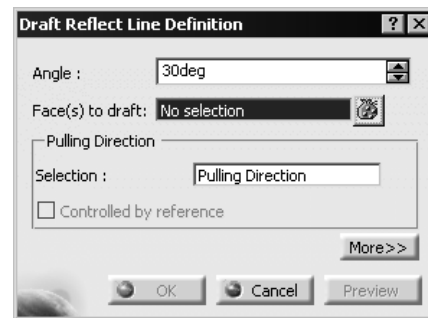
Figure 6-61 Resulting drafted feature

Adding Drafts Using the Reflect Line

Menu: Insert > Dress-Up Features > Reflect Line
Toolbar: Dress-Up Features > Drafts > Reflect Line



The **Draft Reflect Line** tool is used to create the draft feature using the silhouette lines of the selected curved face as the neutral element. To create this type of draft feature, choose the **Draft Reflect Line** button from the **Drafts** toolbar; the **Draft Reflect Line Definition** dialog box is displayed, as shown in Figure 6-62. Select a curved face from the geometry area. You can also filter the selection using the **Filter Selection** button. The faces tangent to the selected face are also selected automatically. You will notice that a pink color sketch will be created along the silhouette of the selected face. Now, expand the dialog box using the **More** button and select the **Define parting element** check box. You are prompted to select the parting element. Select the plane or planar face that will be used as the parting element. Set the value of the draft angle and choose the **OK** button from the **Draft Reflect Line Definition** dialog box. Figure 6-63 shows the face to be drafted and the plane to be selected as the parting element. Figure 6-64 shows the resulting drafted feature.

Figure 6-62 The **Draft Reflect Line Definition** dialog box

Adding a Variable Angle Draft

Menu: Insert > Dress-Up Features > Variable Angle Draft
Toolbar: Dress-Up Features > Drafts > Variable Angle Draft



To create a variable angle draft, choose the **Variable Angle Draft** button from the **Drafts** toolbar; the **Draft Definition** dialog box will be displayed, as shown in Figure 6-65.

Select the face to add the draft. You can select only one face while adding a draft using this

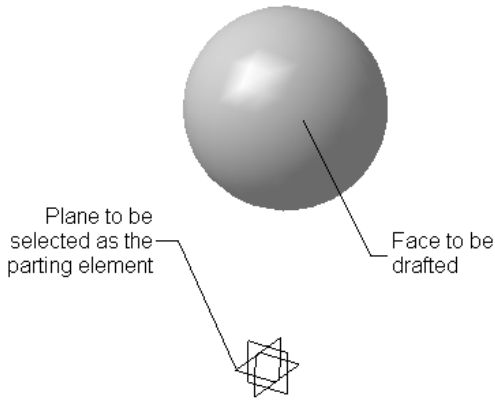


Figure 6-63 Face and plane to be selected

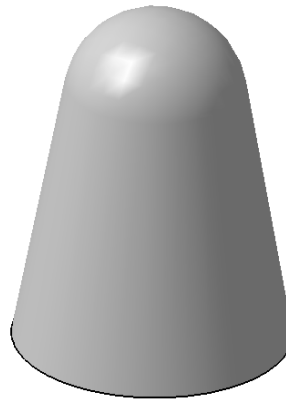


Figure 6-64 Resulting drafted feature

tool. Define the neutral element by selecting a plane or face. You will notice that two angular dimensions are displayed attached to the end points of the selected face. One by one, select both the angles and set their values using the **Angle** spinner. You can also filter the selections using the **Selection Filter** button. Figure 6-66 shows the references to be selected and Figure 6-67 shows the resulting face after adding the draft.

You can also define additional points to specify other variable angles. Note that the point can only be selected on the edge from which the angle is measured. To define an additional point, click anywhere on the edge from where the angle is measured. If you want to define points whose distances need to be controlled,



Figure 6-65 The *Draft Definition* dialog box

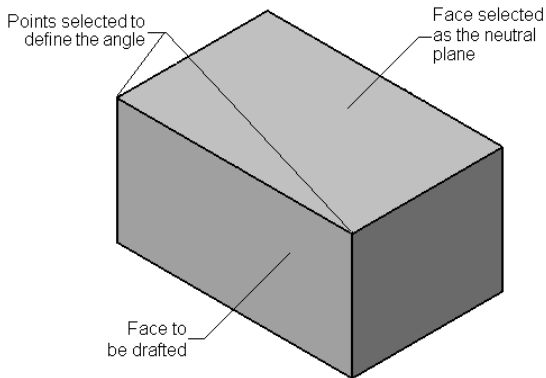


Figure 6-66 References to be selected

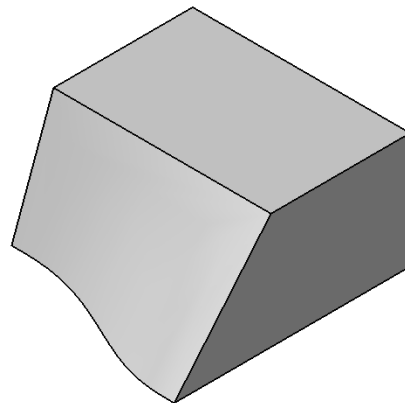


Figure 6-67 Resulting face after adding the draft

right-click in the **Points** selection area and invoke the contextual menu. Create additional points and then set the draft angle by using the options in the contextual menu.

Creating a Shell Feature

Menu: Insert > Dress-Up Features > Shell
Toolbar: Dress-Up Features > Shell



The **Shell** tool is used to scoop out the material from the model and remove the selected faces, thereby resulting in a thin walled structure. To create a shell feature, choose the **Shell** button from the **Dress-Up Features** toolbar; the **Shell Definition** dialog box will be displayed, as shown in Figure 6-68.

