



Chapter 5

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 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, a tapered hole, a counterbored hole, a 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 a plane. Select the face or the plane from the geometry area on which you need to place the hole. The preview of the hole feature with the default values is displayed, along with the **Hole Definition** dialog box. The **Hole Definition** dialog box is shown in Figure 5-1.

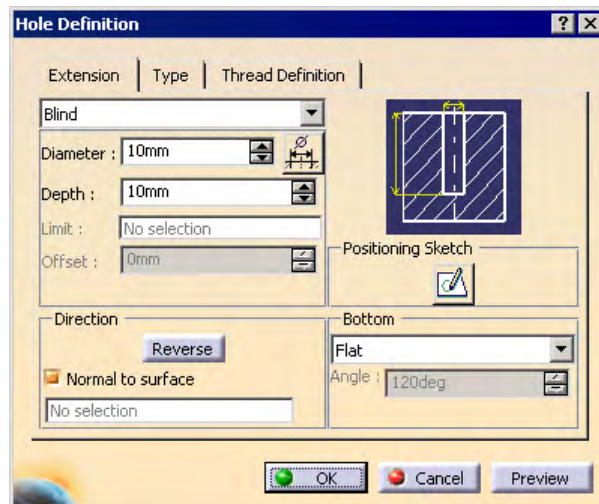


Figure 5-1 The **Hole Definition** dialog box

Creating a Simple Hole

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

creation using the **Reverse** button available 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 and will not be available if you select the **Up To Next** or the **Up To Last** termination types. For other termination types, if you select the **Flat** option, the resulting hole will be a 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 the **Up To Last** termination types, you can also use the trimmed bottom. Figure 5-2 shows all the three types of bottom options.

After setting the hole parameters, choose the **OK** button from the **Hole Definition** dialog box to create a simple hole. Figure 5-3 shows a base plate after creating simple holes using the **Hole** tool.

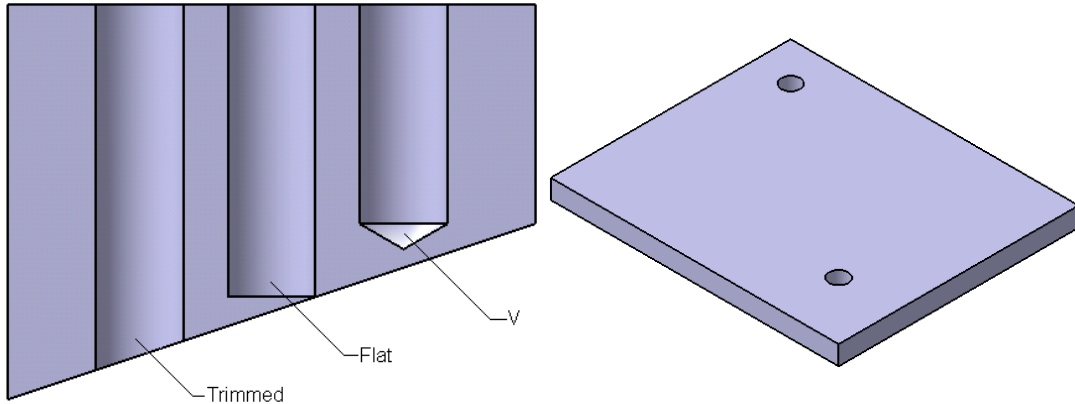


Figure 5-2 Types of bottom termination options **Figure 5-3** Base plate with holes created using the **Hole** tool



Tip. While creating a hole using the **Hole** tool, you can also apply a hole callout to display the hole tolerance. Choose the **Hole Tolerance Callout** 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 hole tolerance using the options available 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** and the information about the hole tolerance callout is displayed in it.

Creating a Threaded Simple Hole

To create the threaded hole, invoke the **Thread Definition** tab from the **Hole Definition** dialog box. By default, the **Threaded** radio button is cleared. Select the **Threaded** radio button to invoke the options available in the **Thread Definition** tab, as shown in Figure 5-4.

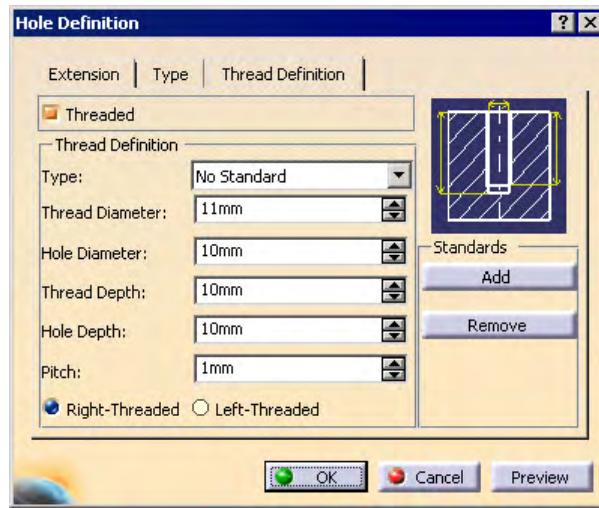


Figure 5-4 The Hole Definition dialog box after selecting the Threaded radio button

By default, the **No Standard** option is selected in the **Type** drop-down list available in the **Thread Definition** area. Therefore, you need to manually specify all 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 set 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 of the thread 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 all parameters, choose the **OK** button from the **Hole Definition** dialog box. A threaded hole is created. Note that the thread is not displayed in the hole because only a cosmetic thread is added to the hole feature. When you generate the drawing view, the thread conversion will be displayed in it. You will learn more about generating the drawing views in the later chapters.

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

Creating a Tapered Hole

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

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.

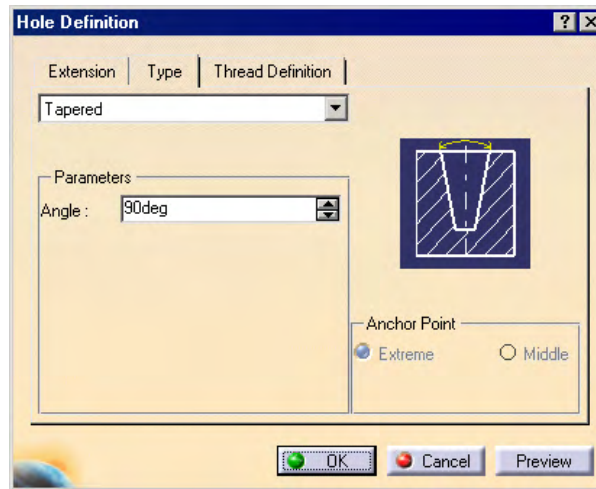


Figure 5-5 The **Hole Definition** dialog box after selecting the **Tapered** option from the drop-down list in the **Type** tab



Tip. You can also add the user-defined thread standards for creating a threaded hole. To add a user-defined thread standard, choose 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 available 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 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. In this type of hole, you will also have to specify two depths: the counterbore depth and the 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 5-6 shows the sectional view of a counterbore hole and Figure 5-7 shows a base plate with counterbored holes.

To create a counterbored hole, select the **Counterbored** option from the drop-down list available in the **Type** tab of the **Hole Definition** dialog box, as shown in Figure 5-8. The preview of counterbored hole is displayed in the geometry area. You can set the value of the counter diameter by using the **Diameter** spinner available in the **Parameters** area. Set the value of the counter depth using the **Depth** spinner. You can set the diameter and the depth of the hole using the options available in the **Extension** 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 bore will be placed on the selected placement plane. You can define the thread parameters also for a

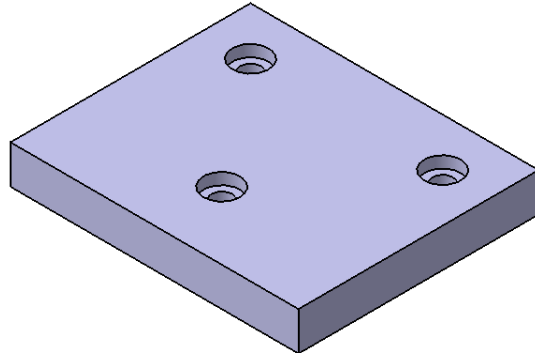
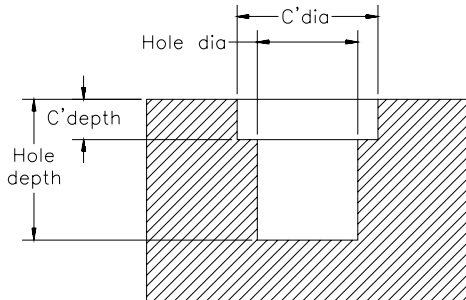


Figure 5-6 Sectional view of counterbored hole

Figure 5-7 Base plate with counterbored holes

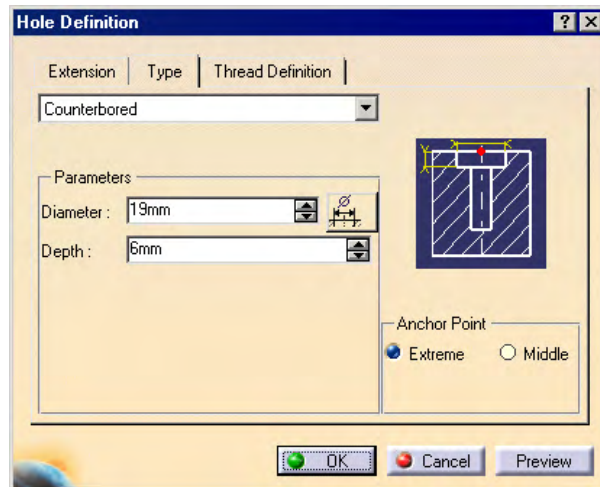


Figure 5-8 The **Hole Definition** dialog box after selecting the **Counterbored** option from the drop-down list in the **Type** tab

counterbored hole. After setting all 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 type of hole, you will have to define the countersunk diameter, the hole diameter, the depth of the hole, and the countersink angle. Figure 5-9 shows the sectional view of a countersunk hole. Figure 5-10 shows the spacer plates after adding the countersunk holes.

To create a countersunk hole, select the **Countersunk** option from the drop-down list available in the **Type** tab, as shown in Figure 5-11. Its preview is displayed in the geometry area.

