

A 3D CAD model of a mechanical assembly, likely a pump or valve component. It features a main cylindrical body with a flange at the top, a central shaft, and a lever arm extending from the side. The model is rendered in a light gray color with some internal features highlighted in yellow and green. The background is white with faint corner crop marks.

Chapter 8

Advanced Modeling Tools-II

Learning Objectives

After completing this chapter, you will be able to:

- *Use the Thread tool for creating external and internal threads.*
- *Add drafts to models.*
- *Add lip features.*
- *Create thin wall features.*
- *Create 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 INTERNAL OR EXTERNAL THREADS

Toolbar: Features > Hole > 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, and external threads are created on the external surface of a cylindrical feature. Note that you need to select only standard size holes or cylinders to create threads. These standard sizes are available in the **Diameter** edit box of the **Hole Options** dialog box. In Solid Edge, the faces that are threaded are shaded in green. To create threads, invoke the **Thread** tool; the **Thread Options** dialog box is displayed, as shown in Figure 8-1. The options available in this dialog box are discussed next.

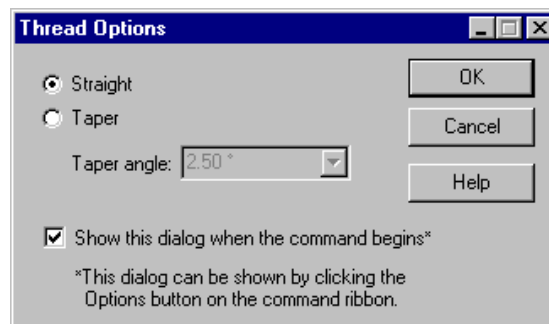


Figure 8-1 The *Thread Options* dialog box

Straight

This radio button is selected to create threads on a straight feature.

Taper

This radio button is selected to create threads on a tapered feature.

Taper angle

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

After specifying the parameters in the **Thread Options** dialog box, this tool works in three steps that are discussed next.



Tip. If you try to create threads on holes or cylinders with non-standard diameters, a warning box will be displayed, informing you that this diameter is not listed in the *HOLES.txt* file.

Select Cylinder Step

This step is automatically invoked when you exit the **Thread Options** dialog box. It enables you to select the cylinder or hole to create threads.

Cylinder End Step

This step will be automatically invoked after you select a cylinder or a hole to create threads. In this step, you can select one of the ends of the selected cylinder or hole as the edge from where the threads will be offset.

Parameters Step

This step enables you to specify the parameters of the thread that you want to create. The options available in the ribbon bar of this step are discussed next.

Offset

This edit box is used to specify the distance between the start of the thread and the cylinder end selected in the second step.

Depth

This drop-down list is used to specify the depth up to which the thread will be created. The options available in this drop-down list are discussed next.

To cylinder extent

This is the default option and it creates threads through the entire length of the cylinder or hole.

Finite value

This option enables you to create threads up to a specified depth. When you select the **Finite value** option, the **Thread Depth** edit box is invoked on the right of this drop-down list. You can specify the depth value in this edit 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 on the size of the selected cylinder or hole. You can select any option from this drop-down list to create threads.



Note

*If the hole or the cylinder feature that you selected does not have a standard diameter, then no option will be available in the **Type** drop-down list and the threads will not be created.*

Thread unit

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

ADDING DRAFTS TO THE MODEL

Toolbar: Features > Add Draft



Adding a draft is the process of tapering the selected faces of a model for their easy removal from casting during manufacturing. You can add a draft using the **Add Draft** tool. In Solid Edge, you can add drafts using four options. You can select the required option to create a draft from the **Draft Options** dialog box, as shown in Figure 8-2. This dialog box will be displayed when you choose the **Draft Options** button from the **Add Draft** ribbon bar. In this chapter, you will learn to create a draft using the first two options. The remaining options will be discussed in the later chapters.

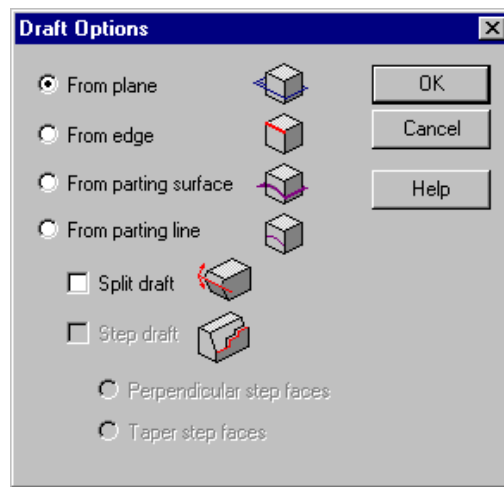


Figure 8-2 The Draft Options dialog box

Creating a Draft Using the From plane Option

This is the default option for creating a draft and is automatically selected in the **Draft Options** dialog box. You can create a draft from a plane in the following three steps:

Draft Plane Step

When you invoke the **Add Draft** tool or exit the **Draft Options** dialog box after selecting the option to create the draft, this step will be activated and you will be prompted to click on a planar face or a reference plane. You can also use the **Create-From Options** drop-down list from the **Add Draft** ribbon bar to create a new reference plane. The draft plane is a plane whose normal is used to define the draft angle. Figures 8-3 and 8-4 show the draft added to a model. In both figures, all parameters are the same, except the draft plane. In Figure 8-3, the top planar face is selected as the draft plane and in Figure 8-4, the bottom planar face is selected as the draft plane.