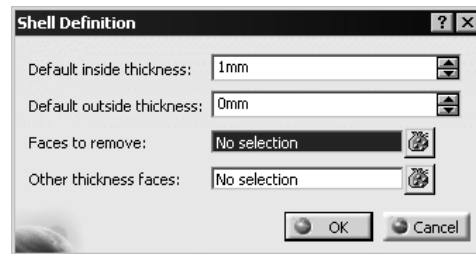


Figure 6-68 The **Shell Definition** dialog box

Next, you need to select the face or faces to be removed. Select them from the geometry area. The faces tangent to the selected face are selected automatically. You can filter the selection using the **Selection Filter** button. Now, set the value of the wall thickness in the **Default inside thickness** spinner in the **Shell Definition** dialog box. You can also define the outside thickness of the shell using the **Default outside thickness** spinner. Now, choose the **OK** button from the **Shell Definition** dialog box. Figure 6-69 shows the faces to be removed and Figure 6-70 shows the resulting shelled model. If you do not select any faces to be removed, the resulting shelled model will be a hollow model with a specified wall thickness.

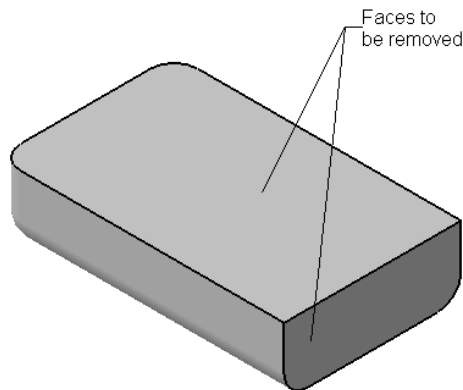


Figure 6-69 Faces to be selected for removal

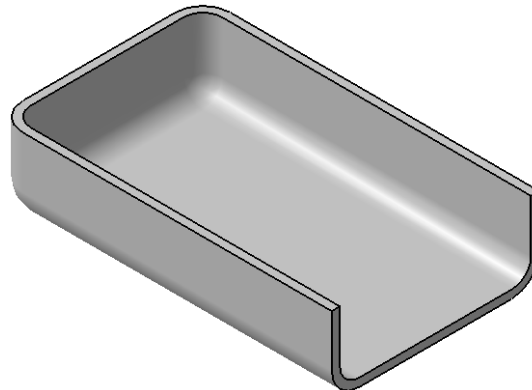


Figure 6-70 Resulting shelled model

Creating a Multithickness Shell

You can also define different shell thickness values for the faces of the shell feature. To create a multithickness shell feature, first select the faces to be removed and then specify the default inside or outside thickness of the shell. Now, click once in the **Other thickness faces** selection area and then select the faces on which you need to define the different shell thickness value. The faces tangent to the selected face are selected automatically. The

selected faces will be highlighted in brown and the shell thickness dimensions will be attached to them. Select the thickness value of one of the highlighted faces from the geometry area; the selected value is displayed in the **Default inside thickness** spinner in the **Shell Definition** dialog box. Modify the thickness value and repeat the process for the remaining highlighted faces. After setting all shell thickness values, choose the **OK** button from the **Shell Definition** dialog box. Figure 6-71 shows the face to be removed and the faces to define different shell thicknesses and Figure 6-72 shows the resulting shelled model.

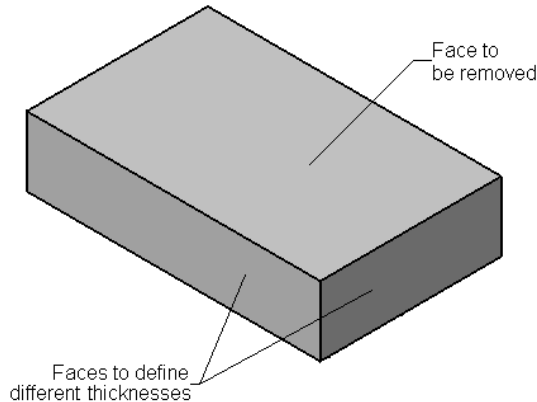


Figure 6-71 Faces to be selected

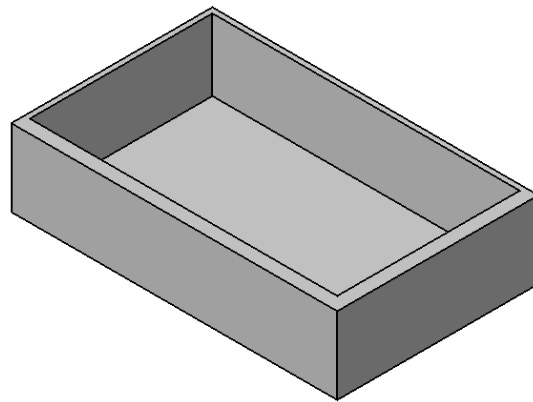


Figure 6-72 Resulting shelled model

TUTORIALS

Tutorial 1

In this tutorial, you will create the model of the nozzle of a vacuum cleaner shown in Figure 6-73. Its views and dimensions are shown in Figure 6-74. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

- Start a new file in the **Part** workbench and create the base feature of the model by extruding the sketch along the selected direction, refer to Figures 6-75 through 6-79.
- Create the second feature of the model by extruding a sketch using the **Drafted Fillet Pad** tool, refer to Figures 6-80 and 6-81.
- Create the third feature of the model, which is a cut feature. It will be used to remove the unwanted portion of the second feature, refer to Figures 6-82 and 6-83.
- Apply fillets to all edges of the model, refer to Figures 6-84 through 6-87.
- Shell the model using the **Shell** tool, refer to Figures 6-88 and 6-89.

Creating the Base Feature of the Model

The base feature of this model is created by first creating a plane at an angle of 26-degree and then extruding a sketch drawn on that plane. The sketch will be extruded along a selected direction. In this model, you will learn a technique to create the reference sketch first and then follow it to create the model. Therefore, you will first draw the reference sketch.