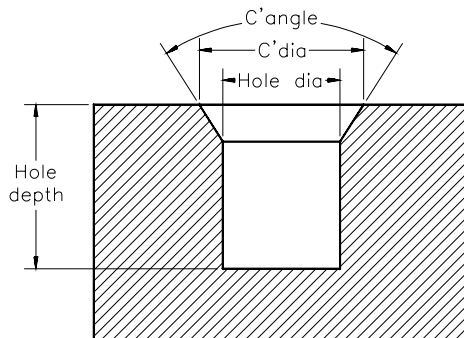


Figure 5-9 Sectional view of countersunk hole

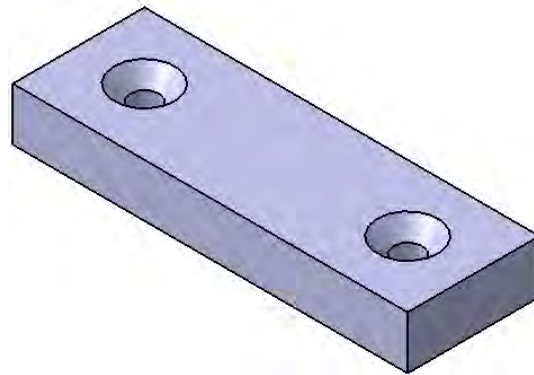


Figure 5-10 Spacer plate with countersunk holes

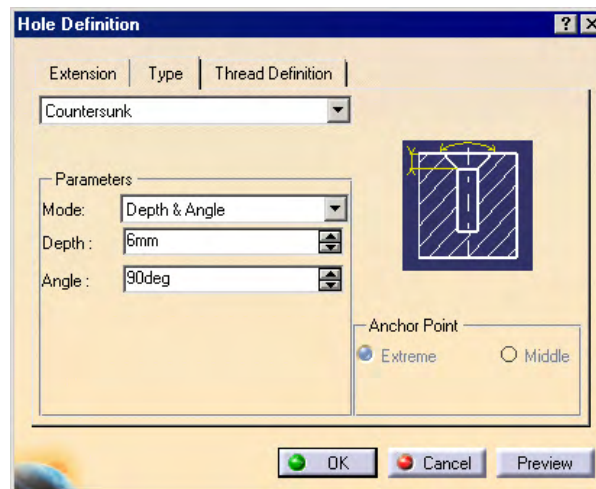


Figure 5-11 The **Hole Definition** dialog box after selecting the **Countersunk** option from the drop-down list in the **Type** tab

You can choose the option for specifying the parameters of countersunk using the **Mode** drop-down list. By default, the **Depth & Angle** option is selected in this drop-down list. Therefore, you need to define the depth and angle of countersunk in the **Depth** and the **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 the **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 all parameters, choose the **OK** button from the **Hole Definition** dialog box.

Creating the Counterdrilled

A counterdrilled hole is a combination of a counterbored and countersunk hole. This type of

hole has two diameters and the transition between the bigger diameter and the smaller diameter after the counterbore depth is in the form of a tapered cone, refer to Figure 5-12. You will have to define the counterbore diameter, the hole diameter, the depth of counterbore, the depth of the hole, and the countersink angle. Figure 5-12 shows the sectional view of a counterdrilled hole. Figure 5-13 shows the spacer plates with counterdrilled holes.

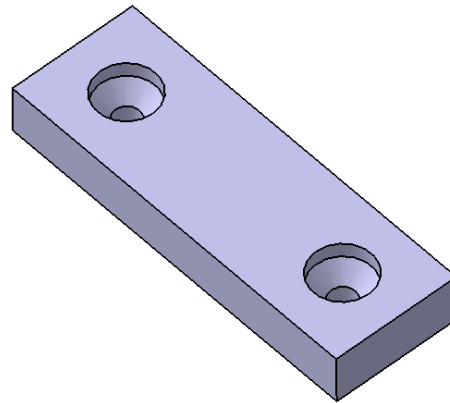
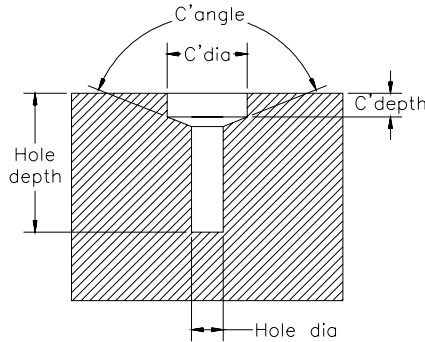


Figure 5-12 Sectional view of counterdrilled hole **Figure 5-13** Spacer plate with counterdrilled holes

To create a counterdrilled hole, select the **Counterdrilled** option from the drop-down list available in the **Type** tab; its preview is displayed in the geometry area. Figure 5-14 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 counter using the **Diameter** spinner and the value of the depth of counter 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.

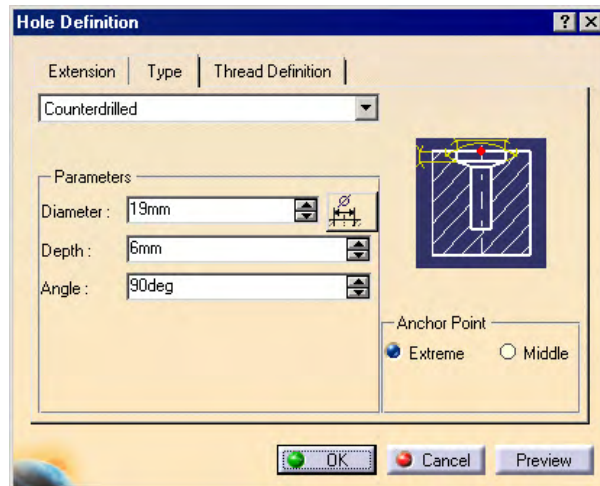


Figure 5-14 The **Hole Definition** dialog box after selecting the **Counterdrilled** option from the drop-down list in the **Type** tab



Tip. If you need to apply cosmetic thread 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 and 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 **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 that it is recommended to use **Hole** command to tap a hole.

Creating Fillets

Fillet is generally provided in order 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 available in the **Part** mode of CATIA V5. Choose the black arrow provided on the right of the **Edge Fillet** button in the **Dress-Up Features** toolbar; the **Fillets** toolbar is invoked as is shown in Figure 5-15.

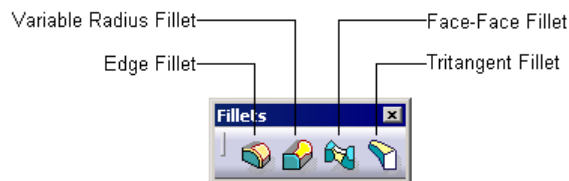


Figure 5-15 The **Fillets** toolbar

The procedure of creating various types of fillets is discussed in the following section.

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 5-16.

You are prompted to select an edge or a face. Select the edge that you need to fillet; the number of the selected edge is displayed in the **Object(s) to fillet** selection area. The default radius value is displayed on the first edge selected. Set the value of the fillet radius using the **Radius** spinner and choose the **OK** button from the **Edge Fillet Definition** dialog box. Figure 5-17 shows the edge selected to be filleted and Figure 5-18 shows the resulting filleted edge.

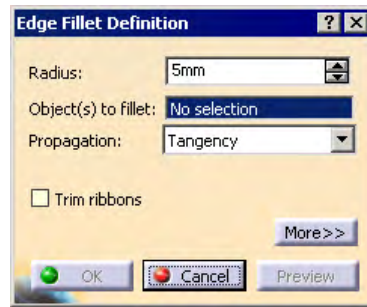


Figure 5-16 The **Edge Fillet Definition** dialog box

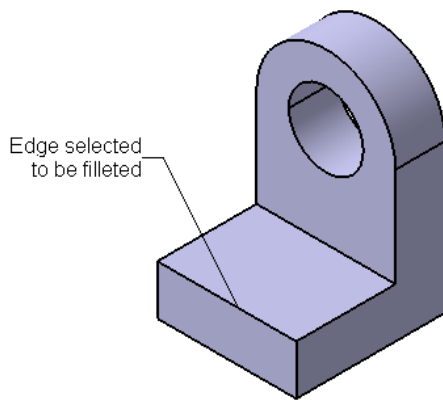


Figure 5-17 Edge selected to be filleted

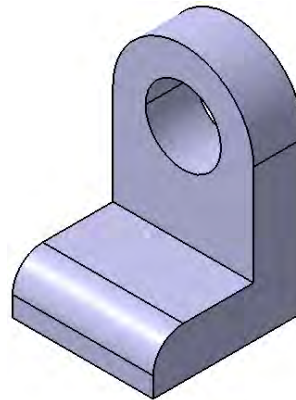


Figure 5-18 Resulting edge fillet

Figure 5-19 shows the face selected to be filleted and Figure 5-20 shows the resulting filleted face.

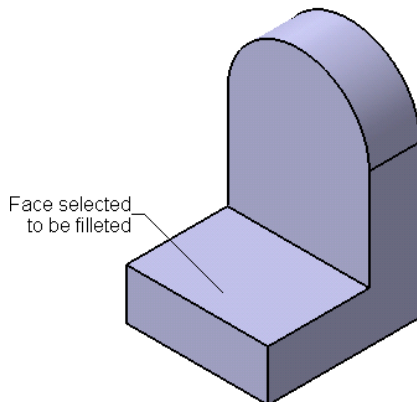


Figure 5-19 Face to be selected

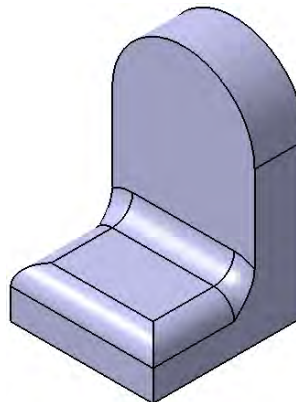


Figure 5-20 Resulting fillet

The options available in the **Edge Fillet Definition** dialog box for creating advance edge fillets are discussed in the following section.

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 5-21 shows the edge to be filleted.

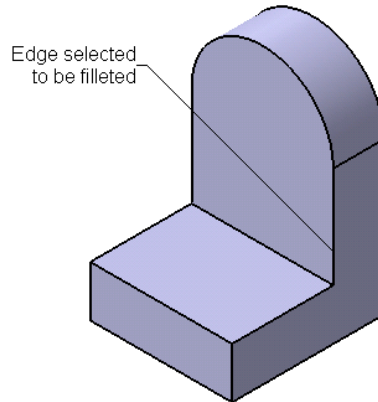


Figure 5-21 Edge selected to be filleted

Figure 5-22 shows the edge filleted using the **Tangent** option and Figure 5-23 shows the edge filleted using the **Minimal** option.

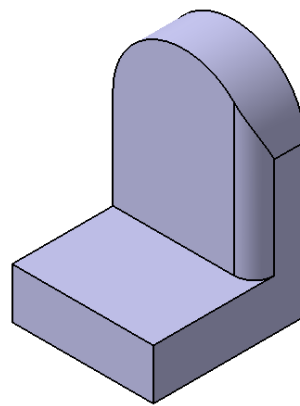
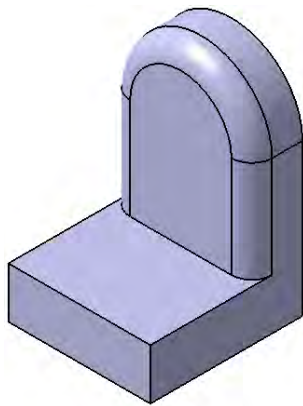


Figure 5-22 Fillet using the tangent propagation Figure 5-23 Fillet using the minimal propagation

Trimming the Overlapping Fillets

You can also use the options available in the **Fillet** tool to trim the intersecting surfaces. Consider the case of the model shown in Figure 5-24. Using the **Fillet** tool, three edges of the model are filleted. If you select the **Trim ribbons** check box available in the **Edge Fillet Definition** dialog box, the intersecting surfaces created, as a result of the fillet, will be trimmed. Figure 5-25 shows the resulting fillet after selecting the **Trim ribbons** check box.