Select Face Step

This step will be automatically invoked as soon as you define the draft plane. This step enables you to select one or more faces on which the draft will be added. You can use the

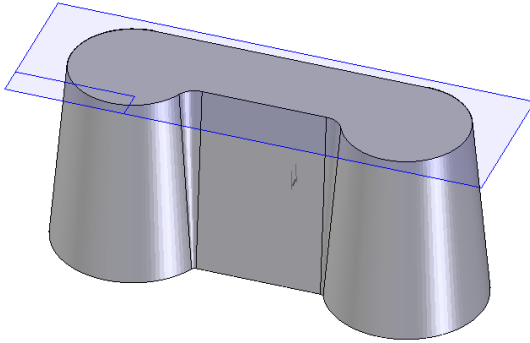


Figure 8-3 Draft added with top face as the draft plane

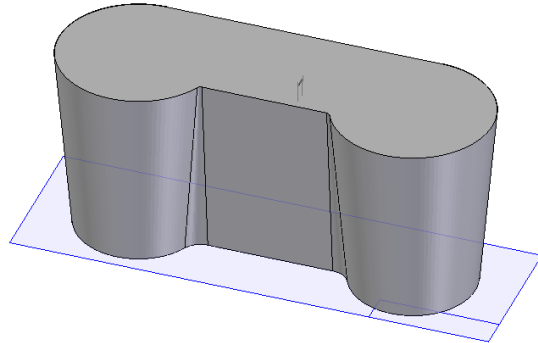


Figure 8-4 Draft added with bottom face as the draft plane

options in the **Select** drop-down list to select the faces to draft. Next, enter the draft angle in the **Draft Angle** edit box. Note that you cannot enter a negative draft angle value because you need to define the side of the draft in the next step.

After setting the parameters in the ribbon bar in the **Select Face Step**, right-click to accept the selection and then choose the **Next** button from the ribbon bar to proceed to the next step. You can also right-click again to proceed to the next step.

Draft Direction Step

This step enables you to define the direction of the draft. In this step, once the edge of the draft plane is selected, a two-sided arrow will be displayed on it. This arrow is used to define the direction of the draft. You can move the cursor in the drawing window and specify a point to define the direction of the draft. Figures 8-5 through 8-8 show the draft directions and the resulting drafts.

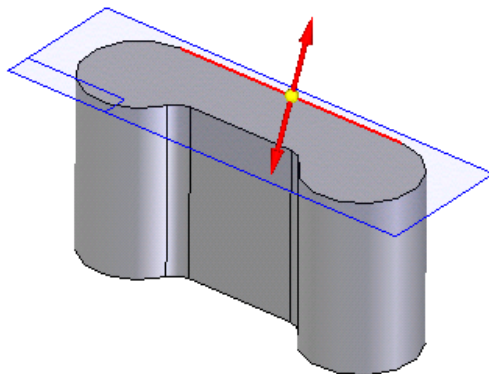


Figure 8-5 Defining the draft direction using an inclined line on the upper horizontal edge

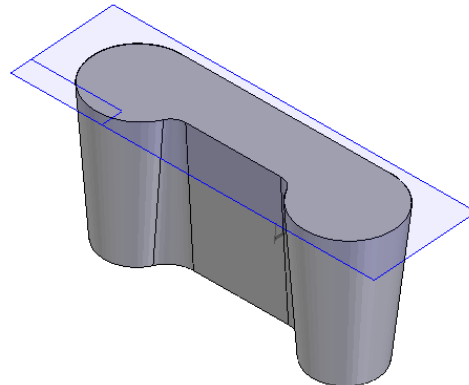


Figure 8-6 Resulting draft

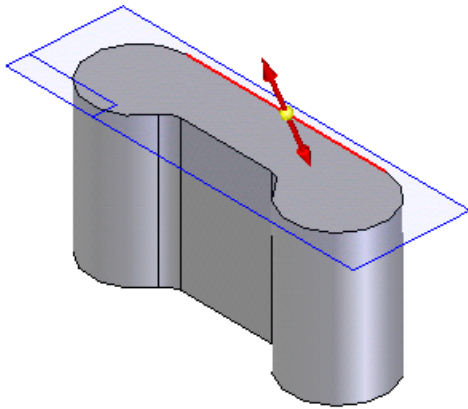


Figure 8-7 Defining the draft direction

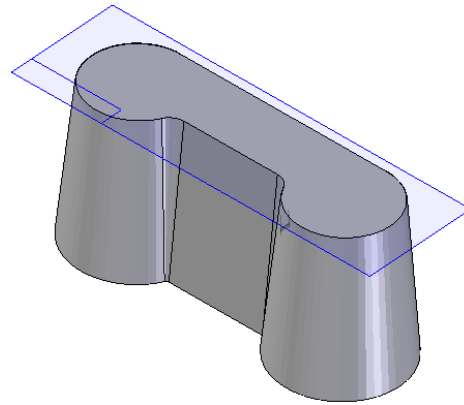


Figure 8-8 Resulting draft

Creating a Draft Using the From edge Option

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.