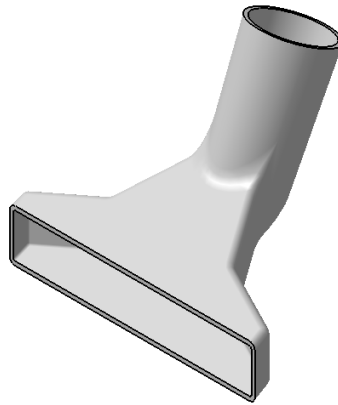


Figure 6-73 Model for Tutorial 1

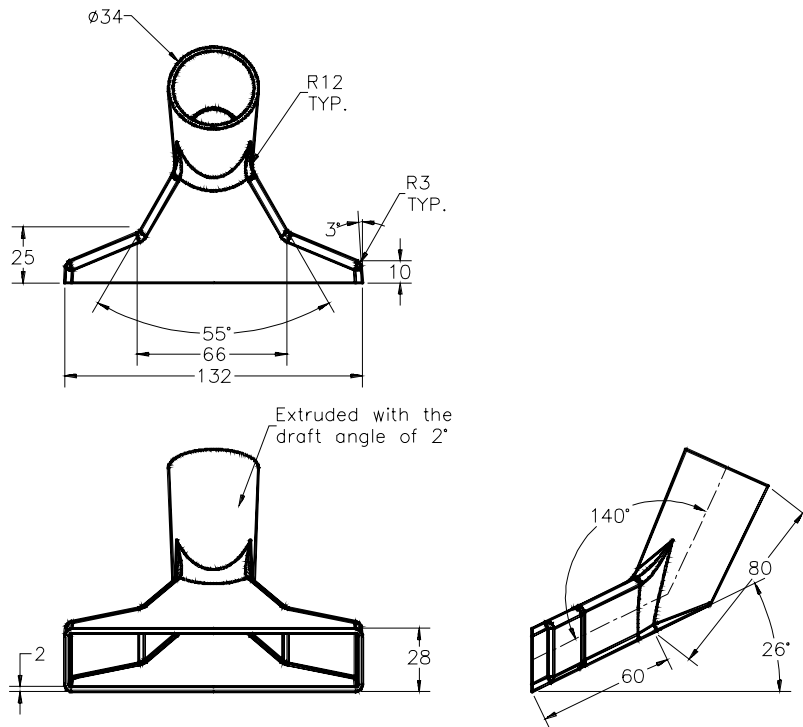


Figure 6-74 Views and dimensions for Tutorial 1

1. Start a new file in the **Part** workbench. Select the zx plane and invoke the **Sketcher** workbench.
2. Draw the sketch, as shown in Figure 6-75, and then exit the **Sketcher** workbench.
3. Select the yz plane and invoke the **Sketcher** workbench. Place a point collinear to the H axis at any distance, as shown in Figure 6-76. Exit the **Sketcher** workbench.

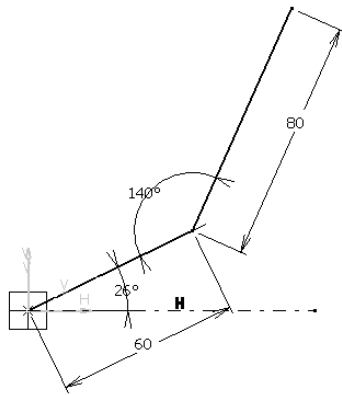


Figure 6-75 Reference sketch

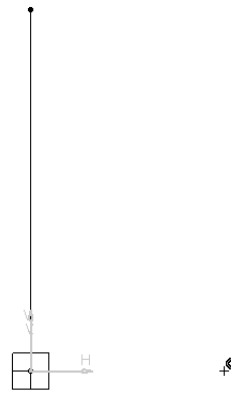


Figure 6-76 Point to be placed

After drawing the reference sketch and placing the point, you need to create a reference plane to create the base feature.

4. Create a plane by selecting three points, as shown in Figure 6-77.
5. Invoke the **Sketcher** workbench after selecting the newly created plane as the sketching plane and draw the sketch of the base feature, as shown in Figure 6-78.

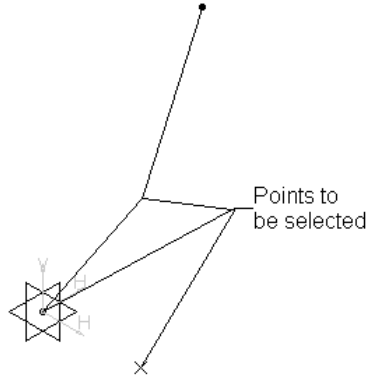


Figure 6-77 Points to be selected to create a plane

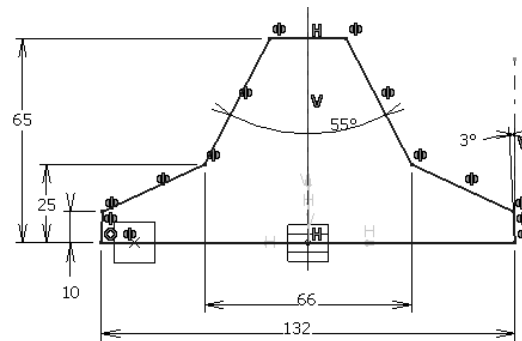


Figure 6-78 Sketch of the base feature

6. Exit the **Sketcher** workbench. Choose the **Pad** button from the **Sketch-Based Features** toolbar; the **Pad Definition** dialog box is displayed.
7. Set the value of the **Length** spinner to **28**. The preview of the extruded feature is displayed in the geometry area. If the sketch is extruded in the downward direction, then choose the **Reverse Direction** button to flip the direction of the feature creation.
8. Now, choose the **More** button to expand the **Pad Definition** dialog box.

9. Clear the **Normal to profile** check box provided in the **Direction** area and select the xy plane as the direction of extrusion.
10. Choose the **OK** button from the **Pad Definition** dialog box to complete the feature creation. The model, after creating the base feature, is shown in Figure 6-79.

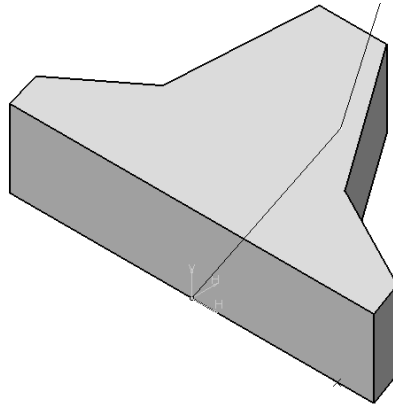


Figure 6-79 Model after creating the base feature

Creating the Second Feature

The second feature of this model is a drafted extrude feature created using the **Drafted Filleted Pad** tool. In this feature, you will extrude the sketch drawn on a plane created normal to the right line of the reference sketch.