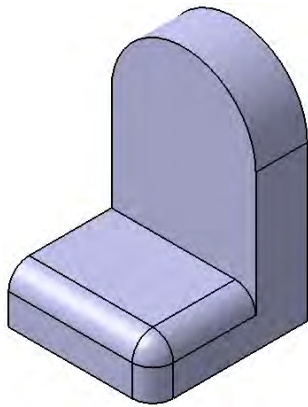


Figure 5-24 Fillet with the **Trim ribbons** check box cleared

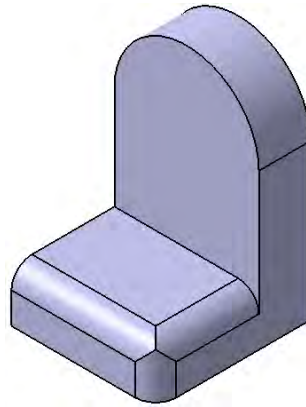


Figure 5-25 Fillet with the **Trim ribbons** check box selected

Selecting the Edges to Keep

Sometimes while filleting the edges, some of them get distorted, in order to accommodate the fillet radius, as shown in Figure 5-26. In this model, the bottom edge of the elliptical extruded feature is filleted. The inclined edges of the model 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 on 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 5-27.

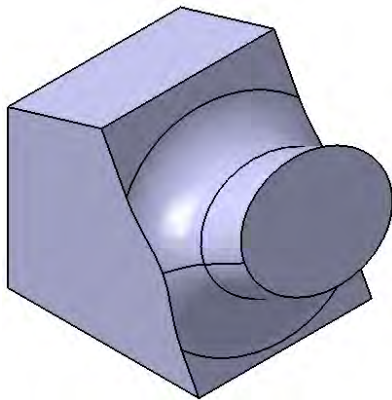


Figure 5-26 Edges distorted to accommodate the fillet radius

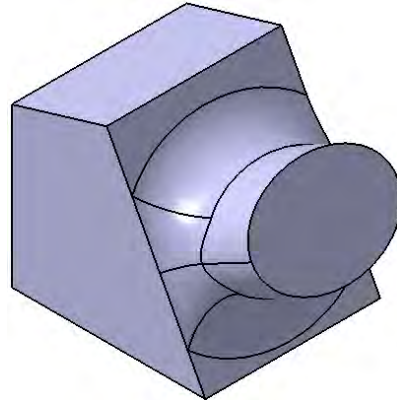


Figure 5-27 Model after selecting the edges to keep



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 limits of the fillet along the selected edge up to which the fillet will be created. Select the edge or edges to fillet and set the value of radius. Now, expand the **Edge Fillet Definition** dialog box using the **More** button. Click once on the **Limiting element(s)** selection area and select the plane up to which you need to create the fillet. An arrow is displayed in the geometry area that defines the direction of fillet creation. You can flip the direction of fillet creation by clicking on that arrow. You can also create a point or a plane within the **Fillet** tool to define the limit of fillet. To create a point or a plane within the **Edge Fillet** tool, right-click on the **Limiting element(s)** selection area; a contextual menu is displayed. Define the limit using the options available in the contextual menu. Figure 5-28 shows the edge to be filleted and the limiting element to be selected. Figure 5-29 shows the resulting fillet.

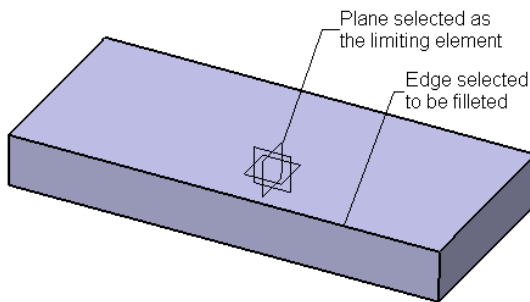


Figure 5-28 Edge to be filleted and the limiting element

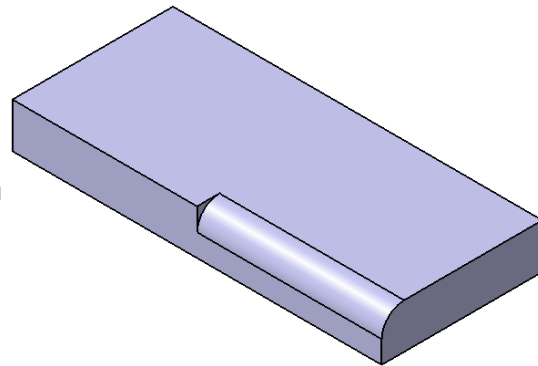


Figure 5-29 Resulting fillet



Note

Instead of selecting or creating a limiting element, you can also specify the limit of fillet by directly selecting points on the edge to fillet. To define the limits using this method, select the edge to fillet and define the radius of fillet. Now, expand the dialog box and click once on the **Limiting element(s)** selection area. Click on the selected edge where you need to define the limit of fillet; a blue circle is displayed on the current selection. The arrow defining the direction of fillet creation is also displayed. If you have selected two elements to limit the fillet, you need to make sure that the arrows of both the limits point in the opposite directions. You can flip the direction of arrows by clicking on them. Figure 5-30 shows the fillet after specifying two limit elements. In this figure, the arrows of both the limits were pointing 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 type of fillet is used to smoothly blend the transition surfaces generated from the edges to the fillet vertex. This smooth transition is created between all selected edges and the selected vertex for the setback type of fillet. To create this type of fillet, select the edges that you need to fillet and then set the value of the fillet radius. Now,

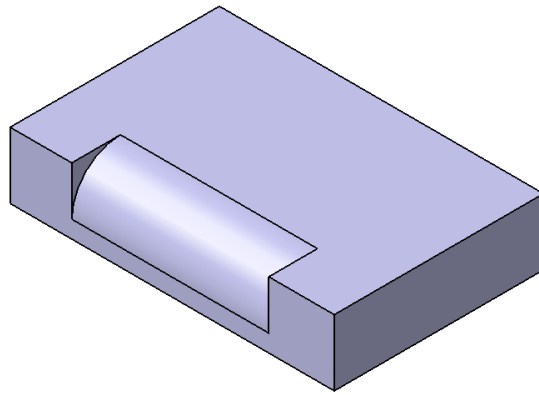


Figure 5-30 Fillet after defining two limit elements

expand the **Edge Fillet Definition** dialog box using the **More** button. Choose the **Blend corner(s)** 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 dimension and set its value in the **Setback distance** spinner. Similarly, set the setback distance for the other edges. Figure 5-31 shows edges selected to be filleted. Figure 5-32 shows the preview of the setback fillet after setting the setback distance. Figure 5-33 shows the resulting setback fillet.

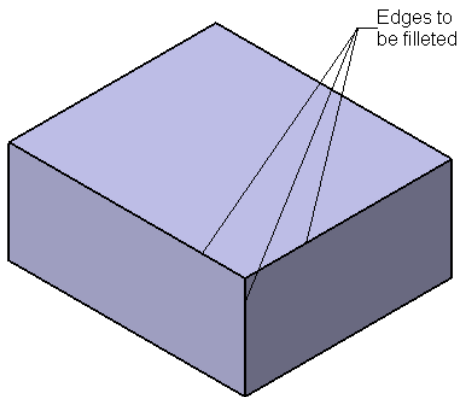


Figure 5-31 Edges to be selected

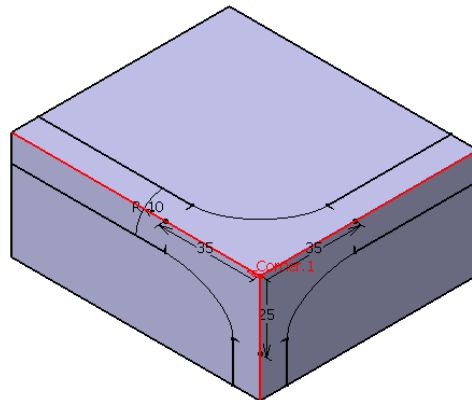


Figure 5-32 Preview of the setback fillet



Note

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

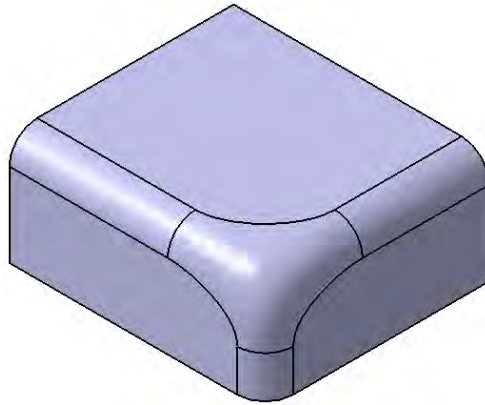


Figure 5-33 Resulting setback fillet

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 by using the **Variable Radius Fillet** tool. Transition of the fillet can be smooth or straight, depending upon the option you select. To create variable radius fillet, choose the **Variable Radius Fillet** button from the **Fillets** toolbar; the **Variable Edge Fillet** dialog box is displayed, as shown in Figure 5-34.

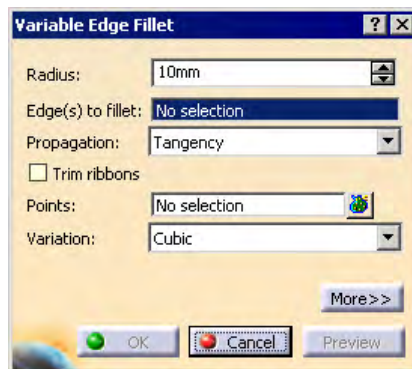


Figure 5-34 The **Variable Edge Fillet** dialog box

You are prompted to select an edge. Select the edge that you need to fillet; two radius callouts are attached to the endpoints of the selected edge. Select the radius callout and set the value of the radius in the **Radius** spinner. Similarly, select the other callout and set the value of the second radius in the **Radius** spinner. Now, choose the **OK** button from the **Variable Edge Fillet** dialog box. The model, after creating variable radius fillet, is shown in Figure 5-35.

You can also define additional control points on the selected edge by clicking on the **Points** selection area and right-clicking to invoke the contextual menu. Create an additional point or a plane and control its distance from the start point by using the options available in this contextual menu. You can create as many control points as you need by repeating this procedure. Now, set the value of the radius for the newly created control points. Figure 5-36 shows a variable radius fillet, after specifying radii at additional control points.

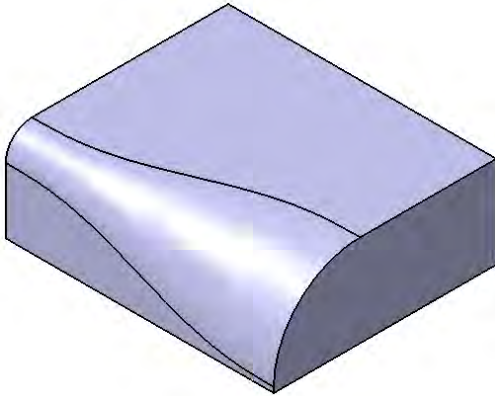


Figure 5-35 Variable radius fillet created by specifying radii to both the end points of the edge

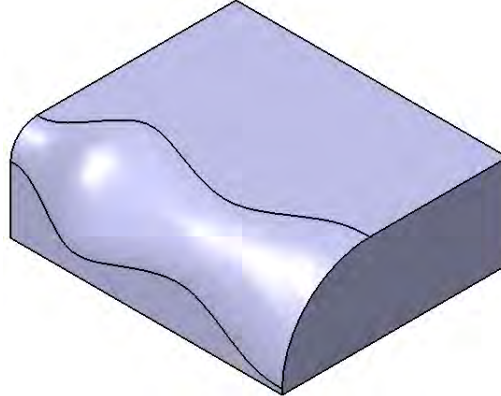


Figure 5-36 Variable radius fillet after defining additional control points

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 **Variable Edge Fillet** dialog box. This option will result in the smooth transition of fillet the surface. If you select the **Linear** option from the **Variation** drop-down list, it results in straight transition of the fillet surface. Figure 5-37 shows the variable radius fillet with the **Cubic** option selected and Figure 5-38 shows the variable radius fillet with the **Linear** option selected.

