Chapter 8

Advanced Modeling Tools-II



Learning Objectives

After completing this chapter, you will be able to:

- *Create external and internal threads.*
- *Add drafts to models.*
- *Add lip features.*
- *Create the thin wall and thin region features.*
- *Create ribs.*
- *Create web networks.*
- *Create vent features.*
- *Create mounting bosses.*

ADVANCED MODELING TOOLS

In this chapter, you will learn about some advanced modeling tools. The remaining advanced modeling tools will be discussed in the later chapters.

CREATING THREADS

Ribbon: Home > Solids > Hole drop-down > Thread



In Solid Edge, you can create internal or external threads using the **Thread** tool. Internal threads are created in a hole or in circular cut features, whereas external threads are created on the external surface of a cylindrical feature. Figure 8-1 shows the external thread. Note that you need to select only the holes or cylinders of standard size to create threads. These standard sizes are available in the **Diameter** edit box of the **Hole Options** dialog box that was discussed earlier. To create threads, invoke the **Thread** tool and choose the **Thread Options** button from the **Thread QuickBar**; the **Thread Options** dialog box will be displayed, as shown in Figure 8-2. The options in this dialog box are discussed next.

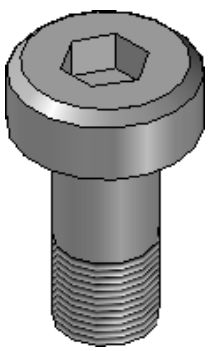


Figure 8-1 *External threads*

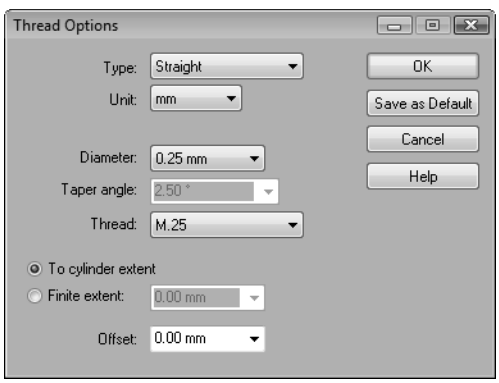


Figure 8-2 *The* ***Thread Options*** *dialog box*

Type

This drop-down list is used to specify the type of thread that you want to create. The options in this drop-down list are available depending upon the size of the selected cylinder. You can select any option from this drop-down list to create threads.

Straight

This option is selected to create threads on a straight feature.

Tapered

This option is selected to create threads on a tapered feature.

Unit

This drop-down list is used to specify the unit of threads.

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Diameter

The **Diameter** drop-down list is used to set the diameter of the hole. You can select the diameter from the predefined values available in this drop-down list or enter a value in this edit box. Note that the units of the diameters in this drop-down list depend on the type of unit selected from the **Unit** drop-down list.

Taper angle

This edit box is available when you select the **Tapered** thread type. You can enter the taper angle in this edit box or select the predefined taper angles from the **Taper angle** drop-down list.

Thread

The **Thread** drop-down list is used to set the thread type for the selected hole diameter. You can select any predefined thread type from this drop-down list.

To cylinder extent

This radio button is available only when you are creating straight thread and it is used to create threads through the entire length of the cylinder.

Finite extent

This option enables you to create threads upto a specified depth. If you select the **Finite extent** radio button, the **Thread Depth** edit box will be invoked below the drop-down list. You can use this edit box to specify the depth value.

Offset

This edit box is used to specify the distance between the start of the thread and end face of the cylinder. This edit box becomes active only when you select the **Straight** option in the **Type** drop-down list.

After you specify the parameters in the **Thread Options** dialog box, select the cylindrical face on which the thread is to be created; a warning box will be displayed, informing that the diameter of the cylinder will be changed to match the diameter of the thread. Click **OK** button to create a thread.

ADDING DRAFTS TO THE MODEL

Ribbon: Home > Solids > Draft



This tool is used for tapering the faces of a model. Tapering the faces enables you to easily remove the component from the mold during the casting. You can add a draft to a model using the **Draft** tool. To do so, invoke the **Draft** tool from the **Solids** group; the **Draft QuickBar** will be displayed. Choose the **Draft Options** button from it; the **Draft Options** dialog box will be displayed, as shown in Figure 8-3. This dialog box consists of four radio buttons (options) to create a draft. In this chapter, you will learn to create a draft using the first two options of the dialog box. The remaining options will be discussed in the later chapters.

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Creating a Draft Using the From plane Method

By default, the **From Plane** radio button is selected in the **Draft Options** dialog box. Choose **OK** to exit the dialog box; you are prompted to click on a planar face or on a reference plane. The draft plane is a plane whose normal is used to define the draft angle. After defining the Draft Plane, you are prompted to select one or more faces on which the draft will be added. You can use the options in the **Select Type** list to select the faces to draft. After selecting the draft faces, a draft direction handle along with the angle edit box will be displayed. Next, enter the draft angle in the **Draft Angle** edit box; the preview of the draft will be displayed. Note that you cannot enter a negative draft angle value because you need to define the direction of the draft using the draft direction handle. Figures 8-4 and 8-5 show the draft directions and the resulting drafts respectively.



Figure 8-3 *The* ***Draft Options*** *dialog box*

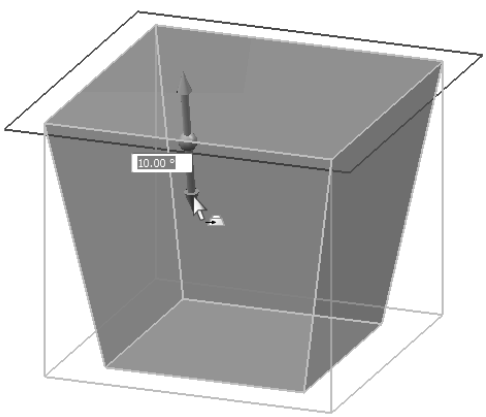


Figure 8-4 *Defining the draft direction using the draft direction handle*

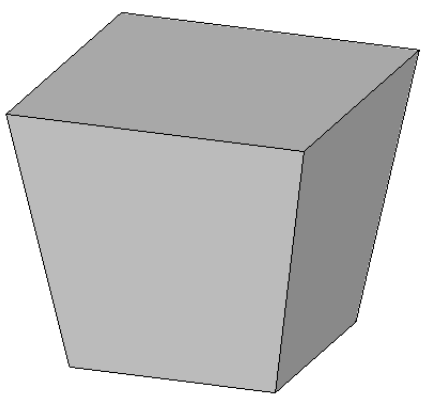


Figure 8-5 *Resulting draft*

Creating a Draft Using the From edge Method

The method of creating a draft from an edge is similar to that of creating a draft from a plane. The only difference is that in the edge draft, you are allowed to select an edge from where the draft angle will be measured. This is done in the **Select Parting Geometry Step** that is invoked after the **Draft Plane** step. Note that out of all the selected faces, the draft will be added only to the face from which the selected edge will pass. Figures 8-6 and 8-7 show the draft directions and the resulting drafts respectively.

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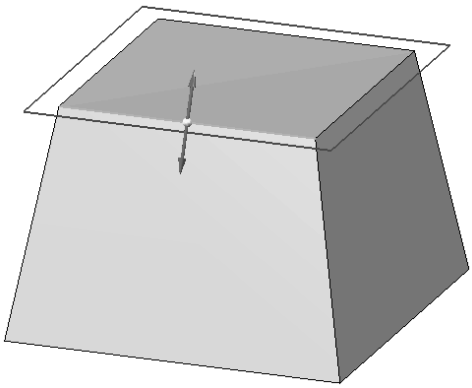


Figure 8-6 *Defining the direction of draft creation*

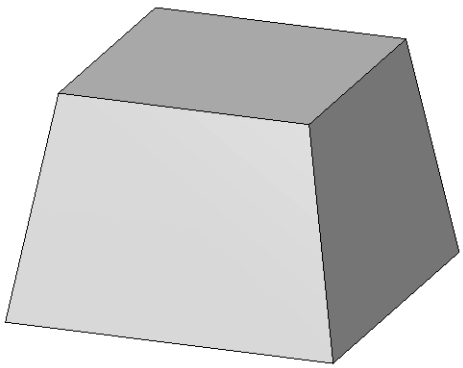


Figure 8-7 *Resultant draft*

The remaining methods used for creating drafts are discussed in the Chapter, 'Surface Modeling'.

ADDING RIBS TO THE MODEL

Ribbon: Solids > Thin wall drop-down > Rib

Ribs are defined as the thin wall-like structures used to bind joints together so that they do not fail under an increased load. In Solid Edge, ribs are created using an open profile, refer to Figures 8-8 and 8-9.

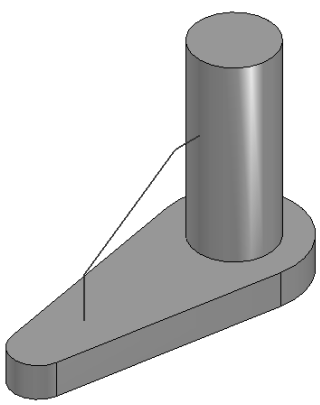


Figure 8-8 *Open profile to create a rib*

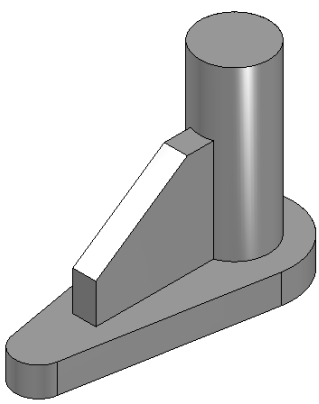


Figure 8-9 *Resulting rib feature*

The process of creating ribs is completed in four steps, which are discussed next.

Sketch Step

This step enables you to select a sketching plane for drawing the profile of the rib feature. You can also select an existing profile using the **Select from Sketch** option from the **Create-From Options** drop-down list. It is recommended that the profile of the rib feature

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should be extruded symmetrically. Therefore, you need to select the sketching plane for drawing the profile accordingly.

Draw Profile Step

This step will be automatically invoked when you select the sketching plane for drawing the profile of the rib feature.

Direction Step

This step is invoked automatically when you select the profile and accept it or draw the profile and exit the sketching environment. This step enables you to define the direction of the rib creation, and therefore, you are prompted to click to accept the displayed side or select the other side in the view. The feature can be created in a direction normal to the profile or parallel to it. If you move the cursor in the drawing window, a dynamic preview of the rib feature will be displayed in various directions. If the feature cannot be created, an error symbol will be displayed in the preview of the rib, as shown in Figure 8-10. However, the rib feature will be successfully created if you define that side for the feature creation in which the profile meets the faces of the existing features, as shown in Figure 8-11. Next, click to specify the side of the feature creation.