1. Invoke the **Plane** tool and select the **Normal to curve** option from the **Plane type** drop-down list.
2. Now, select the right line of the reference sketch as the curve and then select the upper endpoint of the same line as the point on which the plane will be created. The preview of the plane is displayed in the geometry area.
3. Choose the **OK** button from the **Plane Definition** dialog box.
4. Use the newly created plane to invoke the **Sketcher** workbench and draw the sketch of the second feature, as shown in Figure 6-80.
5. Exit the **Sketcher** workbench and invoke the **Drafted Filleted Pad** tool from the **Pads** toolbar.
6. Set the value of the **Length** spinner to **85** and select the newly created plane from the geometry area as the second limit.
7. Set the value of the draft angle in the **Angle** spinner to **2deg**. Choose the **Reverse Direction** button to flip the direction of the feature creation.
8. Clear all the radio buttons in the **Fillets** area and choose the **OK** button from the **Drafted**

Filletted Pad Definition dialog box. Make sure that the second limit is the plane defined while creating the base feature. The model, after creating the second feature, is shown in Figure 6-81.

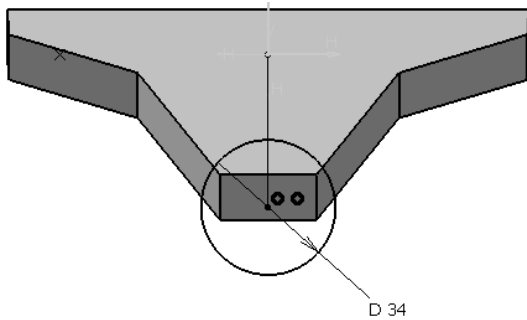


Figure 6-80 Sketch for the second feature

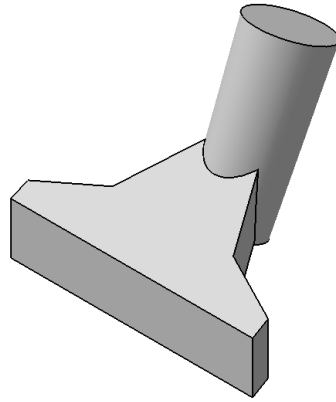


Figure 6-81 Resulting second feature

Creating the Third Feature

Next, you need to create the third feature of the model to remove the unwanted portion of the second feature.

1. Select the zx plane and invoke the **Sketcher** workbench. Draw the open sketch at an angle of 26-degrees, as shown in Figure 6-82, and exit the **Sketcher** workbench.
2. Extrude the sketch using the **Pocket** tool up to the last on both sides of the sketch. You may have to reverse the side of material removal.
3. Using the **Hide/Show** tool, hide Sketch1, Sketch2, Plane1, and Plane2. The model, after creating the third feature, is shown in Figure 6-83.

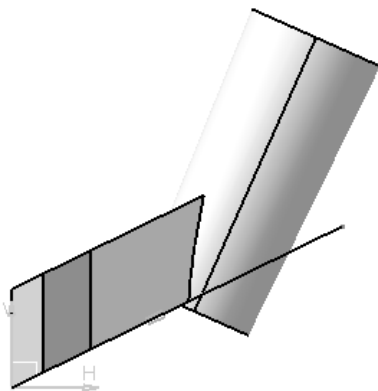


Figure 6-82 Sketch for the pocket feature

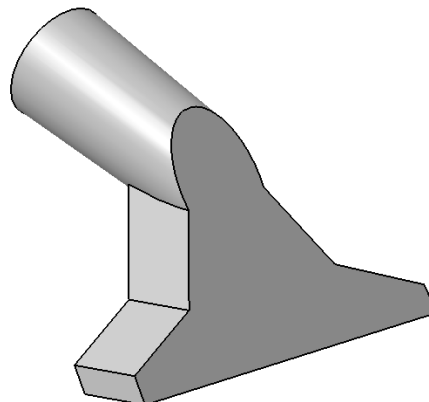



Figure 6-83 Model after creating the pocket feature

Filleting the Edges of the Model

Next, you need to fillet two sets of edges of the model. You need to apply the fillet feature twice because the two sets of edges need different fillet radii. First, you will fillet the set of edges that needs the fillet radius of 12.

1. Double-click on the **Edge Fillet** button in the **Dress-Up Features** toolbar; the **Edge Fillet Definition** dialog box is displayed. 
2. Select the edges, as shown Figure 6-84, and set the value of the **Radius** spinner to **12**.
3. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The model, after creating the first set of fillet, is shown in Figure 6-85.

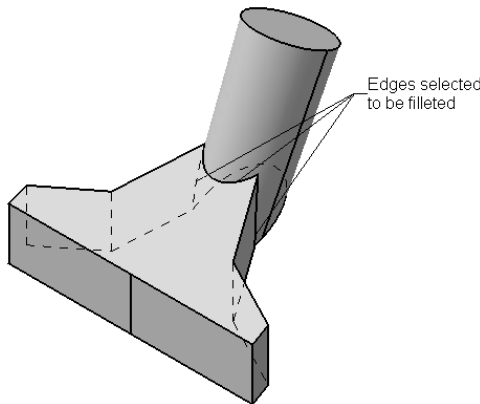


Figure 6-84 Edges selected to be filleted

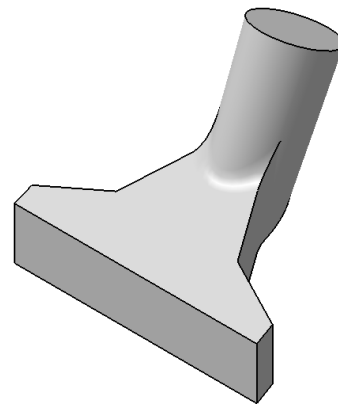



Figure 6-85 The model after creating the fillet

Next, you need to apply the fillet to the second set of edges. As you double clicked on the **Edge Fillet** button, the **Edge Fillet Definition** dialog box is displayed again.

4. Select all edges of the model, except the edges that are shown in Figure 6-86.
5. Set the value of the **Radius** spinner to **3** and choose the **OK** button from the **Edge Fillet Definition** dialog box. Cancel this dialog box when it is displayed again. The model, after applying the fillet to the second set of edges, is shown in Figure 6-87.