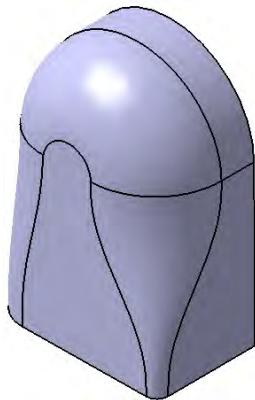


Figure 5-37 Variable radius fillet with the **Cubic** option selected

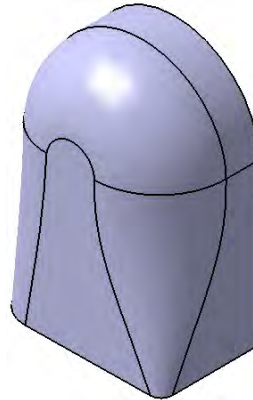


Figure 5-38 Variable radius fillet with the **Linear** option selected

Creating Face-Face Fillets

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



The **Face-Face Fillet** 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 is displayed, as shown in Figure 5-39.

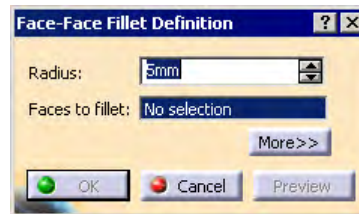


Figure 5-39 The **Face-Face Fillet Definition** dialog box

You are prompted to select a face. Select the first and second faces from the geometry area. Set the value of the radius of 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 radius value of the fillet, after exiting this window. Figure 5-40 shows the faces to be selected to create the face-face fillet and Figure 5-41 shows the resulting face-face fillet.

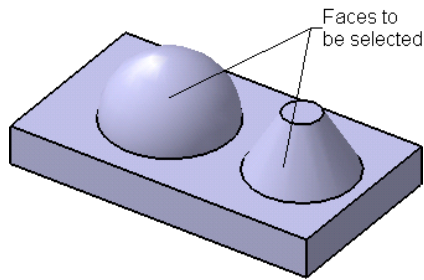


Figure 5-40 Faces to be selected

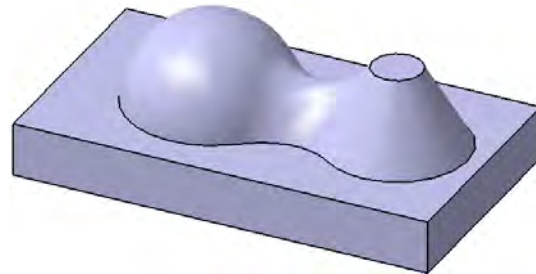


Figure 5-41 Resulting face-face fillet

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, which 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 is displayed, as shown in Figure 5-42.



Figure 5-42 The *Tritangent Fillet Definition* dialog box

You are prompted to select the first face. Hence, do so; you are prompted to select the second face. Upon doing so, you are prompted to select a face to be removed. Select the face to be removed from the geometry area, refer to Figure 5-43. Choose the **Preview** button from the **Tritangent Fillet Definition** dialog box to preview the tritangent fillet. Figure 5-43 shows the faces to be selected and Figure 5-44 shows the resulting tritangent fillet.

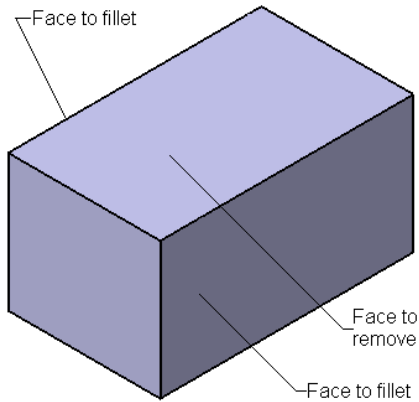


Figure 5-43 Faces to be selected

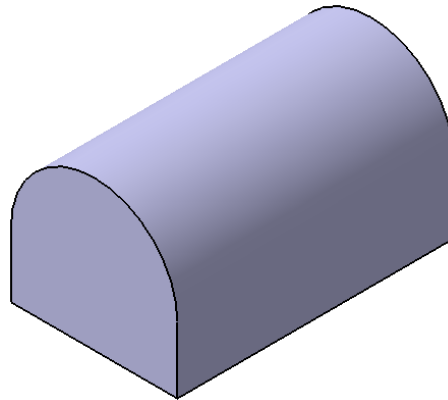


Figure 5-44 Resulting tritangent fillet

Creating Chamfers

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



Chamfering is defined as a process in which the sharp edges are bevelled in order to reduce stress concentration in the model. This process also eliminates the sharp edges that are not desirable. To chamfer the edges of the model, choose the **Chamfer** button from the **Dress-Up Features** toolbar; the **Chamfer Definition** dialog box is displayed, as shown in Figure 5-45.

You are prompted to specify the required data to define the chamfer. First, you need to select 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 is displayed in the **Object(s) to chamfer** selection area.

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 chamfer and the angle of

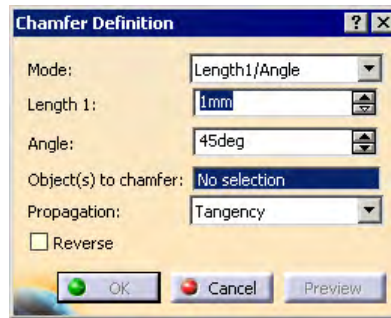


Figure 5-45 The Chamfer Definition dialog box

chamfer in the **Length 1** and the **Angle** spinners, respectively. If you select the **Length1/Length2** option from the **Mode** drop-down list, you need to define the value of the first and second lengths of chamfer in the **Length 1** and the **Length 2** spinners.

If you need to chamfer all the edges tangent to the selected edges, select the **Tangency** option from the **Propagation** drop-down list. If you need 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 5-46 shows the edge selected to be chamfered and Figure 5-47 shows the resulting chamfer.

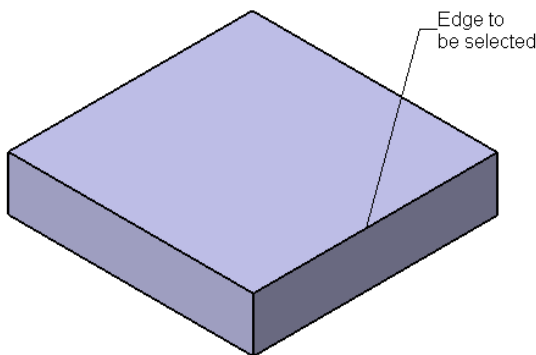


Figure 5-46 Edge to be selected

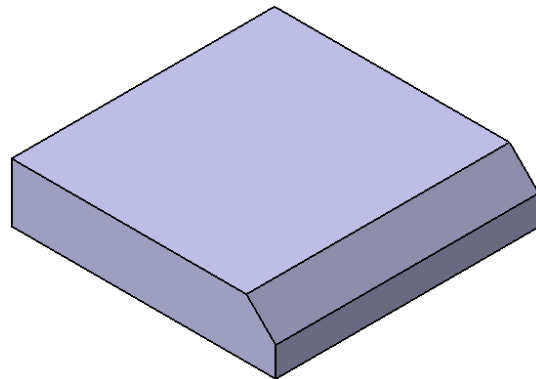


Figure 5-47 Resulting chamfer

Adding a Draft to the Faces of the Model

Draft is defined as the process of adding a taper angle to the faces of the model. Adding draft to the faces of the model is one of the most important operations, especially, while creating the components that needs to be casted, 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 options are discussed in the following section.

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 draft the faces of the model. To draft the faces of the model using this tool, choose the **Draft Angle** button from the **Drafts** toolbar; the **Draft Definition** dialog box is displayed, as shown in Figure 5-48. Also, an arrow is displayed at the origin and points in the default pull direction.

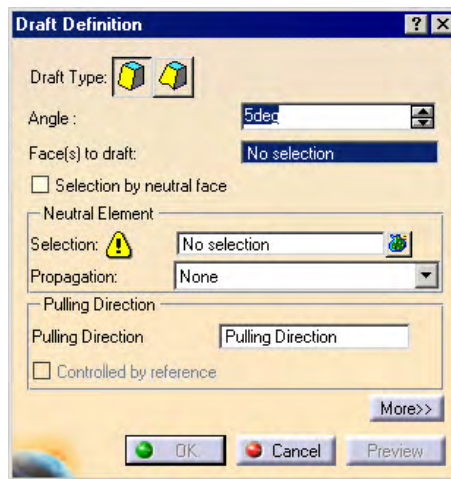


Figure 5-48 The Draft Definition dialog box

You are prompted to select the faces to draft. 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. Next, you need to define a neutral plane. Click once on the **Selection** selection area provided in the **Neutral Element** area and then select a face or a 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 5-49 shows the faces to draft and the face to be selected as the neutral plane and Figure 5-50 shows the resulting drafted faces.

Figure 5-51 shows the mid-plane to be selected as neutral plane and Figure 5-52 shows the resulting drafted faces.



Tip. If you wish to add draft to all the 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.

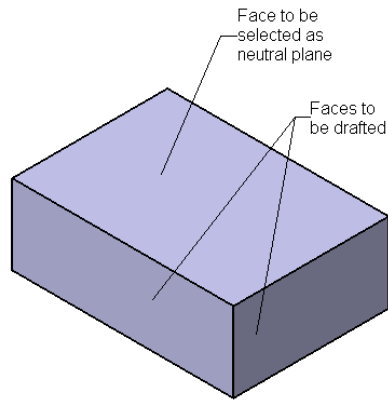


Figure 5-49 Faces to be selected

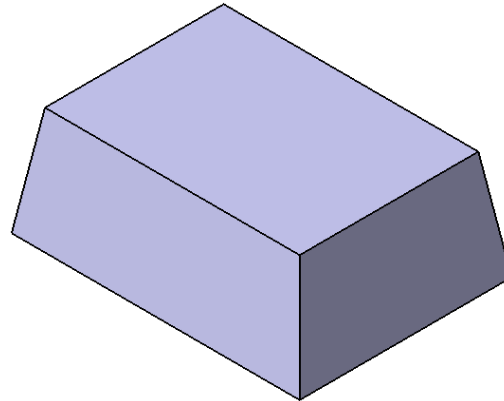


Figure 5-50 Resulting drafted faces

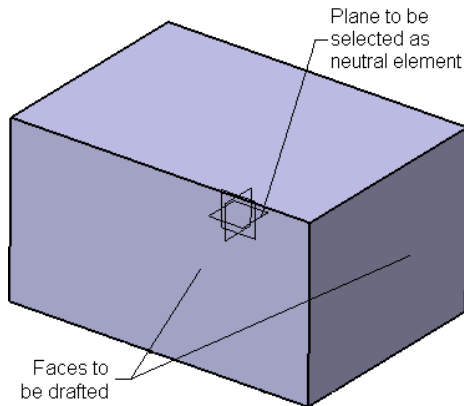


Figure 5-51 Faces and planes to be selected

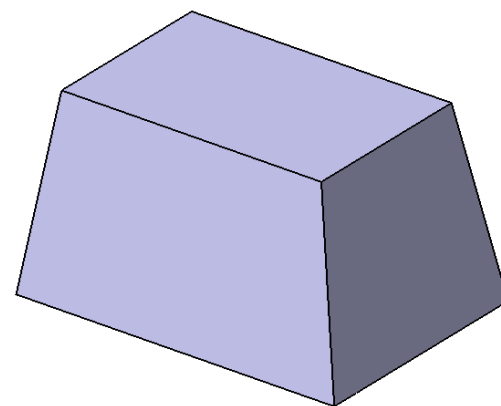


Figure 5-52 Resulting drafted faces

Defining the Parting Element While Adding Draft 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 5-51, in which a plane passing through the center of the model is selected as the neutral plane. Figure 5-53 shows faces drafted with the **Parting = Neutral** check box selected. When you select the **Parting = Neutral** check box, the **Draft both sides** check box is invoked. If you select this check box, the draft is added to both the sides of the parting element, refer to Figure 5-54.