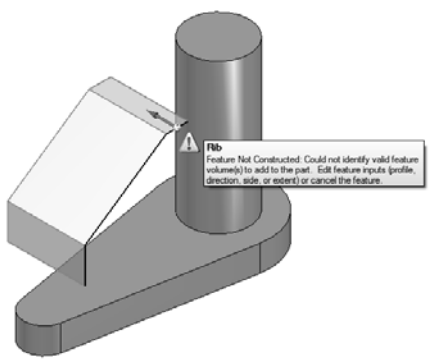


Figure 8-10 *Preview of the direction in which the rib will not be created and the error symbol*

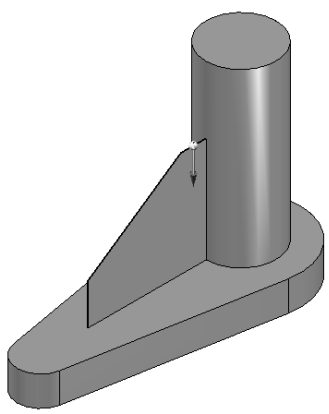


Figure 8-11 *Preview of the direction in which the rib feature will be created*

Side Step

The **Side Step** button will be active in the command bar. Choose this button to define the direction of the rib creation. In the **Side Step**, you are allowed to specify the side of the sketching plane on which the rib will be created. You can move the cursor on either side of the profile to define it. As mentioned earlier, it is recommended to create the rib symmetrically on both sides of the sketching plane. To create a symmetric rib, move the cursor close to the profile; the preview of the symmetric rib will be displayed, as shown in Figure 8-12. Next, click to create the symmetric rib.

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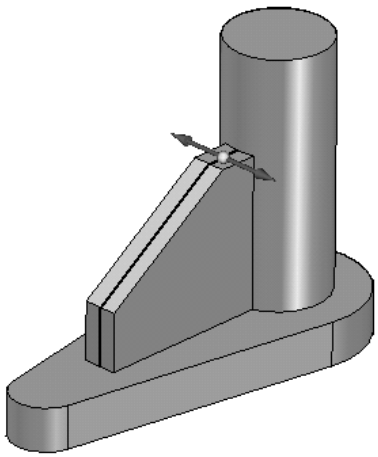


Figure 8-12 *Preview of the symmetric rib*

Note that in the **Direction Step** and the **Side Step**, some additional options are available in the command bar. These options are discussed next.

Extend Profile

This button is chosen by default and is used to extend the rib feature to the adjacent features, even if the profile does not extend to them. Figure 8-13 shows an open profile for creating the rib. As is evident in this figure, the open profile does not extend to the adjacent features. Figure 8-14 shows the rib feature created using the same profile with the **Extend Profile** button chosen. As is evident from this figure, the rib feature has been extended to the adjacent features.

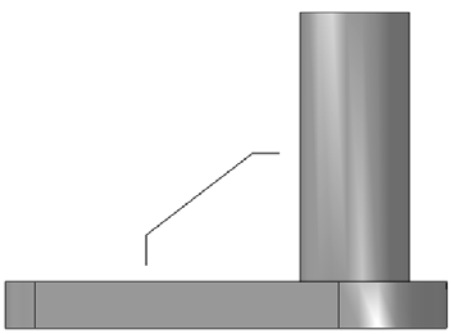


Figure 8-13 *Open profile for creating the rib*

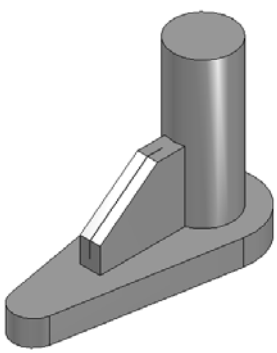


Figure 8-14 *Rib feature created with the* ***Extend Profile*** *button chosen*

No Extend

This button should be chosen when you do not want the rib to extend to the adjacent faces. Figure 8-15 shows the profile and the resulting rib created by choosing the **Extend Profile** button and Figure 8-16 shows the rib feature created by choosing the **No Extend** button.

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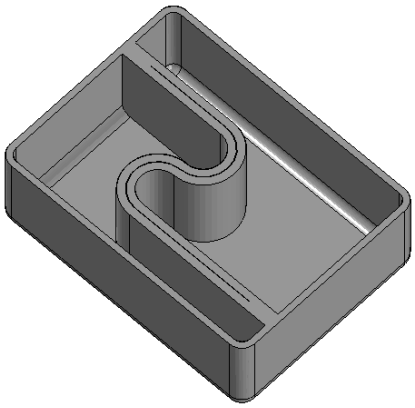


Figure 8-15 *Rib feature created using the* ***Extend Profile*** *button chosen*

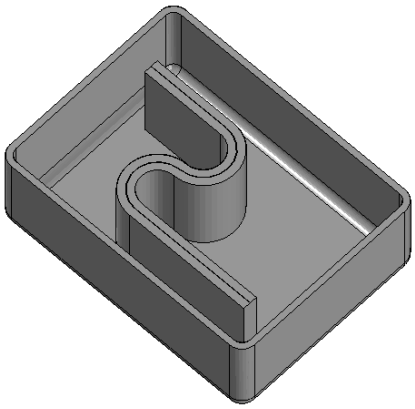


Figure 8-16 *Rib feature created using the* ***No Extend*** *button chosen*

Extend to Next

This button is chosen to extend the rib to the next features in the direction specified in the **Direction Step**. Figure 8-17 shows the rib feature created using this option.

Finite Depth

This button is chosen when you want to extend the rib to a finite depth in the direction that you have specified in the **Direction Step**. You can specify the depth of the rib in the **Depth** edit box that is displayed below the **Thickness** edit box when you choose the **Finite Depth** button. Figure 8-18 shows the rib feature created using this option.

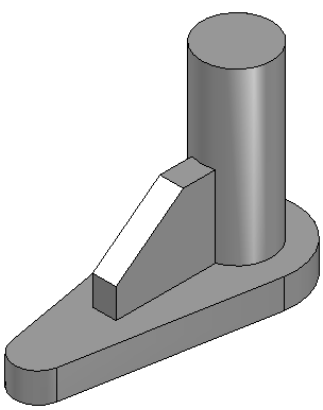


Figure 8-17 *Rib feature created using the* ***Extend to Next*** *button*

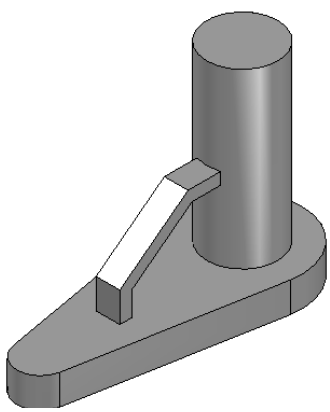


Figure 8-18 *Rib feature created up to a finite depth*

Thickness

This edit box is used to specify the thickness of the rib feature. The default thickness is 0.25 and you can enter any desired thickness value for the rib in this edit box. You can also select the predefined thickness values using the **Thickness** edit box.

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Depth

This edit box will be available when you choose the **Finite Depth** button. It is used to specify the depth of the rib when you want to extend it to a finite depth.

ADDING THIN WALL FEATURES (ORDERED PART)

Ribbon: Home > Solids > Thin Wall



By adding a thin wall feature, you can scoop out the material from a model and make it hollow from inside. The resulting model becomes the structure of walls with a cavity inside. You can also remove some of the faces of the model or apply different wall thicknesses to some of them. Figure 8-19 shows the model with the thin wall feature added and the front face removed.

This tool works in the following three steps:

Common Thickness

This step enables you to specify the common thickness for the thin wall feature. You can also specify the side of the solid toward which the thin wall will be created. The options in the command bar under this step are discussed next.

Offset Outside

The **Offset Outside** button is chosen to define the wall thickness outside the model with respect to its outer faces. In this case, the outer faces of the model will be considered as the inner walls of the resulting thin wall feature.

Offset Inside

The **Offset Inside** button is chosen by default and is used to define the wall thickness inside the model with respect to its outer faces. In this case, the outer faces of the model will be considered as the outer walls of the resulting thin wall feature.

Symmetrical

The **Symmetrical** button is chosen to calculate the wall thickness equally in both the directions of the outer faces of the model.

Common thickness

This edit box is used to specify the common thickness for the thin wall feature.

Open Faces

This step will be automatically invoked when you specify the common thickness and press ENTER. In this step, you can specify the face that you want to remove from the thin wall feature. You can use the **Select Type** list to define the selection method. After specifying the faces to be removed, choose the **Accept** button and then choose the **Preview** button to preview the thin wall feature. Figure 8-20 shows a thin wall model with the front and left side faces removed.

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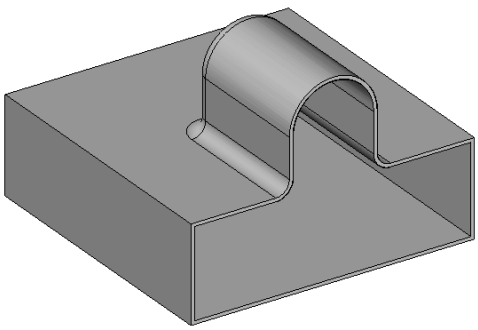


Figure 8-19 *Model with the thin wall feature added*

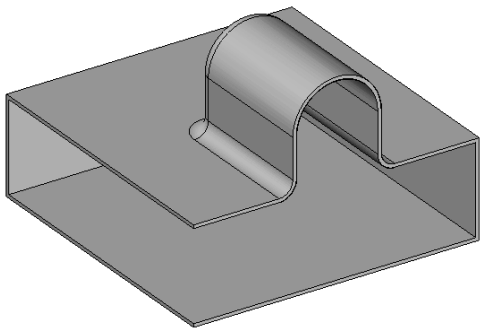


Figure 8-20 *Thin wall model with the front and left faces removed*

Unique Thickness

This step is used specify different wall thicknesses. You need to choose this step to invoke it. After selecting the face or faces, specify the unique thickness value in the **Unique Thickness** edit box in the command bar and press ENTER. Again, select another face or faces to which you want to add different wall thicknesses and specify the wall thickness in the edit box. Continue this process till you have selected all faces to which you want to add different wall thicknesses. Figure 8-21 shows a thin wall model with an open face and a unique thickness added to the left and bottom faces.

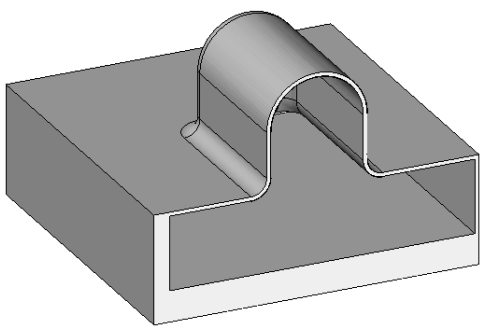


Figure 8-21 *Model with multiple wall thicknesses*

To create thin walls in **Synchronous Part** environment, invoke the **Thin Wall** tool from the **Solids** group. Next, click on face that you want to leave open and then enter the common thickness value in the edit box; the preview of the thin wall will be displayed dynamically. Use the handle to specify whether the thickness should be added with reference to inner faces or outer faces. Right click to accept the thin feature.

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ADDING THIN WALL TO A PARTICULAR REGION

Ribbon: Home > Solids > Thin Wall > Thin Region



Sometimes, you may need to add the thin wall feature to a particular region, instead of the complete model. For example, refer to the model shown in Figure 8-22. You can add thin wall to a particular region using the **Thin Region** tool. This tool is available only in **Ordered Part** environment.

The **Thin Region** tool works in the following four steps:

Faces To Thin Step

This is the first step and it is activated automatically when you invoke the **Thin Region** tool. In this step, you need to select a particular feature or the faces of a region to add the thin wall feature. Next, specify the common thickness in the **Common Thickness** edit box. Note that while selecting the faces, you need to make sure that they result in a closed volume. For example, to create a thin wall region shown in Figure 8-22, you need to select the top curved face, side tangent faces, the back face, and the front face of the region, as shown in Figure 8-23.

Open Faces Step

This step will be automatically invoked when you exit the previous step. In this step, you can select the faces that you want to remove from the thin wall region. For example, to create the model shown in Figure 8-22, you need to remove the front face.