Creating the Shell Feature

Lastly, you need to create the shell feature. The shell feature will also be used to remove the end faces of the model, leaving behind a thin walled structure.

1. Choose the **Shell** button from the **Dress-Up Features** toolbar; the **Shell Definition** dialog box is displayed. 
2. Select the faces to be removed, as shown in Figure 6-88.
3. Set the value of the **Default inside thickness** spinner to **2** and choose the **OK** button

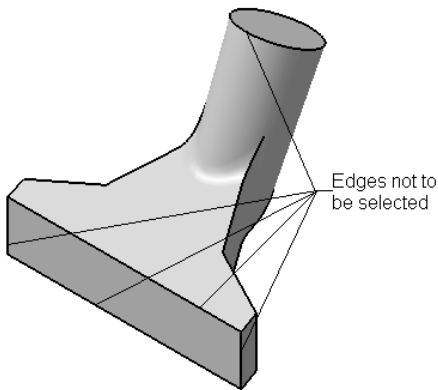


Figure 6-86 Edges not to be selected

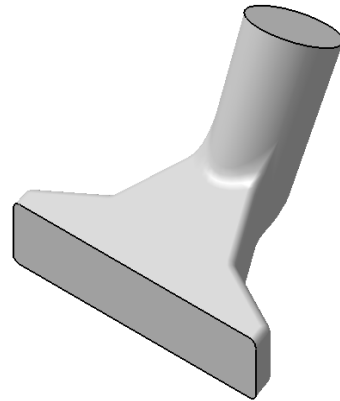


Figure 6-87 Model after creating the second fillet

from the **Shell Definition** dialog box. The final model, after creating the shell feature, is shown in Figure 6-89.

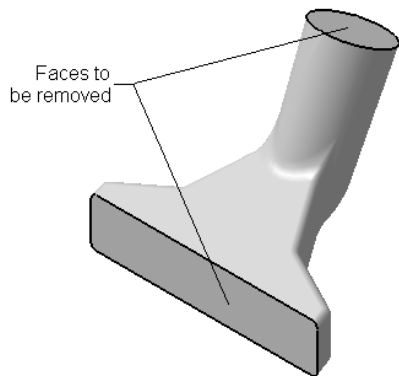


Figure 6-88 Faces to be removed

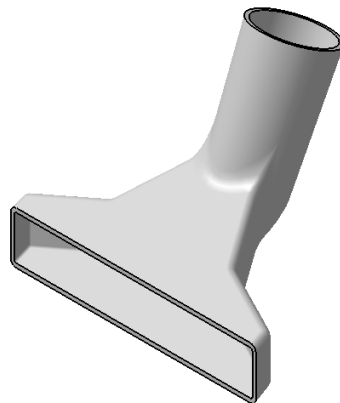


Figure 6-89 Final model after shelling

Saving and Closing the File

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create the *c06* folder inside the *CATIA* folder.
2. Enter the name of the file as *c06tut1* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c06* folder.
3. Close the part file by choosing **File > Close** from the menu bar.

Tutorial 2

In this tutorial, you will create the model of the plastic cover shown in Figure 6-90. Its views and dimensions are shown in Figure 6-91. **(Expected time: 30 min)**

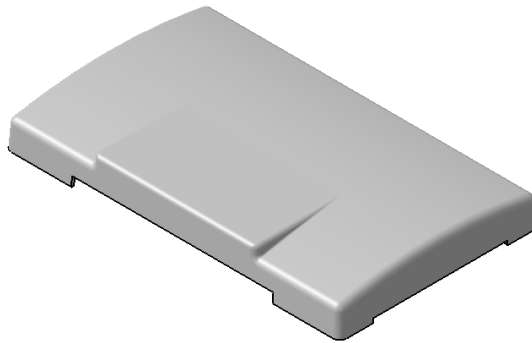


Figure 6-90 Model for Tutorial 2

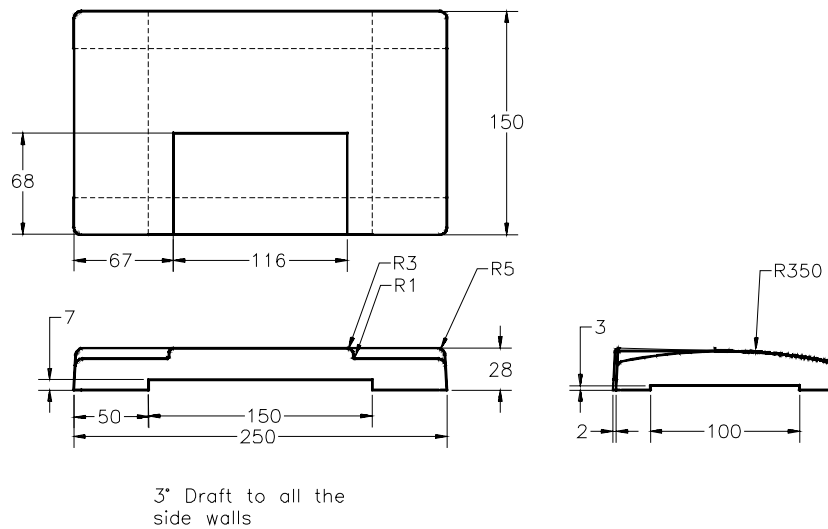


Figure 6-91 Views and dimensions for Tutorial 2

The following steps are required to complete this tutorial:

- Create the base feature of the model by extruding the sketch drawn on the zx plane equally on both the sides of the sketch plane, refer to Figures 6-92 and 6-93.
- Create the second feature by extruding the sketch drawn on a plane created at an offset distance from the xy plane, refer to Figures 6-94 and 6-95.

- c. Add the draft feature to all faces of the model except the upper and lower faces, refer to Figure 6-96.
- d. Fillet the edges of the model, refer to Figures 6-97 through 6-102.
- e. Shell the model using the **Shell** tool by removing the bottom face of the model, refer to Figures 6-103 and 6-104.
- f. Create two pocket features to complete the model, refer to Figure 6-105.

Creating the Base Feature of the Model

First, you need to create the base feature of the model by extruding the sketch drawn on the zx plane. The sketch will be extruded equally on both the sides of the sketching plane using the **Mirrored extent** option.