You can also select user-defined parting element other than the neutral plane. To select a user-defined parting element, select the **Define parting element** button 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 5-55 shows the faces to draft, neutral plane, and the parting plane. Figure 5-56 shows the resulting drafted faces.

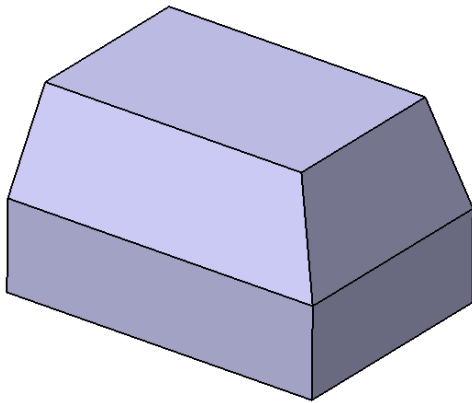


Figure 5-53 Faces drafted with the **Parting = Neutral** check box selected.

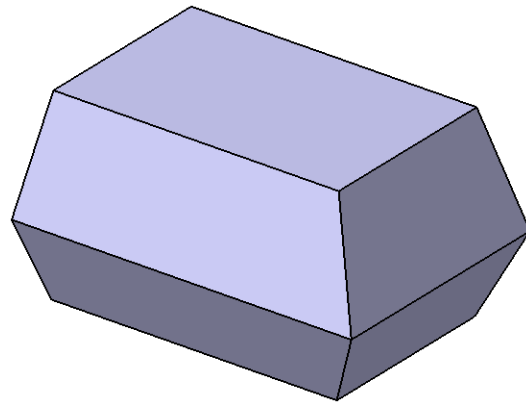


Figure 5-54 Faces drafted with the **Draft both sides** check box selected

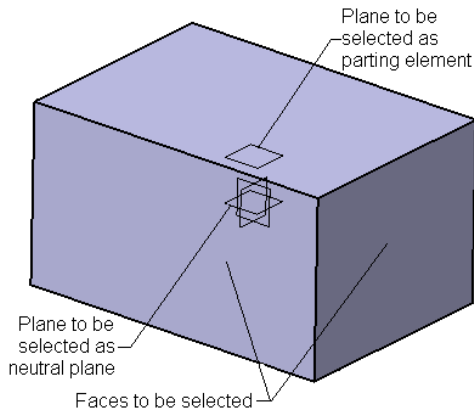


Figure 5-55 References to be selected

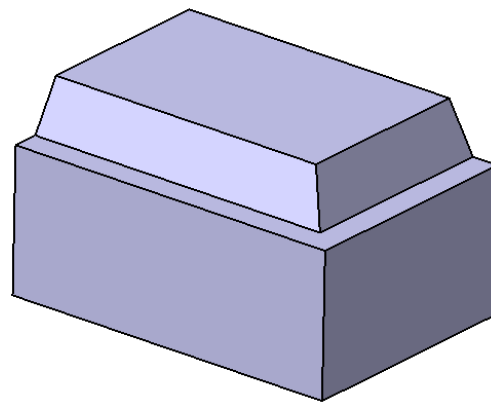


Figure 5-56 Resulting drafted faces

You can also define the limits elements, while adding draft to faces of the model. To define do so, click once on 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 element, the direction of feature creation is opposite. Figure 5-57 shows the limiting elements to be selected and Figure 5-58 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 on the **Pulling Direction** selection area and then selecting the pulling direction from the geometry area.

Adding Draft using Reflect Line

Toolbar: Dress-Up Features > Drafts > Draft 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

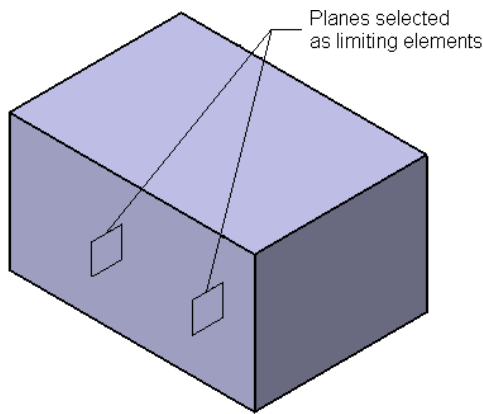


Figure 5-57 Limiting faces to be selected

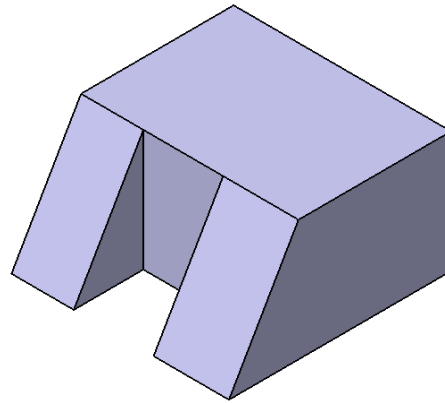


Figure 5-58 Resulting drafted face

feature, choose the **Draft Reflect Line** button from the **Drafts** toolbar; the **Draft Reflect Line Definition** dialog box is displayed, as shown in Figure 5-59.

Select a curved face from the geometry area. The faces tangent to the selected face are also selected automatically. You will notice that a pink color sketch is 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 the 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 5-60 shows the face to add the draft and the plane to be selected as the parting element. Figure 5-61 shows the resulting draft feature.

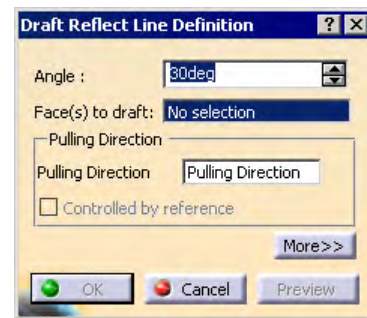


Figure 5-59 The **Draft Reflect Line Definition** dialog box

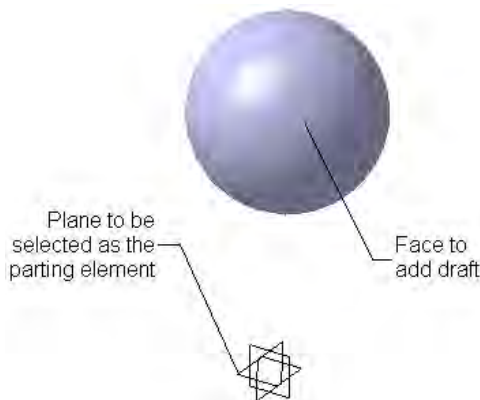


Figure 5-60 Face and plane to be selected



Figure 5-61 Resulting draft feature

Adding Variable Angle Draft

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



To create a variable angle draft, choose the **Variable Angle Draft** from the **Drafts** toolbar. The **Draft Definition** dialog box is displayed, as shown in Figure 5-62.

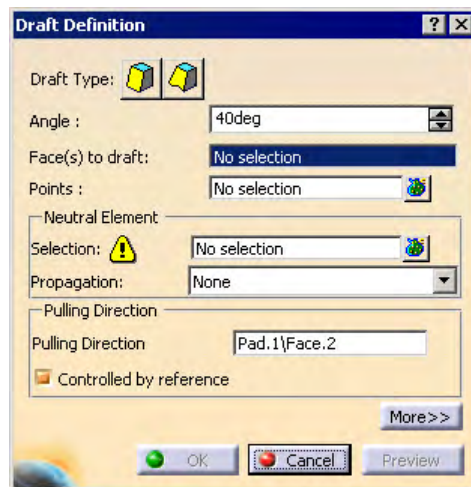


Figure 5-62 The *Draft Definition* dialog box

Select the face on which you need to add the draft. You can select only one face, while adding draft using this tool. Define the neutral element by selecting a plane or a 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. Figure 5-63 shows the references to be selected and Figure 5-64 shows the resulting face, after adding draft.

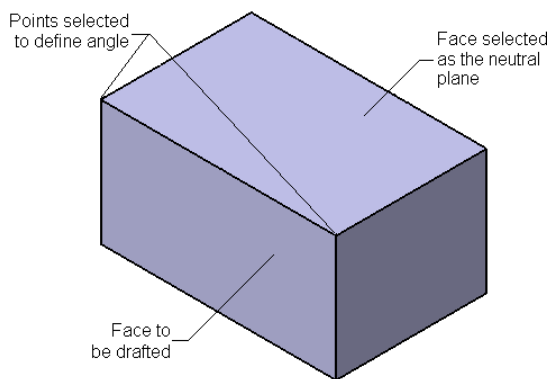


Figure 5-63 References to be selected

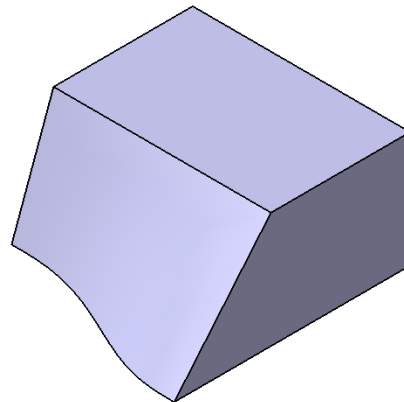


Figure 5-64 Face after adding draft

You can also define additional points to specify other variable angles. Note that the point can only be selected on the edge from the which the angle is measured. To define an additional

point, click anywhere on the edge from which the angle is measured. If you want to define points whose distances need to be controlled, right-click on the **Points** selection area and invoke the contextual menu. Create additional points and then set the draft angle by using the options available in the contextual menu.

Creating a Shell Feature

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



Shell tool is used to scoop out the material from the model and remove the selected faces, which will result 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 is displayed, as shown in Figure 5-65.



Figure 5-65 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. Set the value of the wall thickness in the **Default inside thickness** spinner available 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 5-66 shows the faces to be removed and Figure 5-67 shows the resulting shelled model. If you do not select any of the faces to be removed, the resulting shelled model will be a hollow model with a specified wall thickness.