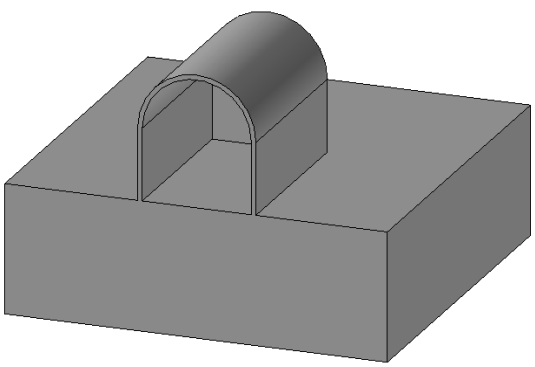


Figure 8-22 *Model with a thin wall feature added to a particular region*

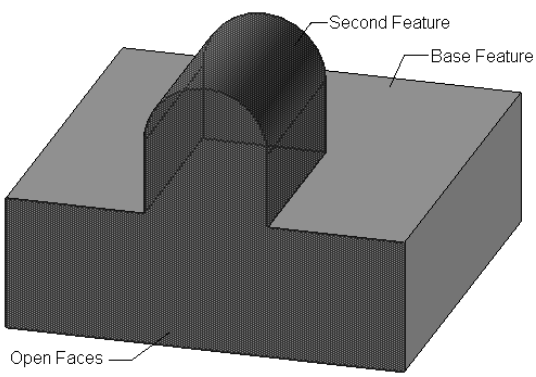


Figure 8-23 *Selecting faces to create a thin wall region*



Note

While creating the thin region feature, if the face that you selected to remove is still displayed, then you need to edit the feature and again select the face to be removed in the ***Open Faces*** *step.*

Capping Faces Step

Capping face can be considered as the face that defines the termination of the thin region. You can select any face of a model or an existing surface to define the capping faces. You can

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also define an offset value from the capping face using the **Offset** edit box that is displayed in the command bar. Figure 8-24 shows a surface used as the capping face and Figure 8-25 shows the same surface used as the capping face but with an offset of 2 units. You will learn more about surfaces in the later chapters.

Figure 8-26 shows the preview of the model. In this preview, the top planar face has been used as the capping face with an offset of 5 units.

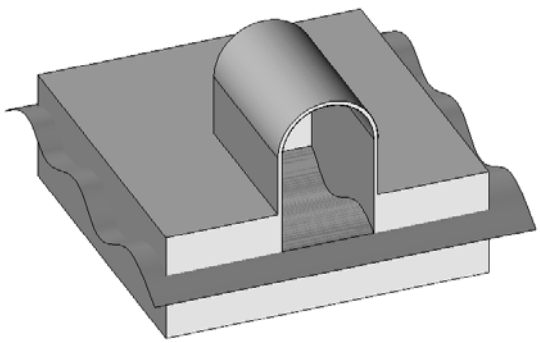


Figure 8-24 *Surface used as the capping face*

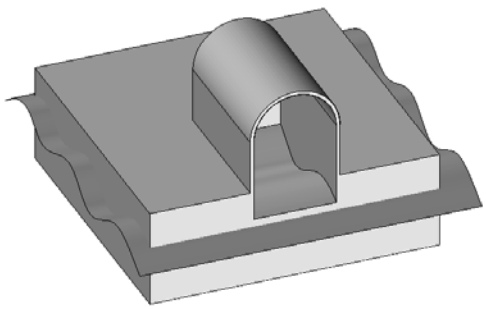


Figure 8-25 *Surface used as the capping face with an offset of 2*

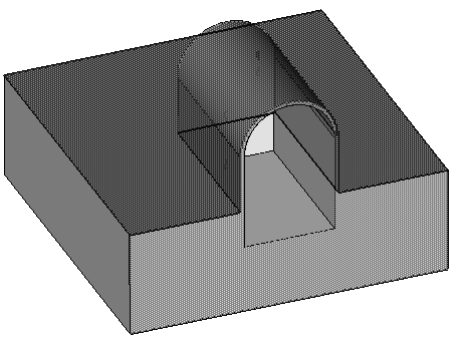


Figure 8-26 *Top face used as the capping face with an offset of 5*

Unique Thickness Step

This step is used to define different thicknesses to the selected faces and it works similar to the **Unique Thickness** step in the **Thin Wall** tool.

ADDING A LIP TO THE MODEL

Ribbon: Home > Solids > Thin Wall drop-down > Lip



The **Lip** tool is used to add a lip to the model by adding material along the selected edges, or by adding a groove to the model by removing the material along the selected edges. The amount of material to be added or removed is defined by a rectangle whose width and height can be specified on invoking this tool.

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The **Lip** tool works in the following two steps:

Select Edge Step

This step allows you to select the edge along which you want to add a lip or groove. You can use the options in the **Select** drop-down list to select an individual edge or a chain of edges.

Direction Step

This step enables you to specify the direction and size of the lip feature. When this step is invoked, the **Width** and **Height** edit boxes will be displayed. Specify the width and height of the rectangle, which defines the profile of the lip, in these edit boxes.

After specifying the width and height of the lip, move the cursor in the drawing window. You will notice that an orange colored rectangle is displayed at the start of the edge. Move the cursor around the start of the edge and specify the location of the lip. Note that if the rectangle is inside the feature, the resulting feature will be a groove and if the rectangle is outside the feature, the resulting feature will be a lip. Figures 8-27 through 8-30 show different positions of the rectangle and the resulting lip features.

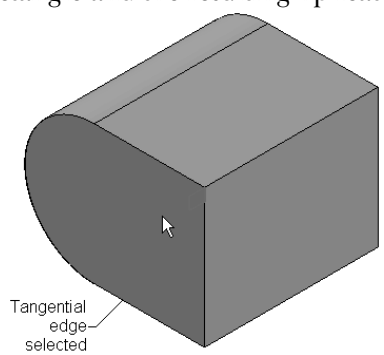


Figure 8-27 *Location of the rectangle*

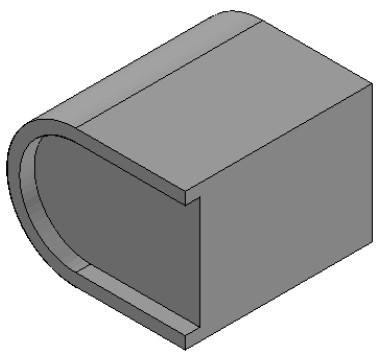


Figure 8-28 *Resulting lip feature*

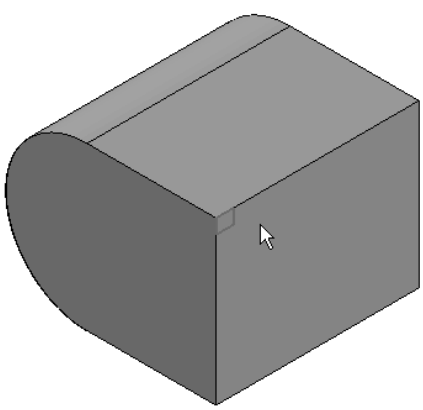


Figure 8-29 *Location of the rectangle*

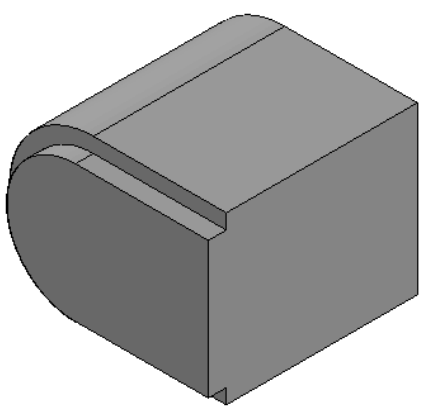


Figure 8-30 *Resulting lip feature*

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CREATING WEB NETWORKS

Ribbon: Home > Solids > Thin Wall drop-down > Web Network



The **Web Network** tool enables you to create a web network using open entities, as shown in Figures 8-31 and 8-32.

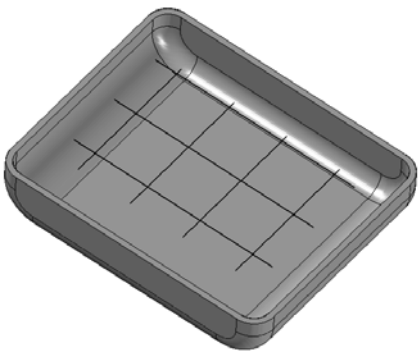


Figure 8-31 *Thin wall model and a network of lines*



Figure 8-32 *Resulting web network*

This tool is similar to the **Rib** tool and performs the work in the following four steps:

Sketch Step

This step enables you to select a sketching plane for drawing the profile of the web network. You can also select an existing profile using the **Select from Sketch** option from the **Create-From Options** drop-down list.

Draw Profile Step

This step will be automatically invoked when you select the sketching plane for drawing the profile of the web network.

Direction Step

This step will be automatically invoked when you select the profile and accept it or draw the profile and exit the sketching environment. This step enables you to define the direction in which the web network will be created. It is recommended to define the direction of the web network toward the bottom of the part.

You can specify the thickness of the webs in the web network in the **Thickness** edit box. The functions of the other buttons in the command bar of this step are the same as those in the **Rib** tool. Figure 8-33 shows a web network in which webs are not extended. Figure 8-34 shows a web network in which webs are extended, but defined up to a finite depth.

Treatment Step

This step is used to add a draft to webs in a web network. It works similar to the **Treatment Step** in the **Protrusion** tool.

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Tip. *To select multiple entities, refer to Figure 8-31, select the* ***Single*** *option from the* ***Select*** *drop-down list and then drag a box around the entities to select them.*

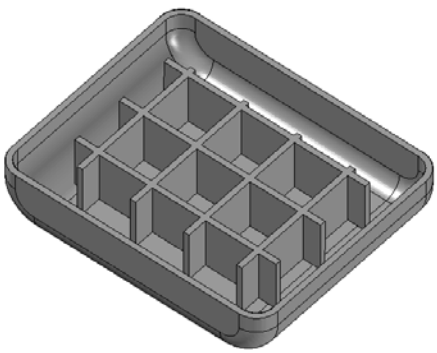


Figure 8-33 *Web network with webs not extended*



Figure 8-34 *Webs defined up to a finite depth*

CREATING VENTS

Ribbon: Home > Solids > Thin Wall drop-down > Vent

The **Vent** tool enables you to create a vent in an existing model by defining the boundary of the vent, ribs, and spars in the vent. This tool is available only after you have drawn the sketch for the vent. Figure 8-35 shows a model, and a profile that defines the boundary. Note that in the vent, all vertical lines are selected as ribs and all horizontal lines are selected as spars. Figure 8-36 shows the resulting model with the vent.

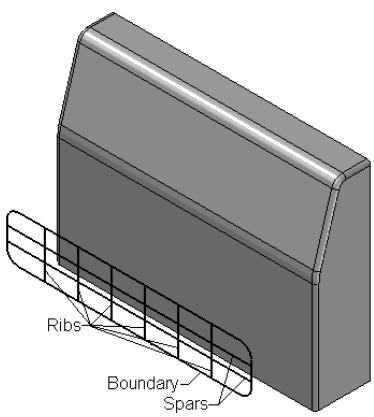


Figure 8-35 *Parameters of the vent*

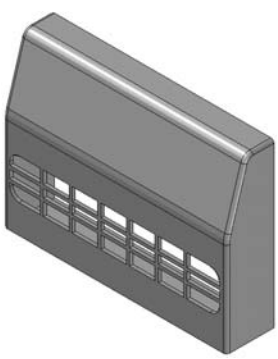


Figure 8-36 *Resulting model with the vent*

To create a vent, invoke the **Vent** tool; the **Vent Options** dialog box will be displayed, as shown in Figure 8-37.