1. Start a new part file. Select the zx plane as the sketching plane and invoke the **Sketcher** workbench.
2. Draw the sketch of the base feature, as shown in Figure 6-92, and exit the **Sketcher** workbench.
3. Invoke the **Pad Definition** dialog box and set the value of the **Length** spinner to **125**.
4. Select the **Mirrored extent** check box and choose the **OK** button from the **Pad Definition** dialog box. The model, after creating the base feature, is shown in Figure 6-93.

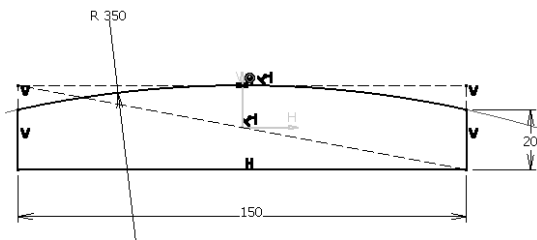


Figure 6-92 Sketch of the base feature

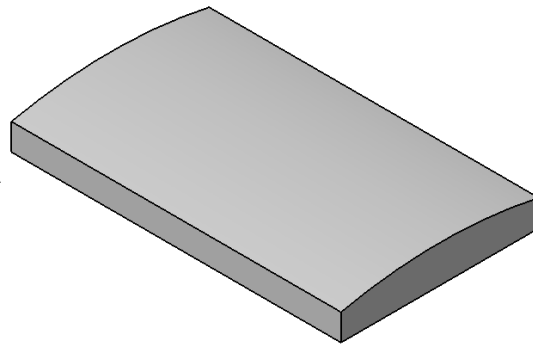


Figure 6-93 The model after creating the base feature

Creating the Second Feature

The second feature of the model will be created by extruding the sketch drawn on a plane created at an offset distance of 14 from the xy plane.

1. Create a plane at an offset distance of 14 mm from the xy plane.
2. Invoke the **Sketcher** workbench using the newly created plane as the sketching plane.
3. Draw the sketch, as shown in Figure 6-94, and exit the **Sketcher** workbench.

4. Invoke the **Pad Definition** dialog box and choose the **Reverse Direction** button.
5. Select the **Up to next** option from the **Type** drop-down list and exit the **Pad Definition** dialog box. The model, after creating the second feature, is shown in Figure 6-95.

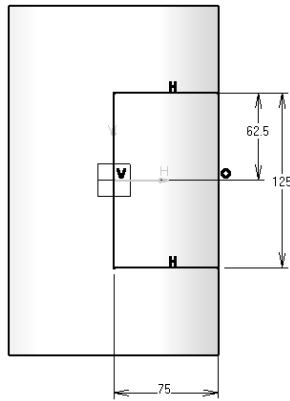


Figure 6-94 Sketch of the second feature

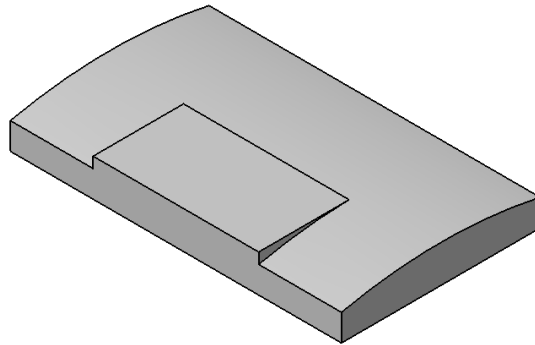



Figure 6-95 The model after creating the second feature


Adding a Draft to the Faces of the Model

Next, you need to add a draft to the faces of the model. The draft angle is added to make sure that the component is smoothly ejected from the die. The draft angle is one of the most important aspects of designing the components to be formed, molded, or cast.

1. Choose the **Draft Angle** button from the **Dress-Up Features** toolbar; the **Draft Definition** dialog box is displayed and you are prompted to select the faces to be drafted. 
2. Select all the vertical faces of the base feature and the second feature from the geometry area.
3. Click once in the **Selection** selection area in the **Neutral Element** area and select the bottom face of the base feature as the neutral element. Make sure that the pulling direction is in the upward direction.
4. Set the value of the **Angle** spinner to **3** and choose the **OK** button from the **Draft Definition** dialog box. The model, after creating the draft feature, is shown in Figure 6-96.

Filleting the Edges of the Model

Next, you need to fillet the edges of the model. In this model, you need to fillet three separate set of edges using the **Edge Fillet** tool.

1. Choose the **Edge Fillet** button from the **Dress-Up Features** toolbar; the **Edge Fillet Definition** dialog box is displayed. 

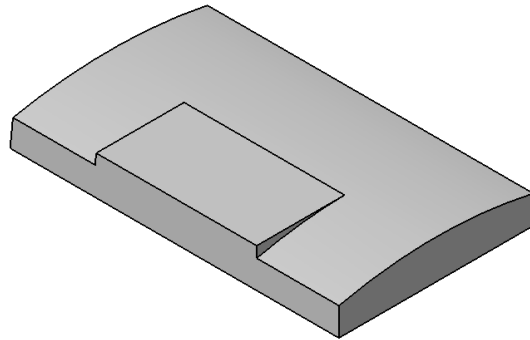


Figure 6-96 The model after drafting all the vertical faces

2. Select the edges, as shown in Figure 6-97 and set the value of the **Radius** spinner to **3**.
3. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The model, after filleting the first set of edges, is shown in Figure 6-98.