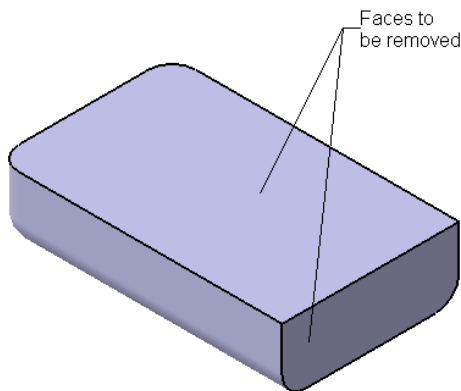


Figure 5-66 Faces to be selected to remove

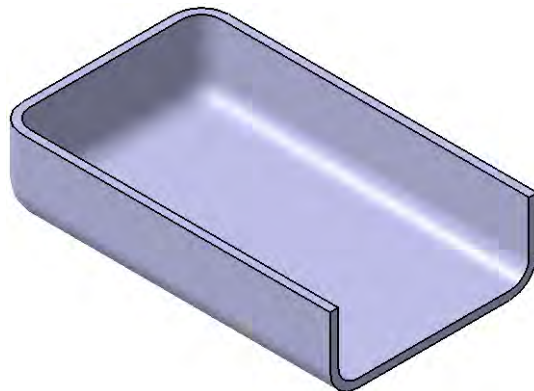


Figure 5-67 Resulting shelled model

Creating Multithickness Shell

You can also define different shell thickness values to 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 on the **Other thickness faces** selection area and then select the faces on which you need to define different shell thickness. The faces tangent to the selected face are selected automatically. The selected faces will be highlighted in brown and the shell thickness dimensions are attached to them. Select the thickness value of one of the highlighted face from the geometry area; the selected value is displayed in **Default inside thickness** spinner available in the **Shell Definition** dialog box. Modify the thickness value and repeat the process for the remaining highlighted faces. After setting all the shell thickness values, choose the **OK** button from the **Shell Definition** dialog box. Figure 5-68 shows the face to be removed and the faces to define different shell thickness and Figure 5-69 shows the resulting shelled model.

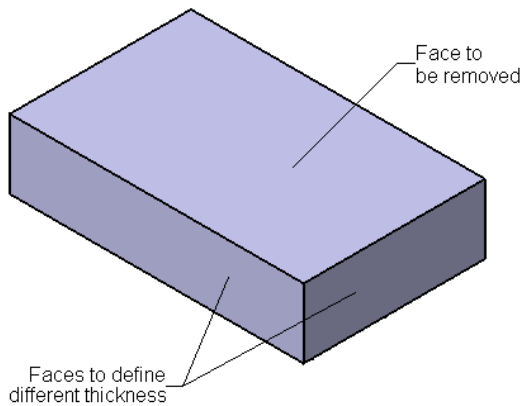


Figure 5-68 Faces to be selected

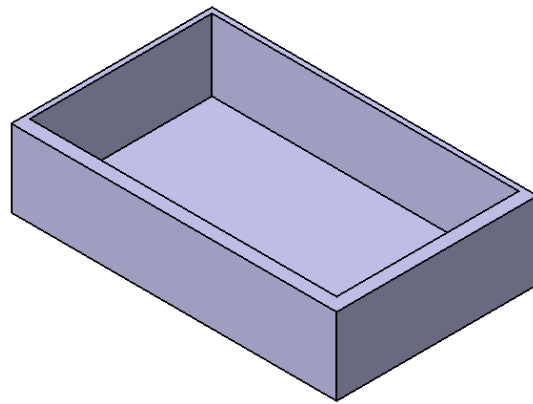


Figure 5-69 Resulting shelled model

TUTORIALS

Tutorial 1

In this tutorial, you will create the model of the nozzle of a vacuum cleaner shown in Figure 5-70. The views and dimensions of this model are shown in Figure 5-71. **(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 5-72 through 5-76.
- Create the second feature of the model by extruding a sketch using the **Drafted Fillet Pad** tool, refer to Figures 5-77 and 5-78.
- 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 5-79 and 5-80.
- Apply fillets to all edges of the model, refer to Figures 5-81 through 5-84.
- Shell the model using the **Shell** tool, refer to Figures 5-85 and 5-86.



Figure 5-70 Model of Vacuum Cleaner for Tutorial 1

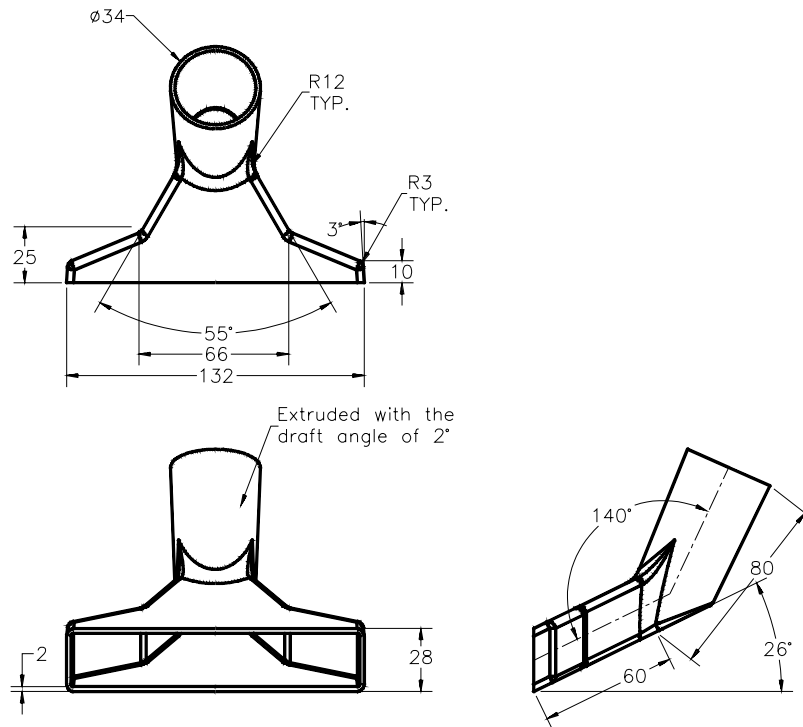


Figure 5-71 Views and dimensions of Vacuum Cleaner for Tutorial 1

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.

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 5-72, and then exit the **Sketcher** workbench.
3. Select the YZ plane and invoke the **Sketcher** workbench. Place a point colinear to the X-axis at any distance, as shown in Figure 5-73. Exit the **Sketcher** workbench.

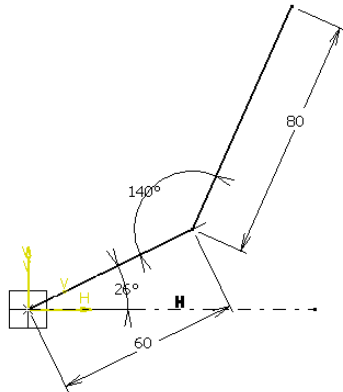


Figure 5-72 Reference sketch

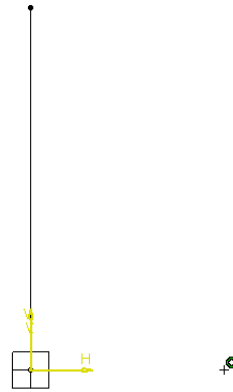


Figure 5-73 Point to be placed

After drawing the reference sketch and placing the point, you need to create a plane that will be used as the reference plane to create the base feature.

4. Create a plane by selecting three points, as shown in Figure 5-74.
5. Invoke the **Sketcher** workbench after selecting the newly created plane as the sketching plane and draw the sketch, as shown in Figure 5-75.

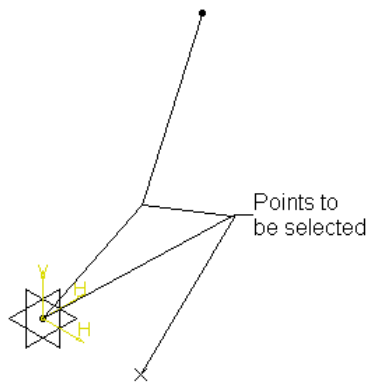


Figure 5-74 Points to be selected to create plane

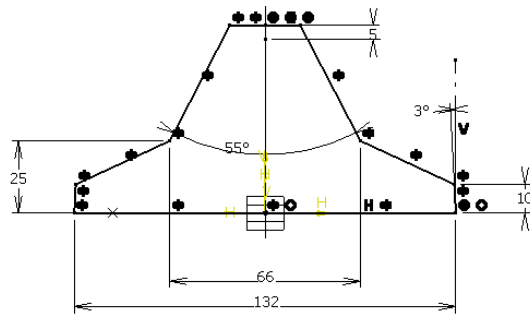


Figure 5-75 Sketch of the base feature

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 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 5-76.

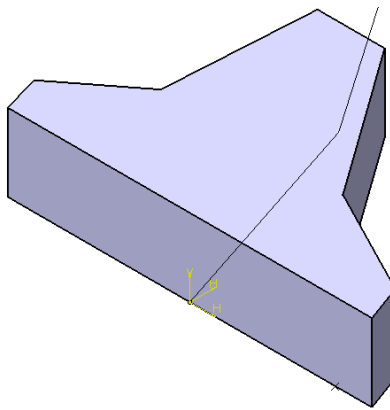


Figure 5-76 Model after creating the base feature

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, as shown in Figure 5-77.
5. Exit the **Sketcher** workbench and invoke the **Drafted Filleted Pad** tool.

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 feature creation.
8. Clear all the radio buttons available in the **Fillets** area and choose the **OK** button from the **Drafted Fillet Pad Definition** dialog box. The model, after creating the second feature, is shown in Figure 5-78.

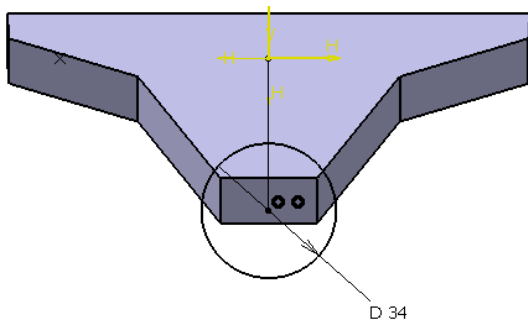


Figure 5-77 Sketch for the second feature

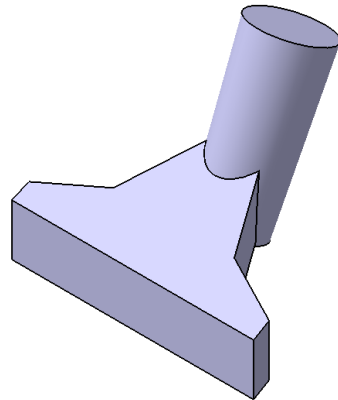


Figure 5-78 Resulting second feature

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

9. Select the ZX plane and invoke the **Sketcher** workbench. Draw the open sketch, as shown in Figure 5-79, and exit the **Sketcher** workbench.
10. Extrude the sketch using the **Pocket** tool up to last on both the sides of the sketch.
11. Using the Hide/Show tool, hide Sketch1, Sketch2, Plane1, and Plane2. The model, after creating the third feature, is shown in Figure 5-80.

Filleting the Edges of the Model

Next, you need to fillet two sets edges of the model. You need to apply the fillet feature twice because 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 5-81, and set the value of the **Radius** spinner to **12**.



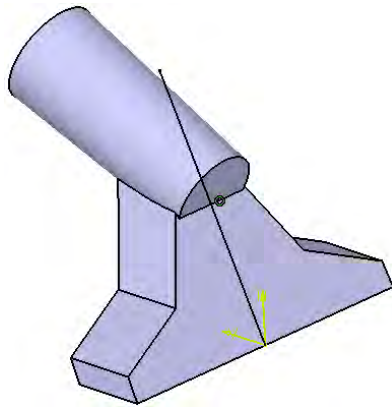
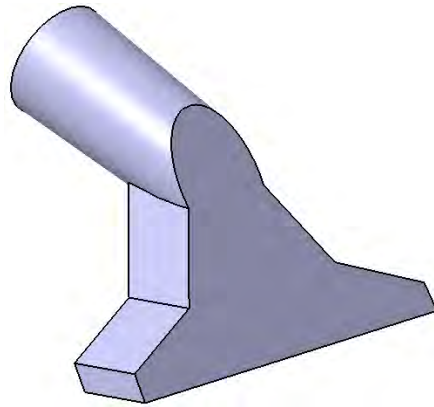
Figure 5-79 Sketch for the **Pocket** feature

Figure 5-80 Model after creating the third feature

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 5-82.