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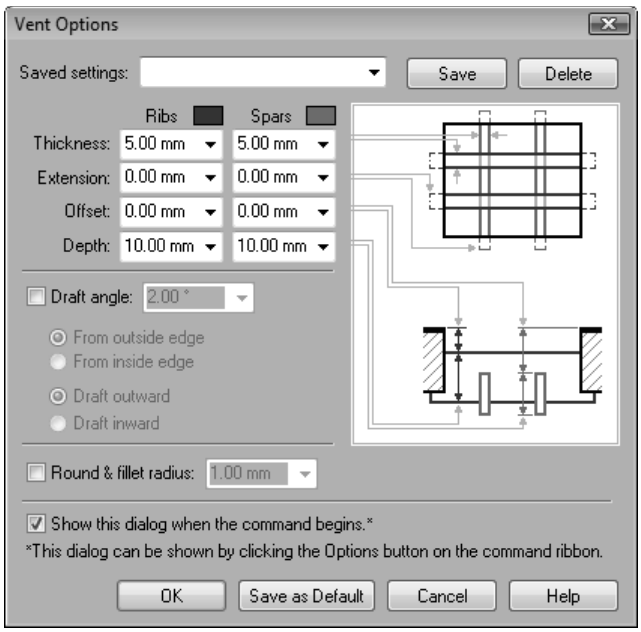


Figure 8-37 *The* ***Vent Options*** *dialog box*

Vent Options Dialog Box

The options in this dialog box are discussed next.

Saved settings

This drop-down list displays the list of settings that you have saved. By default, this drop-down list is blank. To save the settings, set the required parameters in this dialog box, enter a name for the parameters set in the **Saved settings** edit box, and then choose the **Save** button; the settings will be saved. Now, if you want to view the parameters set under any set, select the required name from the **Saved settings** drop-down list. You can delete the unwanted settings by selecting them from this drop-down list and choosing the **Delete** button.

Thickness Ribs/Spars

These edit boxes are used to specify the thickness of ribs and spars. Figure 8-38 shows a vent with the thickness of ribs and spars as 2. Figure 8-39 shows a vent with the thickness of ribs and spars as 5. You can have the same or different thickness values for the ribs and spars.

Extension Ribs/Spars

These edit boxes are used to specify the distance by which the ribs and spars will extend beyond the boundary of the vent. You can have the same or different extension values for the ribs and spars. Figure 8-40 shows the ribs and spars without extension and Figure 8-41 shows the ribs and spars extended beyond the boundary.

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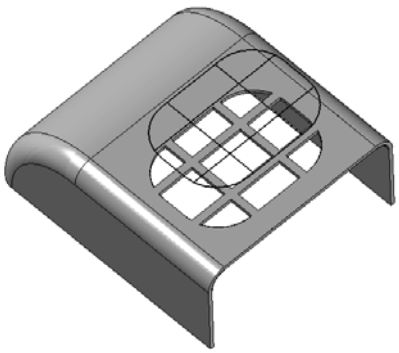


Figure 8-38 *Vent with ribs and spars thickness as 2*

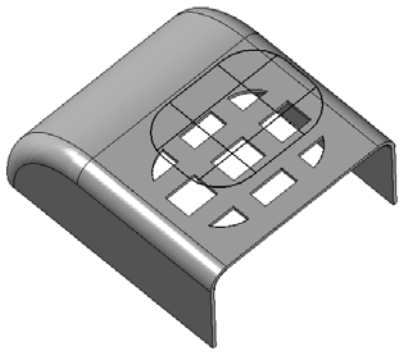


Figure 8-39 *Vent with ribs and spars thickness as 5*

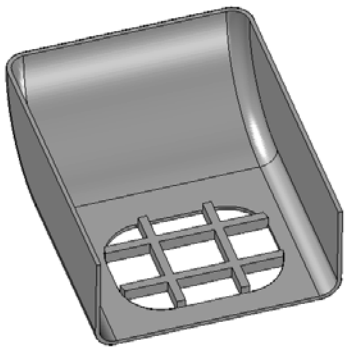


Figure 8-40 *Ribs and spars not extended*

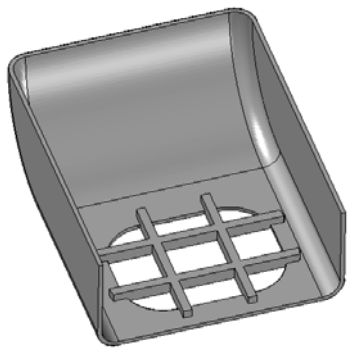


Figure 8-41 *Ribs and spars extended beyond the boundary*

Offset Ribs/Spars

These edit boxes are used to specify the distance by which the ribs and spars will be offset from the face on which the profile is projected. You can specify the same or different offset values for the ribs and spars. Figure 8-42 shows the ribs and spars starting at some offset from the face on which the profile is projected.

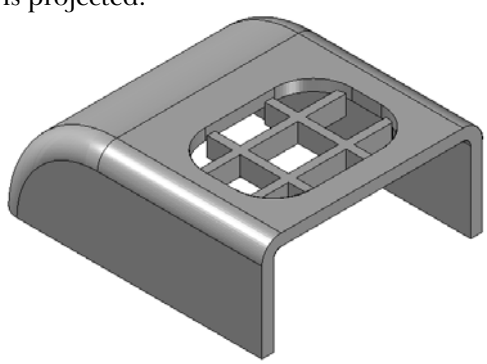


Figure 8-42 *Ribs and spars starting at an offset from the top face*

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Note

If the offset value of the ribs and spars is more than the thickness of the face on which the profile of the vent is created, then the feature may not be created.

Depth Ribs/Spars

These edit boxes are used to specify the depth of ribs and spars. You can have the same or different depth values for each one of them. Figures 8-43 and 8-44 show vent features with different depth values for ribs and spars.



Figure 8-43 *Ribs and spars depth = 2*

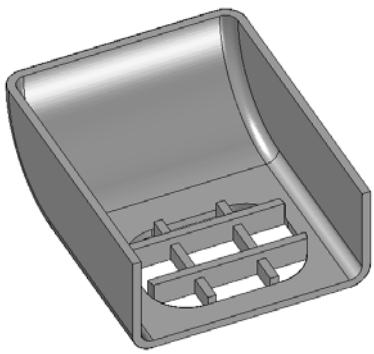


Figure 8-44 *Ribs depth = 4, spars depth = 6*

Draft angle

The **Draft angle** check box is selected to add a draft to the ribs and spars in the vent. The draft angle can be specified in the edit box available on the right of this check box. You can also specify whether the draft should be specified from the outside or the inside edge, and whether the draft should be outward or inward. You can do so by selecting the required radio button below the **Draft angle** edit box.

Round & fillet radius

The **Round & fillet radius** check box is selected to add rounds and fillets to the vent. The radius of the round and fillet can be specified in the edit box available on the right of this check box. Figure 8-45 shows a vent with fillets and rounds.

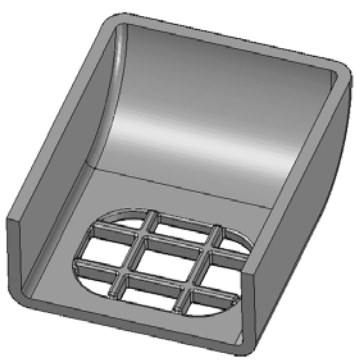


Figure 8-45 *Vent with fillets and rounds*

After specifying the required parameters in the **Vent Options** dialog box, choose **OK**. The **Vent** tool works in the following four steps:

Select Boundary Step

This step will be activated when you exit the **Vent Options** dialog box. In this step, you can select a chain of entities that will act as the boundary of the vent. You can also select the options in the **Select Type** list to select a chain of entities or an individual entity.

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Select Ribs Step

This step will be automatically invoked when you accept the boundary in the **Select Boundary Step**. In this step, you can select closed or open entities to define the ribs in the vent. After selecting the entities, right-click to accept the selection.

Select Spars Step

This step will be automatically invoked when you accept the entities to define the ribs in the **Select Ribs Step**. In this step, you can select the entities that you want to use as spars in the vent. You can select closed or open entities to define the spars. After selecting the entities, right-click to accept the selection.

Extent Step

This step is used to specify the side and the extent of the vent. You can use the buttons in the command bar of this step to define the extent.

CREATING MOUNTING BOSSES (ORDERED PART)

Ribbon: Home > Solids > Thin Wall drop-down > Mounting Boss



The **Mounting Boss** tool enables you to create mounting boss features, which are used in plastic components to accommodate fasteners. Figure 8-46 shows a model with four mounting boss features.

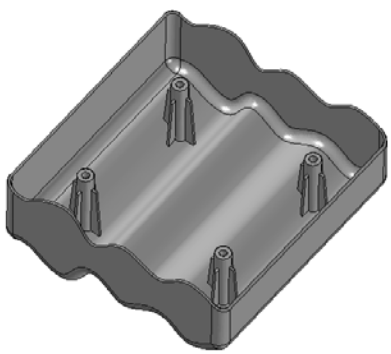


Figure 8-46 *Model with four mounting bosses*

To create mounting boss features, invoke the **Mounting Boss** tool; the **Mounting Boss** command bar will be displayed. It is recommended that before proceeding further, you set the parameters of the mounting boss features in the **Mounting Boss Options** dialog box, as shown in Figure 8-47. This dialog box is invoked on choosing the **Mounting Boss Options** button from the **Mounting Boss** command bar.

Mounting Boss Options Dialog Box

The options in this dialog box are discussed next.

Saved settings

This drop-down list displays the list of settings that you have saved. By default, this drop-down list is blank. To save the settings, set the required parameters in this dialog box, enter a name for the parameters set in the **Saved settings** edit box, and then choose the **Save** button; the settings will be saved. Now, if you want to view the parameters set, select the required name from the **Saved settings** drop-down list. You can delete the unwanted settings by selecting them from this drop-down list and choosing the **Delete** button.

Settings

The options available in this area are used to set the parameters of the mounting boss feature. All these options have a grey arrow on the right that leads to a parameter in the preview

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window. The preview window explains the use of the options available in the **Settings** area. These options are discussed next.

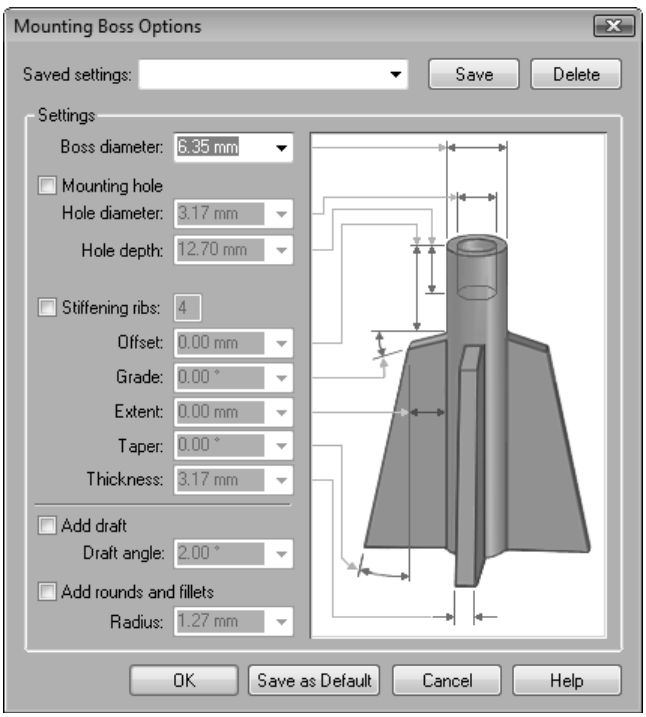


Figure 8-47 *The* ***Mounting Boss Options*** *dialog box*

Boss diameter

This edit box is used to specify the diameter of the mounting boss.