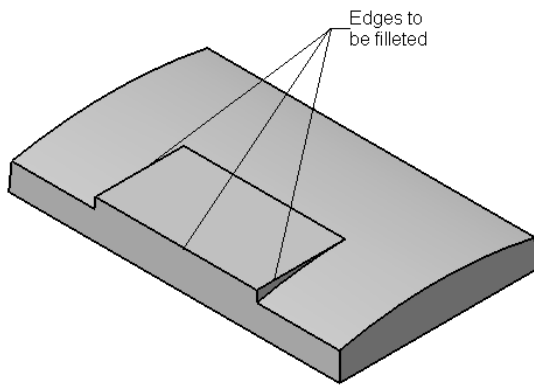


Figure 6-97 Edges to be filleted

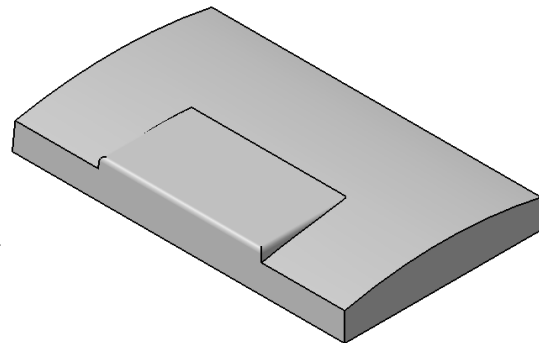


Figure 6-98 The model after filleting the first set of edges

4. Invoke the **Edge Fillet Definition** dialog box again to fillet the second set of edges.
5. Select the edge shown in Figure 6-99 and set the value of the **Radius** spinner to **1**.
6. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The model, after filleting the second set of edges, is shown in Figure 6-100.
7. Invoke the **Edge Fillet Definition** dialog box again to fillet the third set of edges.
8. Select all the edges of the model, except the edges shown in Figure 6-101, and set the value of the **Radius** spinner to **5**.

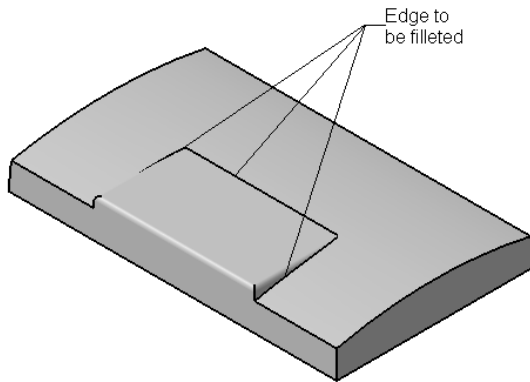


Figure 6-99 Edge to be filleted

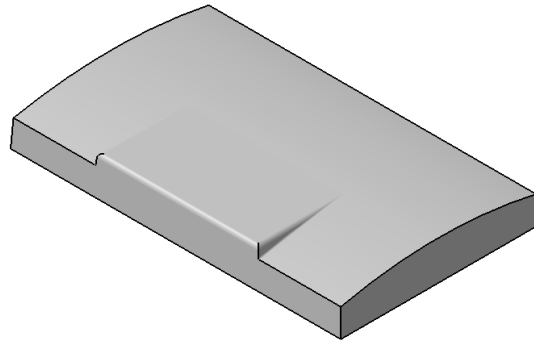


Figure 6-100 The model after filleting the second set of edges

9. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The resulting filleted model is shown in Figure 6-102.

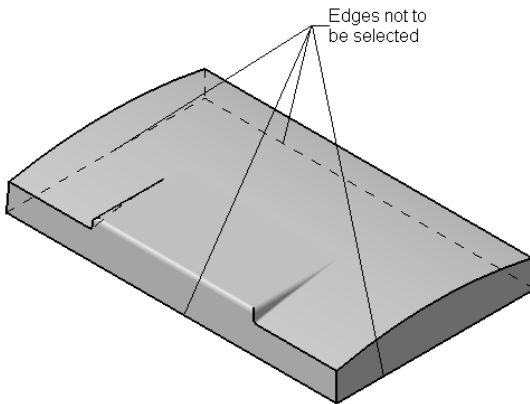


Figure 6-101 Edges not to be selected

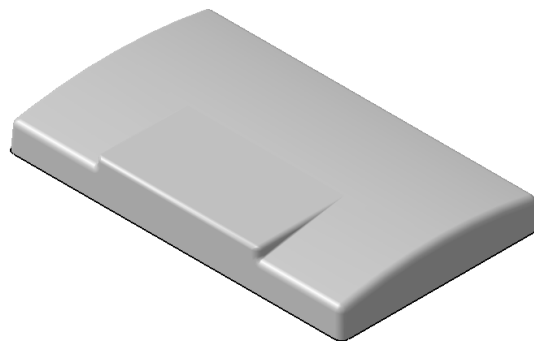



Figure 6-102 Resulting filleted model

Creating the Shell Feature

Finally, you need to shell the model and remove its bottom face.

It is always recommended to shell the model after adding the draft angle and the fillet feature to maintain the draft angle and fillet curvature on the inside walls of the shelled model.

1. Choose the **Shell** button from the **Dress-Up Features** toolbar; the **Shell Definition** dialog box is displayed. 
2. Select the face to be removed, as shown in Figure 6-103, and set the value of the **Default inside thickness** spinner to **2**.

3. Choose the **OK** button from the **Shell Definition** dialog box. The rotated view of the model, after adding the shell feature, is shown in Figure 6-104.

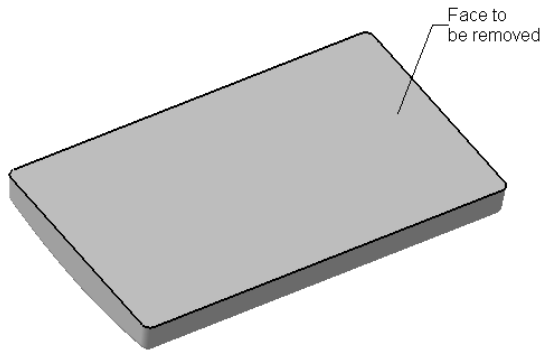


Figure 6-103 Face to be removed



Figure 6-104 Resulting shelled model

4. Use the **Pocket** tool to create two pocket features. The final model, after creating the other two features, is shown in Figure 6-105.

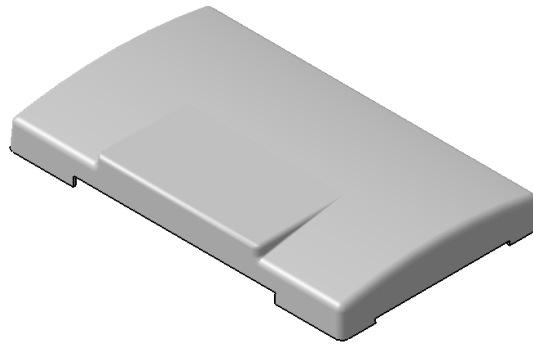


Figure 6-105 Final model after creating the remaining features

Saving and Closing the File

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.
2. Enter the name of the file as *c06tut2* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c06* folder.
3. Close the part file by choosing **File > Close** from the menu bar.