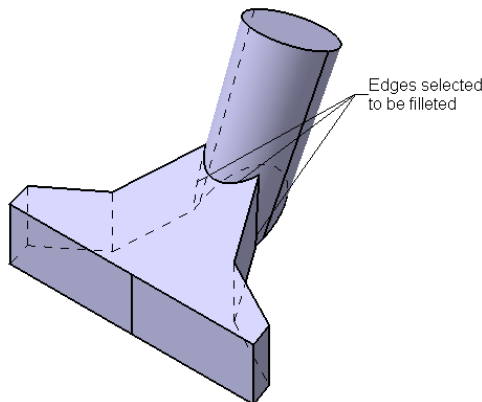


Figure 5-81 Edges to be selected

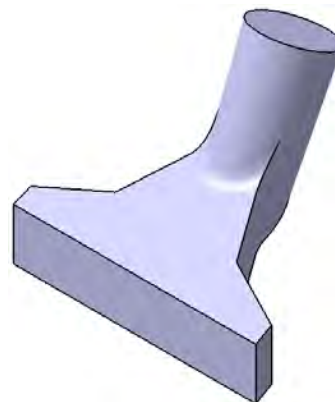


Figure 5-82 Model after creating the fillet

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

4. Select all edges of the model, except the edges that are shown in Figure 5-83.
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 again displayed. The model, after applying fillet to the second set of edges, is shown in Figure 5-84.

Creating the Shell Feature

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

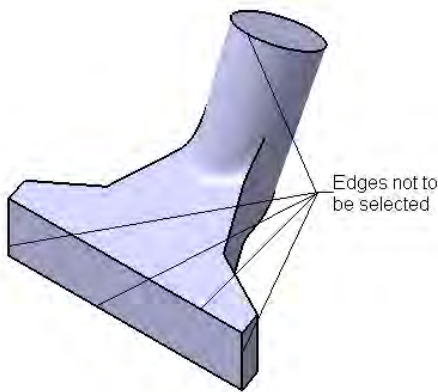


Figure 5-83 Edges not to be selected

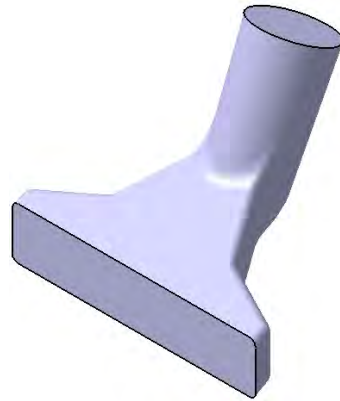



Figure 5-84 Model after creating second fillet

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 5-85.
3. Set the value of the **Default inside thickness** spinner to **2** and choose the **OK** button from the **Shell Definition** dialog box. The final model, after creating the shell feature, is shown in Figure 5-86.

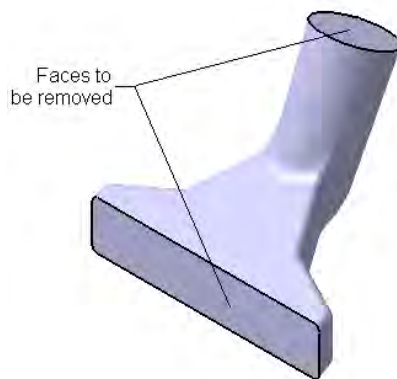


Figure 5-85 Faces to be removed



Figure 5-86 Final model after shelling

Saving and Closing the Files

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box. Create *c05* folder inside the *CATIA* folder.
2. Enter the name of the file as *c05tut1* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c05* 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 5-87. The views and dimensions of this model are shown in Figure 5-88. **(Expected time: 30 min)**



Figure 5-87 Model of the Plastic Cover for Tutorial 2

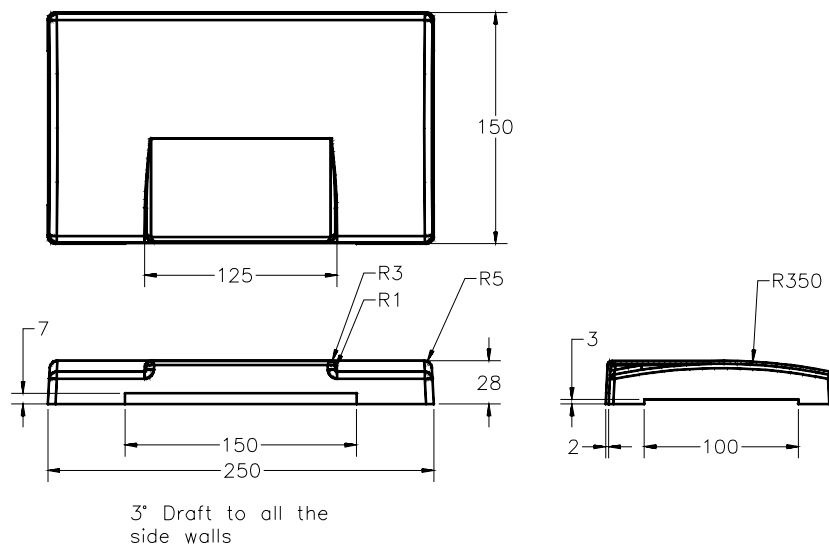


Figure 5-88 Views and dimensions of the Plastic Cover for Tutorial 2

The following steps are required to complete this tutorial:

- Create the base feature of the model by extruding the sketch drawn on ZX plane equally to both the sides of the sketch plane, refer to Figures 5-89 and 5-90.
- Create the second feature by extruding the sketch drawn on a plane created at an offset distance from the XY plane, refer to Figures 5-91 and 5-92.

- c. Add the draft feature to all faces of the model except the upper and the lower faces, refer to Figure 5-93.
- d. Fillet the edges of the model, refer to Figures 5-94 through 5-99.
- e. Shell the model using the **Shell** tool by removing the bottom face of the model, refer to Figures 5-100 and 5-101.
- f. Create two pocket features to complete the model, refer to Figure 5-102.

Creating the Base Feature of the Model

First, you need to create the base feature of the model. The base feature of the model will be created by extruding the sketch drawn on the ZX plane and the sketch will be extruded equally to 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, as shown in Figure 5-89, 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 9-90.

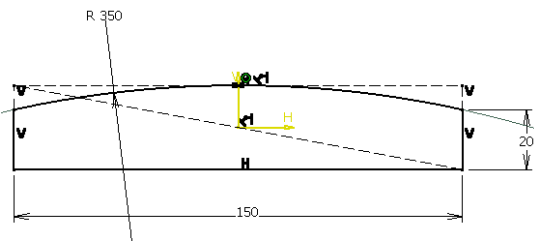


Figure 5-89 Sketch of the base feature

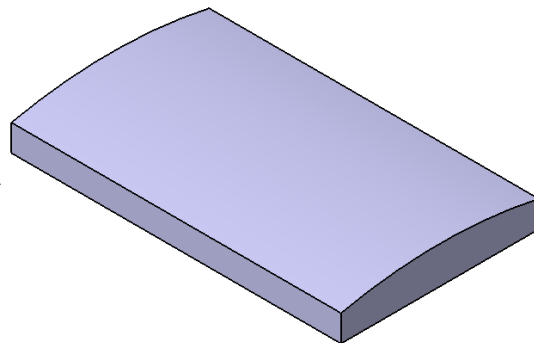


Figure 5-90 Model after creating the base feature

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 5-91, 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 5-92.

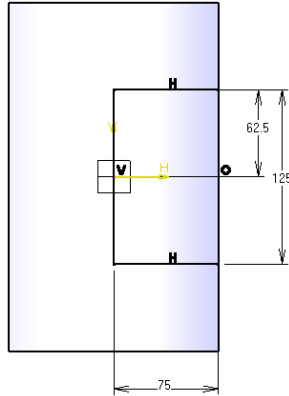


Figure 5-91 Sketch of the second feature

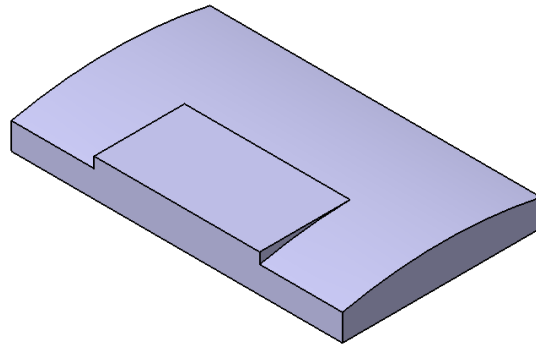



Figure 5-92 Model after creating the second feature


Adding Draft to the Faces of the Model

After creating the second feature of the model, you need to add draft to faces of the model. The draft angle is added to the model to make sure that the component is smoothly ejected from the die. Draft angle is one of the most important aspects of designing the components to be formed, molded, or casted.

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 draft. 
2. Select all the vertical faces of the base feature and the second feature from the geometry area.
3. Click once on the **Selection** selection area available 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 upwards 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 5-93.

Filletting 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. 

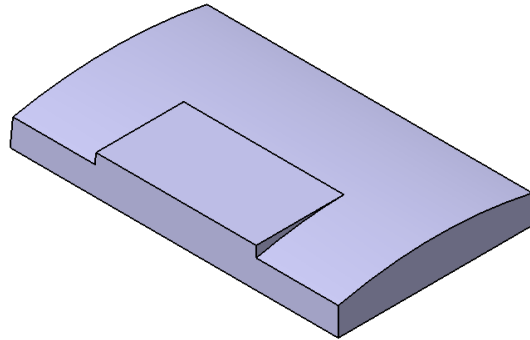


Figure 5-93 Model after drafting all the vertical faces

2. Select the edges shown in Figure 5-94 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 5-95.

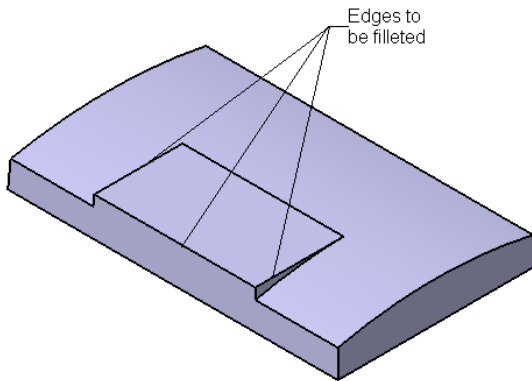


Figure 5-94 Edges to be selected

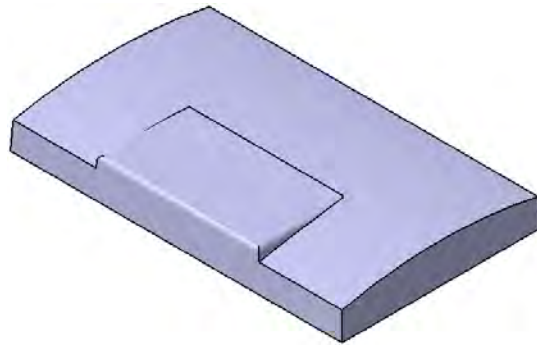


Figure 5-95 Model after filleting the first set of edges

4. Invoke the **Edge Fillet Definition** dialog box again to fillet the second set of edges.
5. Select the edge shown in Figure 5-96 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 the second set of edges, is shown in Figure 9-97.
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 5-98, and set the value of the **Radius** spinner to **5**.