Mounting hole

This check box is selected to create a hole on the top face of the mounting boss. On selecting this check box, the **Hole diameter** and the **Hole depth** edit boxes will become available and you can specify the diameter and the depth of the hole in these edit boxes, respectively.

Stiffening ribs

This check box is used to create a mounting box with ribs. If this check box is not selected, only a cylindrical feature will be created as the mounting boss. You can specify the number of stiffening ribs in the edit box available on the right of this check box.

Offset

This edit box will be available only when you select the **Stiffening ribs** check box and is used to specify the distance between the start of the rib and the top face of the mounting boss.

Grade

This edit box is used to specify the angle of the top face of the ribs with respect to the top face of the mounting boss.

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Extent
This edit box is used to specify the extrusion depth of the top face of the ribs from the cylindrical surface of the mounting boss.

Taper
This edit box is used to specify the taper angle value of the rib. Note that you can enter only a positive taper angle value for the rib.

Thickness
This edit box is used to specify the thickness of the rib.

Add draft
This check box is selected to add a draft to the mounting boss. On selecting this check box, the **Draft angle** edit box will be activated and then you can enter the draft angle in this edit box. Figure 8-48 shows a model with mounting bosses but without a draft and Figure 8-49 shows the mounting bosses with draft.

Add rounds and fillets
This check box is selected to add rounds and fillets to the mounting boss. On selecting this check box, the **Radius** edit box will be activated and now you can enter the radius value in this edit box.

After setting the options in the **Mounting Boss Options** dialog box, you need to use the following three steps to create the mounting bosses:

Plane Step

This step enables you to select a plane for placing the profiles of the mounting bosses. Note that the profiles of the mounting bosses are placed at a planar face or a reference plane parallel to the face on which you want to project them. The distance between the parallel plane and the face on which the mounting bosses are projected defines their depth. It is similar to extruding the profile from the parallel plane up to the face.

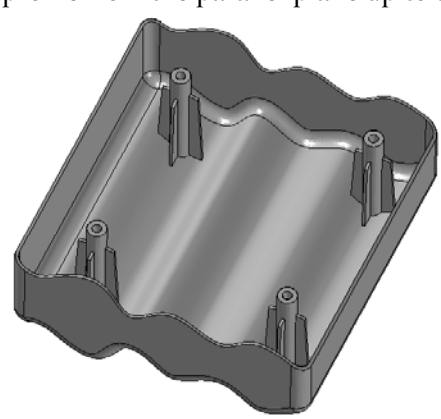


Figure 8-48 *Mounting bosses without draft*

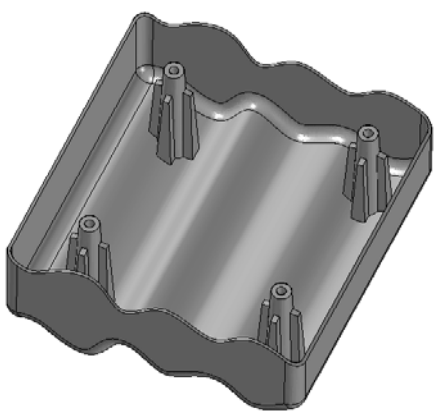


Figure 8-49 *Mounting bosses with draft*

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By default, the **Parallel Plane** option is selected in the **Create-From Options** drop-down list. If the face on which you want to project the profiles is curved, you can select the base reference plane that is parallel to the face.

Mounting Boss Step

This step will automatically be invoked when you select the parallel plane. On doing so, the sketching environment is invoked and the **Mounting Boss Location** button will be chosen in the **Features** group of the **Home** tab. In addition, the profile of the mounting boss will be attached to the cursor. Now, you can place the profiles of the mounting boss on the selected plane to place the mounting boss.

You can also modify the mounting boss options by choosing the **Mounting Boss Options** button. This button will be available in the command bar when you choose the **Mounting Boss Location** button from the **Features** group of the **Home** tab. Figure 8-50 shows the profiles of the four mounting bosses placed on a parallel plane.

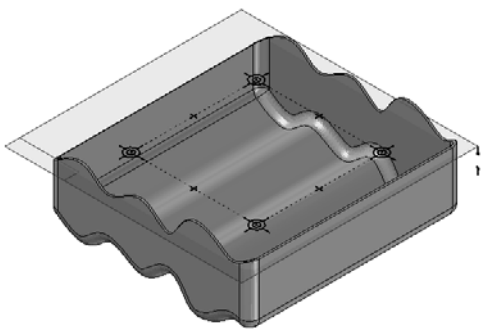


Figure 8-50 *Profiles of four mounting bosses*

Extent Step

This step will automatically be invoked when you choose the **Close Sketch** button from the sketching environment of the **Mounting Boss** step. In this step, you can specify the side for creating mounting boss. You can move the cursor on the side of the face where you want to project the profiles and click to accept the side. As soon as you specify the side, the preview of the mounting boss will be displayed. If the feature is correct, you can choose the **Finish** button to accept the feature; else, choose the **Options** button or the button of any other step to modify the options.

REORDERING FEATURES

While working on designs, you may sometime need to reorder the features. By reordering features, you can change the sequence in which the features were created in the model. For example, in the model shown in Figure 8-51, cavities were created first, followed by the thin wall feature. As a result, a thin wall is also created around the cavities, resulting in a protrusion feature. The original model required is the one shown in Figure 8-52.

To resolve this problem, Solid Edge allows you to change the order of the feature creation in the model. You can move a feature before or after another feature. However, note that the reordering is possible only between the features that are independent of each other. For example, if a part of any feature is dependent on another feature, then you cannot reorder the dependent feature above the parent feature.

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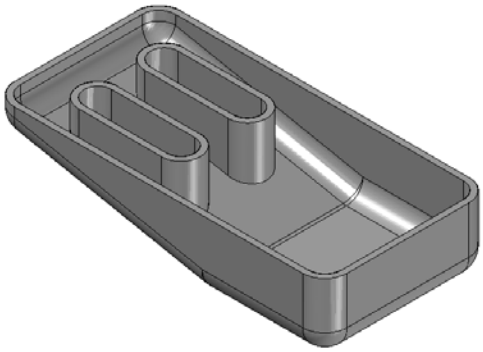


Figure 8-51 *Thin wall created around cavities*

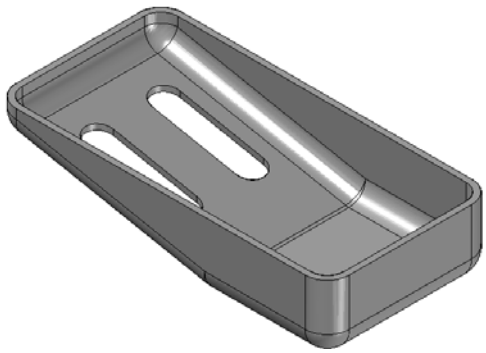


Figure 8-52 *Original model*

In Solid Edge, the features are reordered using the **PathFinder**. Select the feature in the **PathFinder** and drag it above or below the other features. If a feature cannot be dragged above a feature in the **PathFinder**, then you cannot reorder the feature before it because the selected feature is dependent on the feature above which you want to drag it. However, if the feature is not dependent, a green arrow will be displayed on the left of the feature in the **PathFinder** while you reorder it. Figure 8-53 shows the thin wall feature being dragged above the cutout feature to reorder it before the cutout feature. If you reorder the thin wall feature before the cutout, you will get the model shown in Figure 8-52..

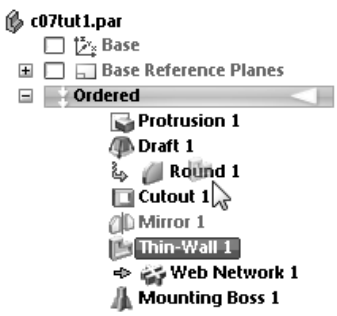


Figure 8-53 *Reordering features in the* ***PathFinder***

TUTORIALS

Tutorial 1 — Synchronous and Ordered

In this tutorial, you will create the model shown in Figure 8-54. Its dimensions are given in the drawing views shown in Figure 8-55. Save the model with the name *c08tut1.par* at the location given below:

\Solid Edge\c08 **(Expected time: 30 min)**

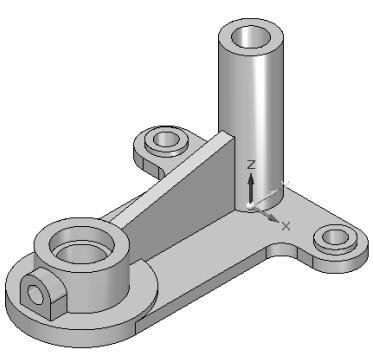


Figure 8-54 *Model for Tutorial 1*

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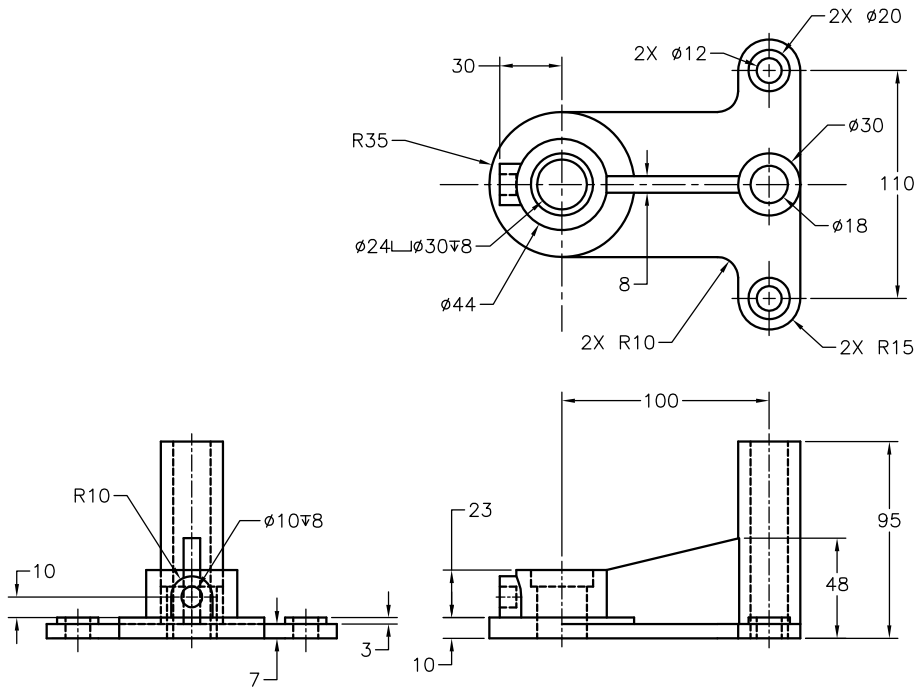


Figure 8-55 *Dimensions of the model for Tutorial 1*

The following steps are required to complete this tutorial:

a. Start a new part file and create the base feature on the top plane, refer to Figure 8-56.
b. Add the remaining protrusion features to the base feature, refer to Figure 8-57.
c. Add holes to the model, refer to Figure 8-58.
d. Switch to the **Ordered Part** environment and create the rib feature, refer to Figure 8-59 and Figure 8-60.
e. Save the model and close the file.