SELF-EVALUATION TEST

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

1. While creating a hole using the **Hole** tool, you can also apply a hole callout to display the hole tolerance. (T/F)
2. You can create a countersunk hole using the **Hole** tool. (T/F)
3. You can add user-defined thread standards for creating a threaded hole. (T/F)
4. You cannot set the limits of the fillet along the selected edge. (T/F)
5. Instead of selecting or creating a limiting element, you can also specify the limit of the fillet by directly selecting the points on the edge to fillet. (T/F)
6. The _____ tool is used to create the draft feature using the silhouette lines of the selected curved face as the neutral element.
7. The _____ tool is used to scoop out the material from the model and remove the selected faces, resulting in a thin-walled structure.
8. By default, the pulling direction is selected in the _____ axis of the selected neutral face while creating the draft feature.
9. The _____ tool is used to apply a fillet between the selected faces of the model.
10. _____ is defined as a process in which the sharp edges are bevelled in order to reduce the area of stress concentration.

REVIEW QUESTIONS

Answer the following questions:

1. You can choose the _____ option in the **Edge Fillet Definition** dialog box to trim the intersecting surfaces.
2. The _____ fillet is created when three or more than three edges are merged into a vertex.
3. You cannot create a counterdrilled hole using the **Hole** tool. (T/F)
4. You cannot apply a different shell thickness value to the faces of the model while creating the shell feature. (T/F)
5. To create an edge fillet, choose the **Face-Face Fillet** button from the **Fillet** toolbar. (T/F)

6. Which tool is used to taper the faces of the model?
- (a) **Draft Angle** (b) **Edge Fillet**
(c) **Chamfer** (d) **Shell**
7. When you define **Up To Plane** or **Up To Surface** as the feature termination condition of a hole feature, then which option is selected automatically from the drop-down list in the **Bottom** area of the **Extension** tab?
- (a) **Extend** (b) **Edge Fillet**
(c) **Trimmed** (d) **Tangent**
8. Which tool is used to create a fillet feature tangent to three faces?
- (a) **Face-Face Fillet** (b) **Variable Radius Fillet**
(c) **Tritangent Fillet** (d) **Edge Fillet**
9. Which tab of the **Hole Definition** dialog box is used to define the parameters to create a tapped hole?
- (a) **Extension** (b) **Type**
(c) **Hole** (d) **Thread Definition**
10. Which tool is used to create a variable angle draft?
- (a) **Draft Angle** (b) **Draft Reflect Line**
(c) **Face-Face Fillet** (d) **None of these**

EXERCISES

Exercise 1

Create the model of the Clutch Lever shown in Figure 6-106. Its views and dimensions are shown in Figure 6-107.

(Expected time: 30 min)

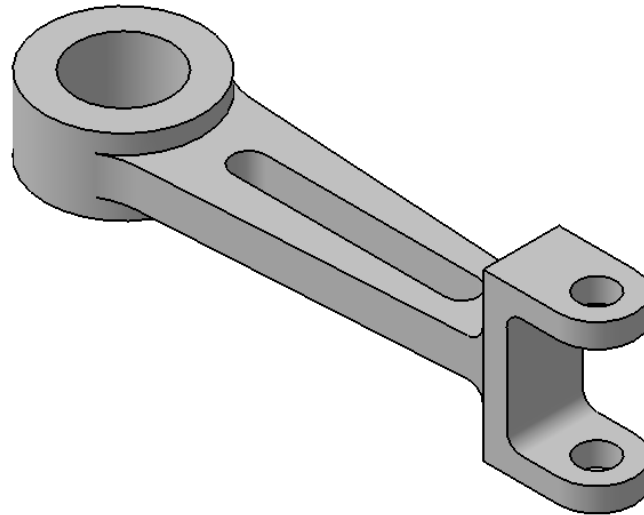


Figure 6-106 Model for Exercise 1

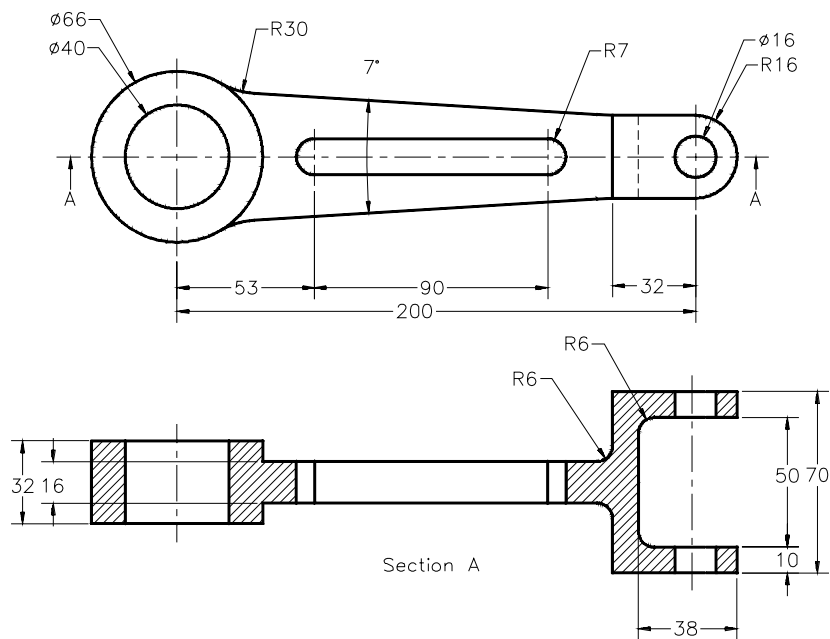
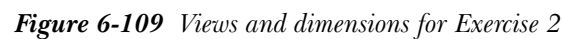
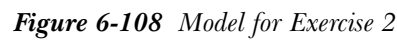


Figure 6-107 Views and dimensions for Exercise 1

(Expected time: 1 hr)



1. T, 2. T, 3. T, 4. F, 5. T, 6. Draft Reflect Line, 7. Shell, 8. Z, 9. Face-Face Fillet, 10. Chamfering