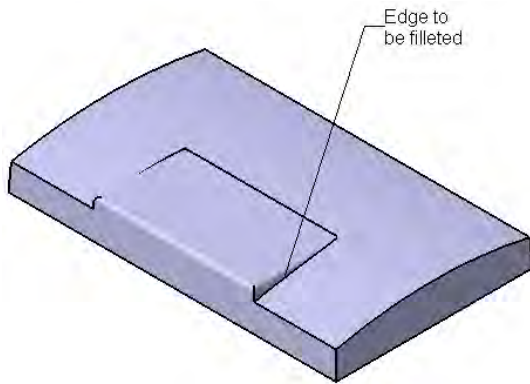


Figure 5-96 Edge to be filleted

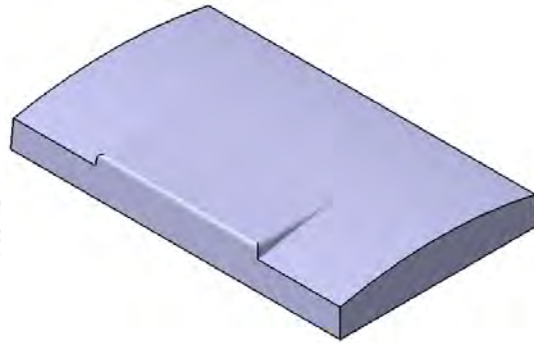


Figure 5-97 Model after filleting the second set

9. Choose the **OK** button from the **Edge Fillet Definition** dialog box. The resulting filleted model is shown in Figure 5-99.

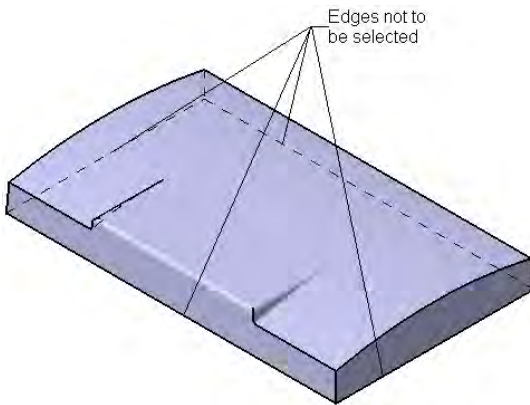


Figure 5-98 Edges not to be selected

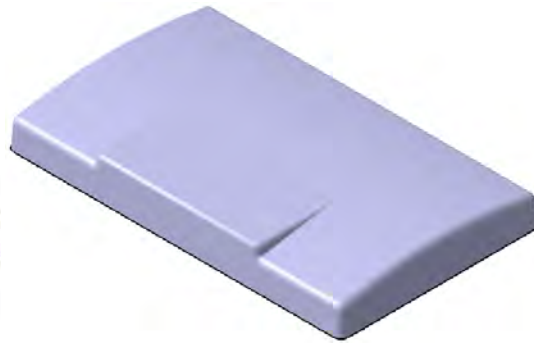



Figure 5-99 Resulting filleted model.

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 the 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 5-100, and set the value of the **Default inside thickness** 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 5-101.

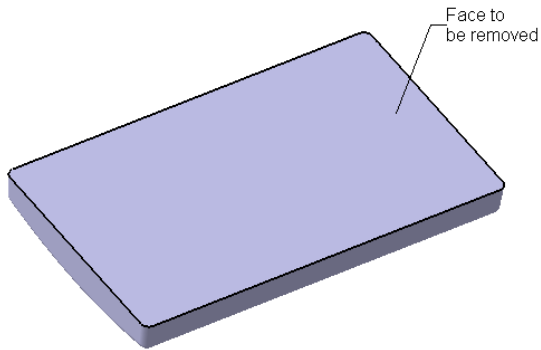


Figure 5-100 Face to be removed

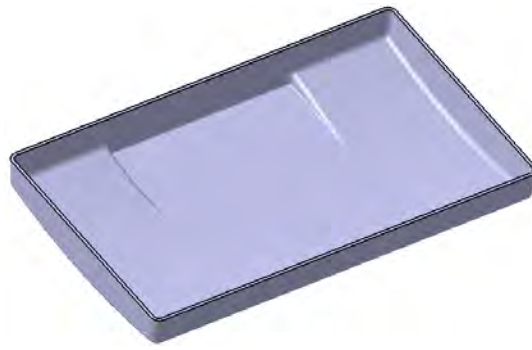


Figure 5-101 Resulting shelled model

4. Use the **Pocket** tool to create the two pocket features. The final model, after creating the other two features, is shown in Figure 5-102.



Figure 5-102 Final model after creating the remaining features

Saving and Closing the Files

1. Choose the **Save** button from the **Standard** toolbar to invoke the **Save As** dialog box.
2. Enter the name of the file as *c05tut2* in the **File name** edit box and choose the **Save** button. The file will be saved in the *\My Documents\CATIA\c05* folder.
3. Close the part file by choosing **File > Close** from the menu bar.

SELF-EVALUATION TEST

Answer the following questions and then compare your answers with 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 also 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 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 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 select the _____ option available 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 the a counterdrilled hole using the **Hole** tool. (T/F)
4. You cannot define different shell thickness values 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 the which option is selected automatically in the drop-down list available 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 5-103. The views and dimension of the model are shown in Figure 5-104. **(Expected time: 30 min)**

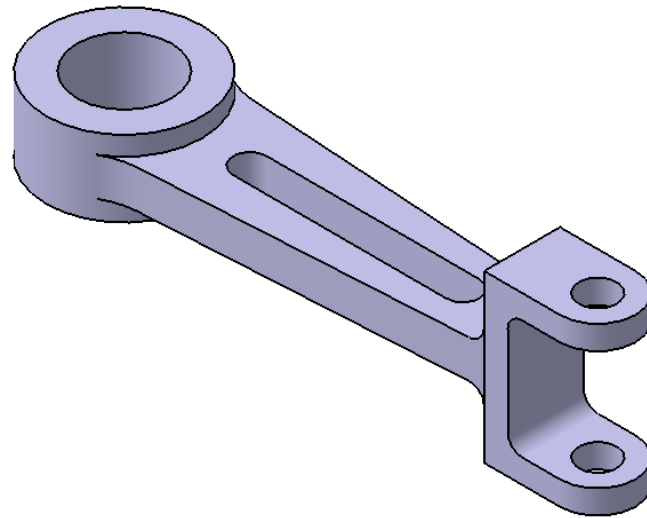


Figure 5-103 Model of the Clutch Lever for Exercise 1

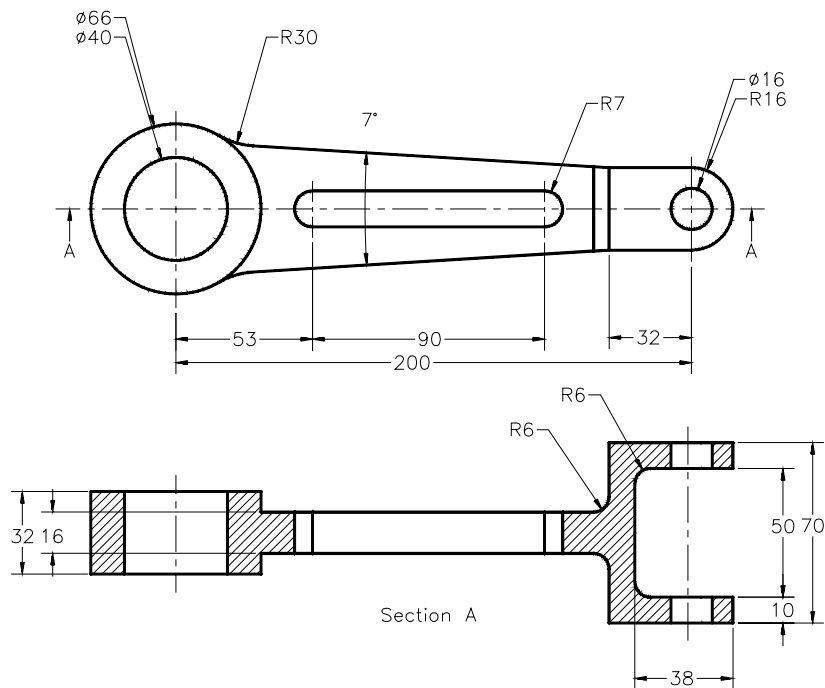


Figure 5-104 Views and dimensions of the Clutch Lever for Exercise 1

Exercise 2

Create the model of the Clamp Stop shown in Figure 5-105. The views and dimension of the model are shown in Figure 5-106. **(Expected time: 1 hr)**

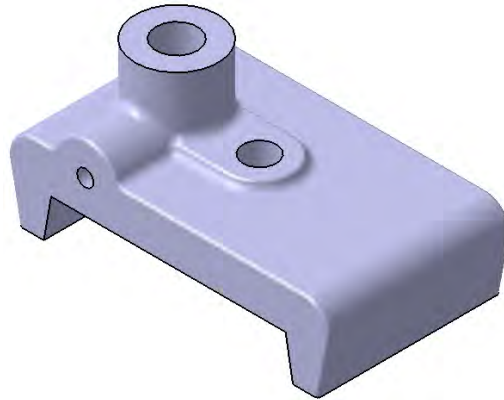


Figure 5-105 Model of the Clamp Stop for Exercise 2

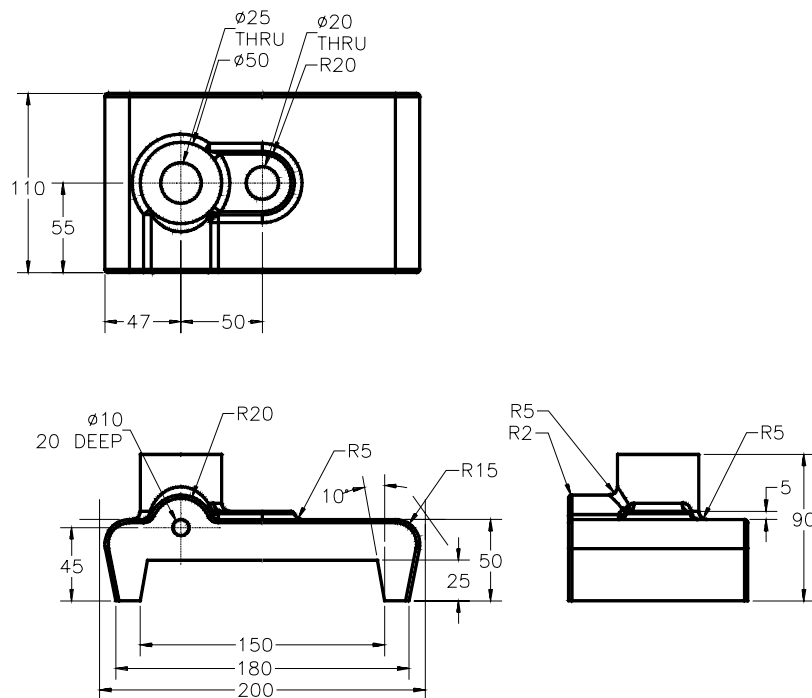


Figure 5-106 Views and dimensions of the Clamp Stop for Exercise 2

Answers to Self-Evaluation Test

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