Creating the Base Feature

1. Start a new part document in Solid Edge. Create the sketch of the base feature on the top(xy) plane and then extrude it. The extruded base feature is shown in Figure 8-56.

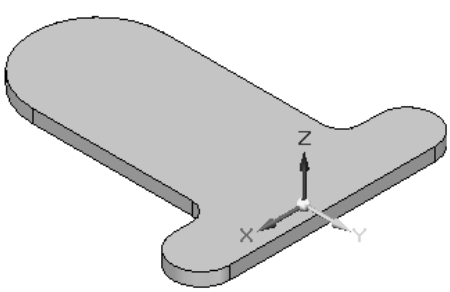


Figure 8-56 *Base feature of the model*

Adding the Remaining Extruded Features and Holes

1. Add the remaining protrusion features to the model, as shown in Figure 8-57.

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2. Create holes in the model, as shown in Figure 8-58.

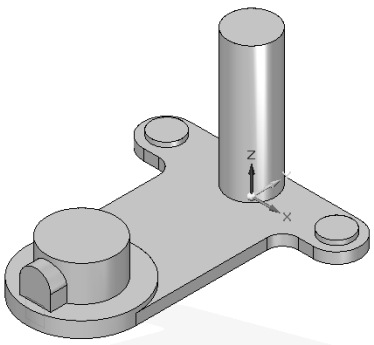


Figure 8-57 *Model after adding the remaining protrusion features*

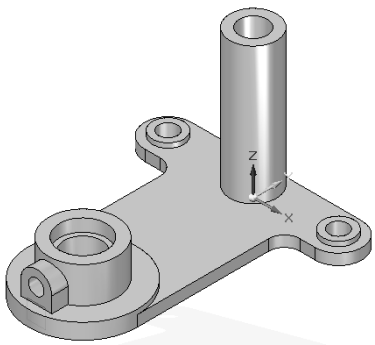


Figure 8-58 *Model after adding holes*

Creating the Rib Feature

Next, you need to switch to the **Ordered Part** environment and create the rib feature. The profile for this rib is a single line and needs to be extruded symmetrically on both sides of the sketching plane.

1. Choose the **Rib** tool from the **Thin Wall** drop-down in the **Solids** group of the **Home** tab; the **Sketch Step** is activated in the Command bar.

2. Select the **Coincident Plane** option from the **Create-From Options** drop-down list, if it is not already selected; you are prompted to click on a planar face or on a reference plane.

3. Select or create a reference plane passing through the center of the circular features in the middle of the model.

4. Draw a single line as the profile of the rib feature and add the required relationships and dimensions to it, as shown in Figure 8-59.

5. Exit the sketching environment; the **Direction Step** is activated and you are prompted to click to accept the displayed side or select the other side in the view.

6. Enter **8** in the **Thickness** edit box and specify the direction of the rib, as shown in Figure 8-60.

   As you specify the direction, the **Side Step** is invoked and you are prompted to click to accept the displayed side or select the other side in the view.

7. Move the cursor close to the profile and click when the preview of the rib feature is displayed symmetrically in both the directions of the sketch.

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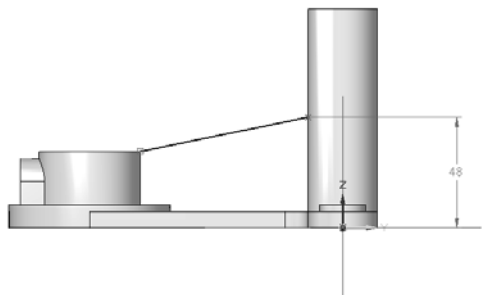


Figure 8-59 *Profile for the rib feature*

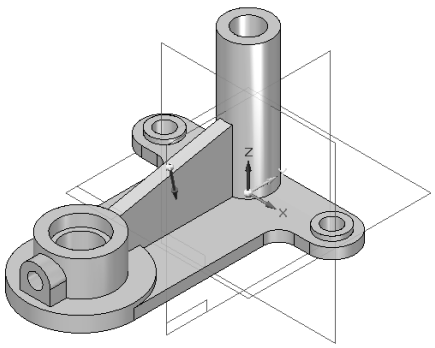


Figure 8-60 *Specifying the direction of the rib*

8. Choose **Finish** and then **Cancel** from the command bar to create the rib feature. The final model after creating the rib feature is shown in Figure 8-61.

Saving the Model

1. Save the model with the name *c08tut1.par* at the location given below and then close the file.

   \Solid Edge\c08

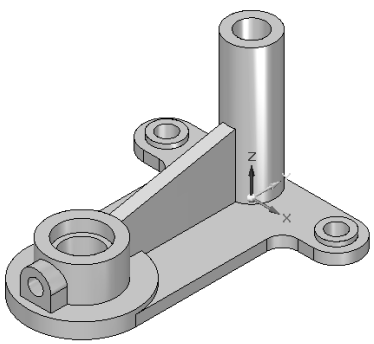


Figure 8-61 *Model after creating the rib feature*

Tutorial 2 Synchronous and Ordered

In this tutorial, you will create the model of the ice tray shown in Figure 8-62. Its dimensions are given in the drawing views shown in Figure 8-63. Save the model with the name *c08tut2.par* at the location given below:

\Solid Edge\c08 **(Expected time: 30 min)**

The following steps are required to complete this tutorial:

a. Start a new part file and create the base feature on the top plane, refer to Figure 8-64.
b. Add a draft to the base feature, refer to Figure 8-65.
c. Switch to the **Ordered Part** environment
d. Add a thin wall feature to the model, refer to Figure 8-66.
e. Create a web network in the model, refer to Figures 8-67 and 8-68.
f. Add rounds to the sharp edges of the model, refer to Figure 8-69.
g. Add another thin wall feature, refer to Figure 8-70.
h. Save the model and close the file.

Evaluation Copy. Do not reproduce. For information visit www.cadcim.com

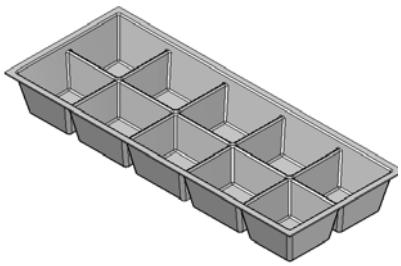


Figure 8-62 *Model for Tutorial 2*

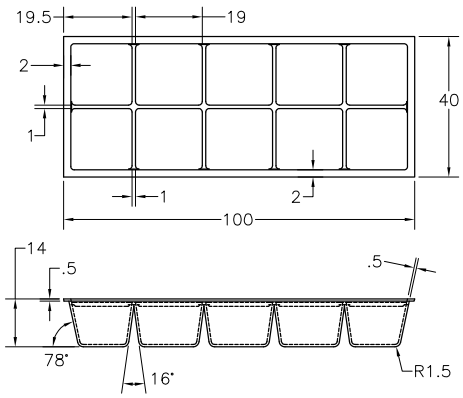


Figure 8-63 *Dimensions of the model for Tutorial 2*

Creating the Base Feature

1. Start a new part file and then create the base feature of the model on the xy plane. The base feature is a box of 100X40X14 size and is shown in Figure 8-64.

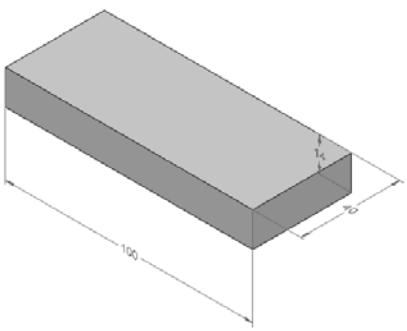


Figure 8-64 *Base feature of the model*

Adding the Draft to the Base Feature

You need to add a draft to the outer faces of the base feature using the top planar face of the base feature as the draft plane.

1. Choose the **Draft** tool from the **Solids** group of the **Home** tab; the **Draft** command bar is displayed.



On invoking the **Draft** tool, you are prompted to click on a planar face or on a reference plane as you need to define the **Draft Plane**.

2. Select the top planar face of the base feature as the draft plane.

Next, you need to select the faces to be tapered.

3. Select all the four side-faces of the model to add the draft and then enter **12** in the **Draft angle** edit box.

Next, you need to define the direction of the draft creation.

4. Specify the direction of draft creation inward by using the Draft direction handle.

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5. Right-click in the drawing area to create the draft. The model after adding the draft is shown in Figure 8-65.

Adding the Thin Wall Feature (Ordered Part)

Next, you need to scoop out the material from inside the model to create a thin wall model. This is done by using the **Thin Wall** tool. Also, you need to remove the top face of the model while creating the thin wall.

1. Switch to the **Ordered Part** environment and choose the **Thin Wall** tool from the **Solids** group of the **Home** tab; the **Thin Wall** command bar is displayed with the **Common Thickness** step activated. Also you are prompted to key in a common thickness value.

2. Enter **2** in the **Common thickness** edit box and press ENTER; the **Open Faces** step is invoked and you are prompted to click on a face chain.

3. Select the top planar face of the model as the face to be removed and then right-click to accept the selection.

4. Choose the **Preview** button and then choose **Finish** to create the thin wall feature. The model after creating the thin wall feature is shown in Figure 8-66.

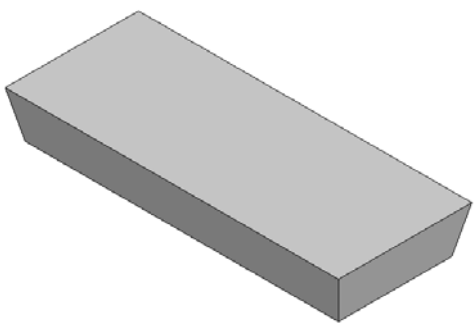


Figure 8-65 *Model after adding the draft*

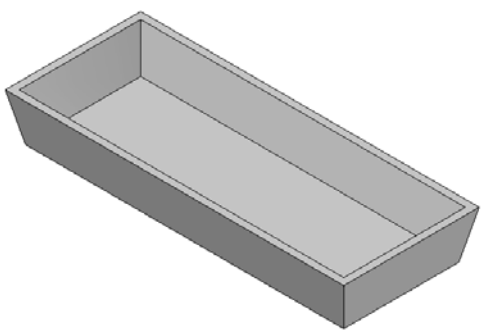


Figure 8-66 *The model after adding the thin wall feature*

Creating the Web Network

Next, you need to create the web network to accommodate the ice cubes. To create the web network, you need to use an open profile consisting of mutually perpendicular lines. Note that this profile needs to be created on a reference plane located at an offset distance of 2 from the top planar face of the thin wall model.

1. Choose the **Web Network** button from the **Thin Wall** drop-down in the **Solids** group; the **Web Network** command bar is displayed with the **Sketch Step** activated. Also, you are prompted to click on a planar face or on a reference plane.

2. Define a plane parallel to the top planar face of the thin wall feature. The plane should be offset 2 units in the downward direction.

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3. Draw the profile of the web network, as shown in Figure 8-67.

4. Exit the sketching environment; the **Direction Step** is invoked and you are prompted to click to accept the displayed side or select the other side in the view.

   You will notice that only one line is selected in the preview. But, it is just for the display. While creating the web network, all lines will be used.

5. Enter **1** in the **Thickness** edit box and move the cursor to the lower side of the model to create the web network in the downward direction.

   To add a draft of 8 degrees to the web network, you need to invoke the **Treatment Step** manually.

6. Choose the **Treatment Step** beside the **Direction Step** in the **Web Network** command bar. Next, choose the **Draft** button from the **Treatment Step**.

7. Enter **8** in the **Angle** edit box. Choose the **Flip 1** button from the command bar to make sure that the draft is applied in the outward direction.

8. Choose the **Preview** button and then choose the **Finish** button from the command bar to complete the web network, see Figure 8-68.

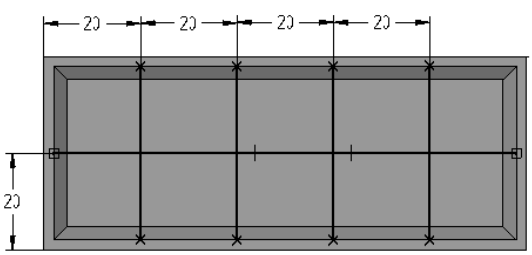


Figure 8-67 *Sketch for the web network*

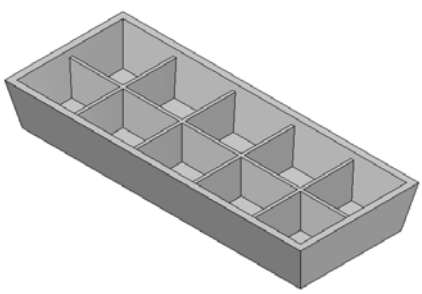


Figure 8-68 *Model after creating the web network*

Creating Rounds

1. Add a round of radius 1 mm to the inner edges of the cavities created by the web network.

2. Next, add a round of radius 0.5 unit to the top face of the web network. The model after adding rounds is shown in Figure 8-69.

Adding the Thin Wall Feature

Next, you need to add a thin wall feature such that all side faces and the bottom face of the model are removed.

1. Choose the **Thin Wall** button from the **Solids** group; the **Thin Wall** command

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bar is displayed and the **Common Thickness** step is activated. Hence, you are prompted to key in a common thickness value.

2. Enter **0.5** in the **Common thickness** edit box and press ENTER; the **Open Faces** step is invoked and you are prompted to click on a face chain.

3. Select the four side-faces and the bottom face of the model as the faces to be removed and then right-click to accept the selection.

4. Choose **Preview** and then the **Finish** button to create the thin wall feature. The final model of the ice tray after creating the thin wall feature is shown in Figure 8-70.

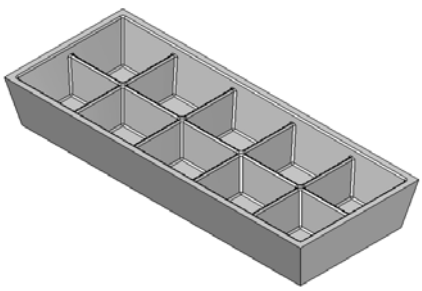


Figure 8-69 *Model after adding rounds*

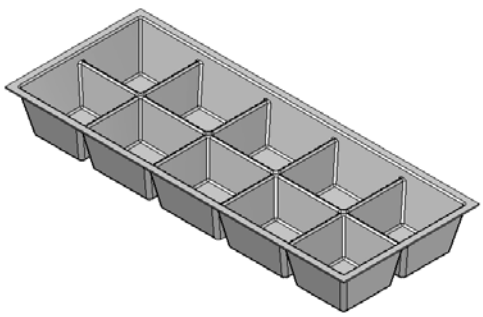


Figure 8-70 *Final model of the ice tray*

Saving the Model

1. Save the model with the name *c08tut2.par* at the location given below and then close the file.

   \Solid Edge\c08

Tutorial 3 Synchronous and Ordered

In this tutorial, you will create the model of the cover shown in Figure 8-71. Its dimensions are given in Figure 8-72. The outer fillet in Figure 8-72 is removed for the purpose of dimensioning. The radius of this fillet is 8. A draft of 1 degree needs to be added to the base feature of the model. The parameters of the mounting bosses are given next.

Boss diameter = 4, hole diameter = 2, hole depth = 5, rib offset = 3, rib grade = 10 degrees, rib extent = 1, rib taper = 10-degrees, rib thickness = 1.

Save the model with the name *c08tut3.par* at the location given below:

\Solid Edge\c08 **(Expected time: 45min)**

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Figure 8-71 *Model for Tutorial 3*

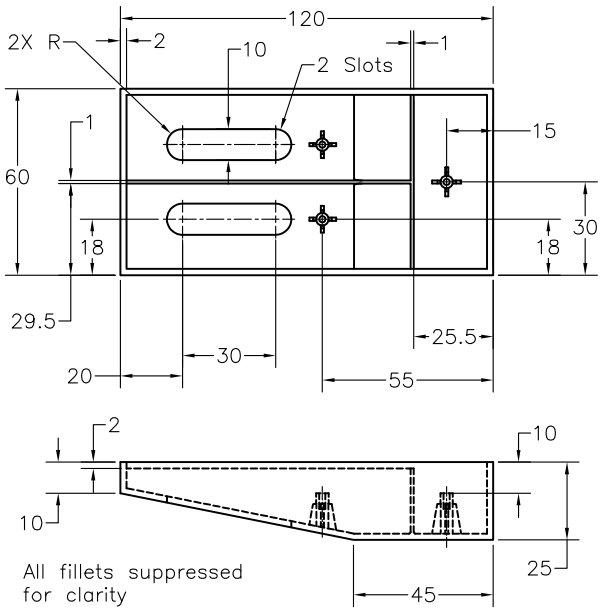


Figure 8-72 *Dimensions of the model for Tutorial 3 with the fillets removed for clarity*

The following steps are required to complete this tutorial:

a. Start Solid Edge in the **ISO Part** environment. Create the base feature on the front (xz) plane, refer to Figure 8-73.
b. Add a draft to the base feature.
c. Add rounds to the sharp edges of the model, refer to Figure 8-74.
d. Add a thin wall feature to the model, refer to Figure 8-75.
e. Create two cutouts in the model, refer to Figure 8-76.
f. Create a web network in the model, refer to Figure 8-78.
g. Switch to the **Ordered Part** environment and add mounting bosses to the model, refer to Figure 8-80.
h. Save the model and close the file.

Creating the Base Feature

1. Start Solid Edge in the **ISO Part** environment and then select the front(xy) plane as the sketching plane for the protrusion feature.

2. Create the profile of the base feature and extrude it symmetrically. The base feature of the model is shown in Figure 8-73.

Adding the Draft to the Base Feature

As mentioned earlier, drafts are added for the easy removal of component from casting. Therefore, you need to add draft to the side walls of this model before you proceed further.

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To do so, you need to use the top face as the draft plane.

1. Choose the **Draft** tool from the **Solids** group of the **Home** tab; the **Draft QuickBar** is displayed.

   You need to add draft with the top face of the base feature as the draft plane. By default, the **From plane** radio button is selected in the **Draft Options** dialog box. Therefore, you do not need to invoke the **Draft Options** dialog box to select this option.

   When you invoke the **Draft** tool, the **Draft Plane** button is activated and you are prompted to click on a planar face or on a reference plane.

2. Select the top planar face of the base feature as the draft plane; the **Draft Faces** button is activated.

3. Select all four side-faces of the model to add the draft. Enter **1** in the **Draft Angle** edit box and press ENTER.

   Next, you need to set the direction of draft.

4. Click on the Draft Direction handle and make sure that the lower half of the handle lies inside the model.

5. Right-click in the drawing area to create the draft.

6. Add a round of radius 8 to the sharp edges of the model, as shown in Figure 8-74.

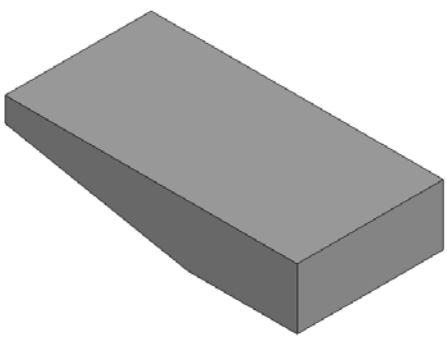


Figure 8-73 *Base feature of the model*

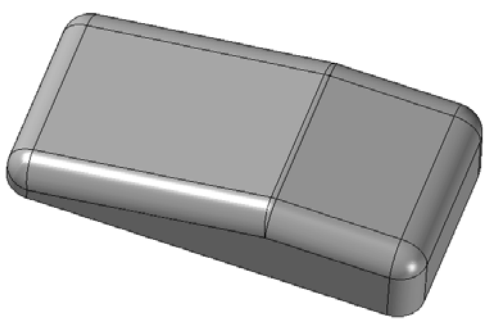


Figure 8-74 *Viewing the model from the bottom after adding the draft and the round*

Adding the Thin Wall Feature

Next, you need to scoop out the material from inside the model to create a thin wall model. This is done using the **Thin Wall** tool. Also, you need to remove the top face of the model while creating a thin wall.

1. Choose the **Thin Wall** tool from the **Solids** group; the **Thin Wall QuickBar** is displayed and the **Open Faces** button is activated. In addition, the handle and the edit box are displayed on the part.



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2. Enter **2** in the edit box; you are prompted to click on a face chain.

3. Select the top planar face of the model as the face to be removed and then right-click to accept the selection. The model after creating the thin wall feature is shown in Figure 8-75.

Creating Cutouts

1. Create two cutouts in the model, as shown in Figure 8-76.

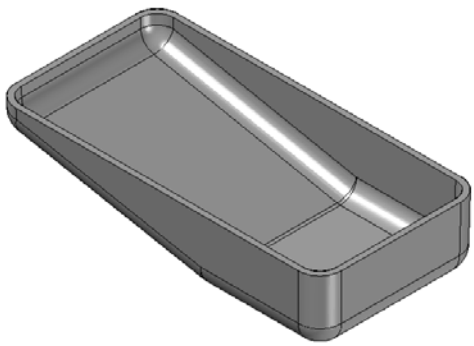


Figure 8-75 *Model after creating the thin wall feature*



Figure 8-76 *Model after creating the cutouts*

Creating the Web Network

Next, you need to create a web network by using an open profile that consists of two mutually perpendicular lines. Note that this profile needs to be created on a reference plane located at an offset distance of 2 units below the top planar face of the thin wall model.

1. Define a plane parallel to the top planar face of the thin wall feature. The plane should be offset 2 units inside the model.

2. Draw the profile for the web network, as shown in Figure 8-77.

3. Choose the **Web Network** tool from the **Thin Wall** drop-down in the **Solids** group; the **Web Network** command bar is displayed and the **Select Sketch Step** is activated. Also, you are prompted to select the sketch.



4. Right-click to accept the sketch; the **Direction Step** is invoked and you are prompted to click to accept the displayed side or select the other side in the view.

   You will notice that only one line is selected in the preview. But, this is just for the display. While creating the web network, both lines will be used.

5. Enter **1** in the **Thickness** edit box and move the cursor to the bottom of the model to create the web network in the downward direction.

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6. Choose the **Finish** button and then the **Cancel** button from the command bar to complete the web network, see Figure 8-78.

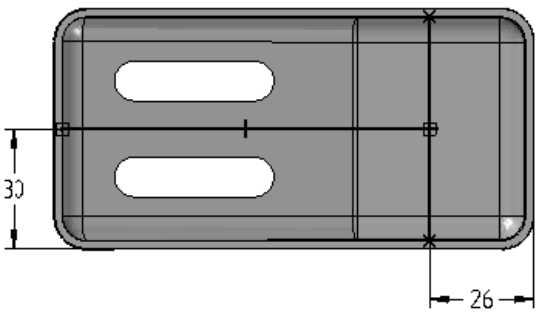


Figure 8-77 *Profile for the web network*

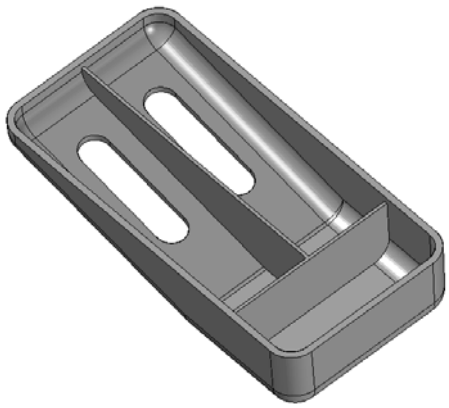


Figure 8-78 *Model after creating the web network*

Creating Mounting Bosses (in the Ordered Part environment)

Next, you need to create mounting bosses. To do so, you need to switch to the **Ordered Part** environment because the **Mounting Boss** tool is available only in the **Ordered Part** environment. The profiles of the mounting bosses need to be placed on a reference plane located at an offset distance of 10 from the top planar face of the thin wall model.

1. Choose the **Mounting Boss** tool from the **Thin Wall** drop-down in the **Solids** group of the **Ribbon**.



The **Mounting Boss** command bar is displayed and the **Plane Step** is activated. Also, you are prompted to click on a planar face or on a reference plane. Note that you need to define a reference plane parallel to the top planar face of the thin wall feature.

2. Define a plane parallel to the top planar face of the thin wall feature. The plane should be offset 10 units inside the model.

As you define the parallel plane, the **Mounting Boss Location** step is activated and the sketching environment is invoked. Now, before placing the mounting boss profile, you need to modify the parameters of the mounting boss. To do so, follow the steps given below.

3. Make sure the **Mounting Boss Location** button is chosen in the **Features** group of the **Ribbon**. Next, choose the **Mounting Boss Options** button from the command bar.

4. Set the parameters in the **Mounting Boss Options** dialog box, based on the values given in the tutorial statement.

5. Place three instances of the mounting boss profiles and then add the required dimensions, as shown in Figure 8-79. It is recommended that you select the edge of the model as the first entity and the mounting boss profile as the second entity while the dimensioning.

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6. Exit the sketching environment and specify downward as the direction of the feature creation. Choose the **Finish** button and then the **Cancel** button to exit the tool. The final model of the cover after creating the mounting bosses is shown in Figure 8-80.

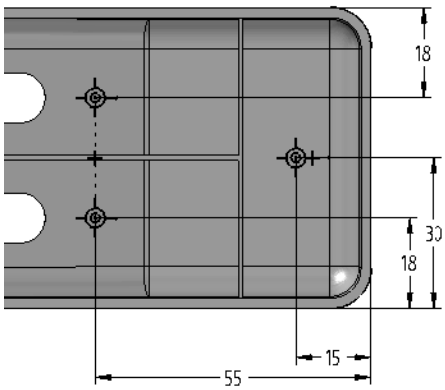


Figure 8-79 *Partial view of the model with profiles for mounting bosses*



Figure 8-80 *Final model after creating the mounting bosses*

Saving the Model

1. Save the model with the name *c08tut3.par* at the location given below and then close the file.

\Solid Edge\c08

Self-Evaluation Test

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

1. By adding a thin wall feature, you are allowed to scoop out the material from a model and make it hollow from inside. (T/F)

2. The **Taper** radio button needs to be selected to create threads on a taper feature. (T/F)

3. Internal threads are created in holes or circular cut features, and external threads are created on the external surface of a cylindrical feature. (T/F)

4. Ribs are defined as thin wall-like structures used to bind joints together so that they do not fail under an increased load. (T/F)

5. The _____ features are used to accommodate fasteners in plastic components.

6. In Solid Edge, features are reordered using the _____.

7. The _____ tool is used to taper the selected faces of a model.

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8. After setting the options in the **Mounting Boss Options** dialog box, the **Mounting Boss** tool works in _____ steps.

9. You can add a thin wall feature to a particular region of a model by using the _____ tool.

10. In the **Lip** tool, the amount of material to be added or removed is defined by a _____ whose width and height can be specified based on the requirement.

Review Questions

Answer the following questions:

1. Which of the following tools enables you to create a network of webs using open entities?

   (a) **Lip** (b) **Rib**
   (c) **Web Network** (d) **Thin Wall**

2. Which of the following tools enables you to add a taper to the selected faces of a model?

   (a) **Draft** (b) **Taper**
   (c) **Rib** (d) None of these

3. Which of the following tools enables you to create a vent in an existing model by defining its boundary, ribs, and spars?

   (a) **Draft** (b) **Taper**
   (c) **Rib** (d) **Vent**

4. Which of the following tools is used to add a thin wall to the entire model?

   (a) **Draft** (b) **Thin Region**
   (c) **Rib** (d) **Thin Wall**

5. Which of the following steps is used to place the profiles of the mounting boss on the plane?

   (a) **Mounting Boss Location** (b) **Sketch Step**
   (c) **Profile Step** (d) None of these

6. Which of the following buttons is chosen to extend the rib feature to the adjacent features, even if the profile does not extend to them?

   (a) **Extend Profile** (b) **Extend**
   (c) Both of these (d) None of these

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7. The **Lip** tool enables you to add a lip to a model by adding the material or by adding a groove to the model by removing the material. (T/F)
8. In Solid Edge, you can create internal or external threads by using the **Thread** tool. (T/F)

9. In Solid Edge, you can add drafts in five ways. (T/F)

10. Using the **Thin Wall** tool, you can also remove some of the faces of a model or apply different wall thicknesses to some of them. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 8-81. The dimensions of the model are shown in the same figure. After creating the model, save it with the name *c08exr1.par* at the location given below:

\Solid Edge\c08 **(Expected time: 30 min)**

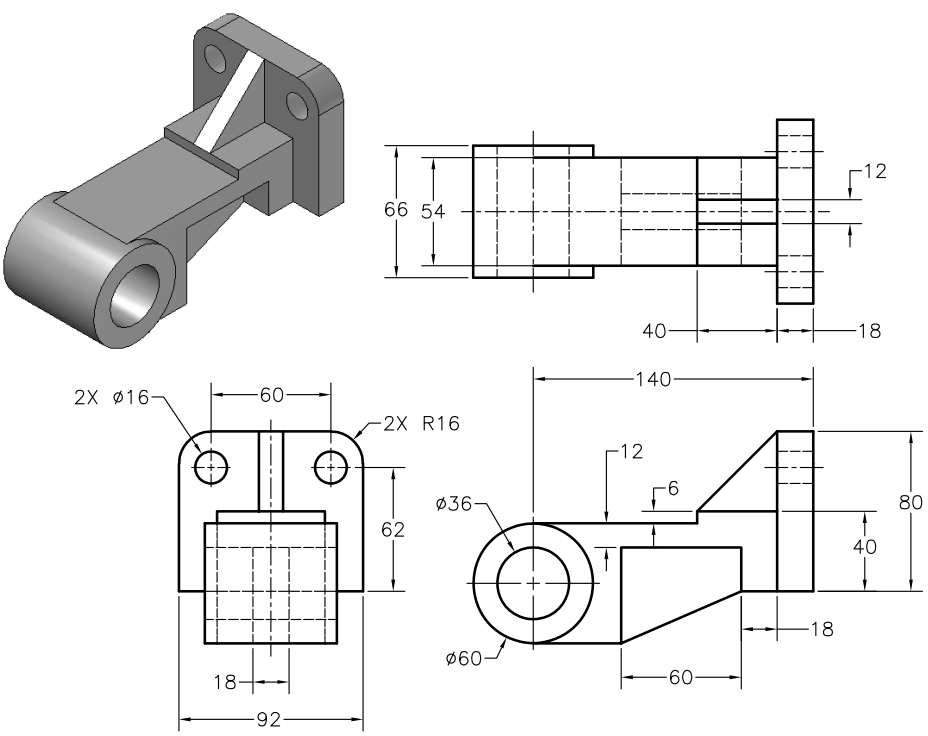


Figure 8-81 *The model and dimensions for Exercise 1*

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Exercise 2

Create the model shown in Figure 8-82. Its dimensions are given in Figure 8-83. Save it with the name *c08exr2.par* at the location given below:

\Solid Edge\c08 **(Expected time: 30 min)**

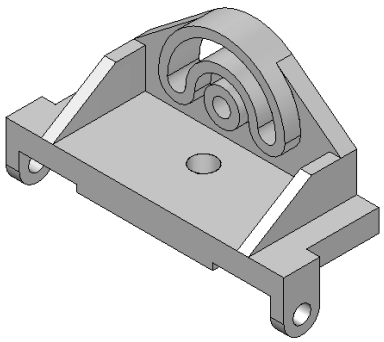


Figure 8-82 *Model for Exercise 2*

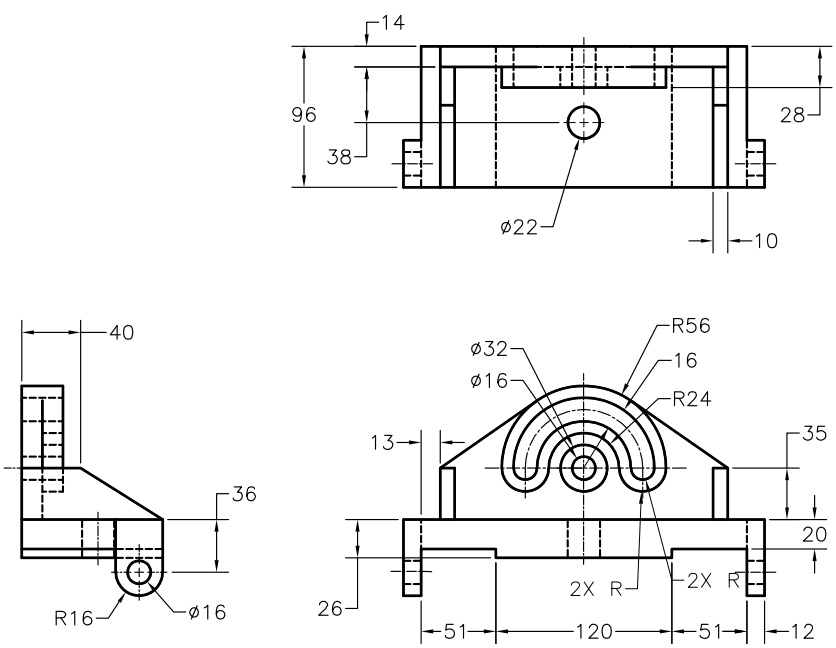


Figure 8-83 *Dimensions of the model*

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Answers to Self-Evaluation Test

1. T, **2.** F, **3.** T, **4.** T, **5.** mounting boss, **6. PathFinder**, **7. Draft**, **8.** three, **9. Thin Region**, **10.** rectangle