ADDING RIBS TO THE MODEL

Toolbar: Features > Rib



Ribs are defined as thin wall-like structures which are used to bind the joints together so that they do not fail under an increased load. In Solid Edge, ribs are created using an open profile, see Figures 8-9 and 8-10.

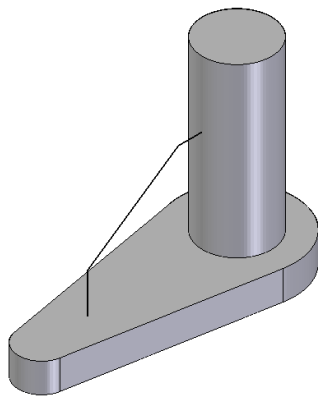


Figure 8-9 Open profile to create a rib

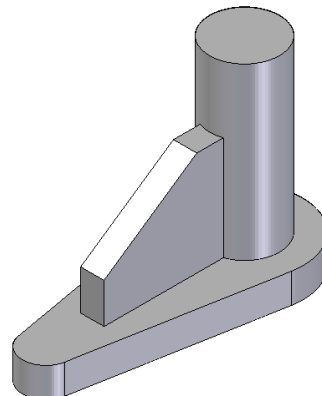


Figure 8-10 Resulting rib feature

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

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

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 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 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. Note that the rib feature will be successfully created only if you define that side for the feature creation in which the profile meets the faces of the existing features. If the feature cannot be created, an error symbol will be displayed, as shown in Figure 8-11. Figure 8-12 shows the preview of the direction in which the rib feature will be successfully created.

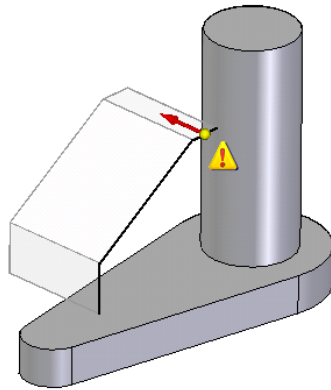


Figure 8-11 Preview of the direction in which the rib will not be created and the error symbol

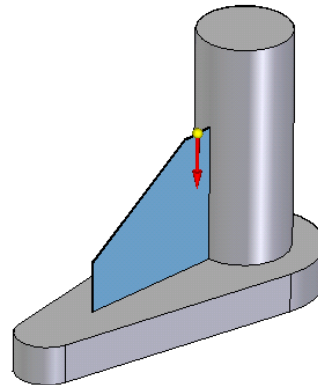


Figure 8-12 Preview of the direction in which the rib will be created

Side Step

The **Side Step** button will be active in the ribbon 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 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 8-13. Next, click to create the symmetric rib.

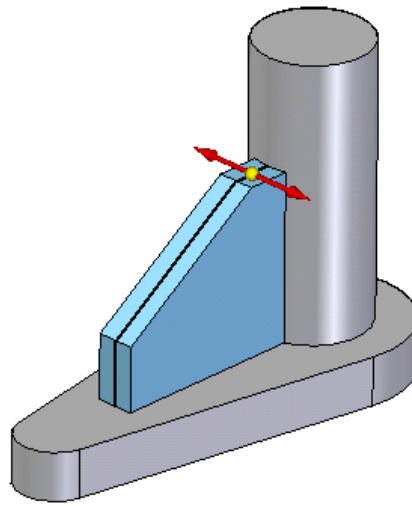


Figure 8-13 Preview of the symmetric rib

Note that in the **Direction** and **Side** steps, some additional options are available in the ribbon 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 8-14 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 8-15 shows the rib feature created using the same profile with the **Extend Profile** button chosen. As is evident in this figure, the rib feature has been extended to the adjacent features.

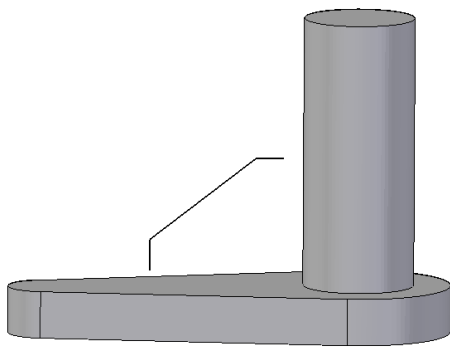


Figure 8-14 Open profile not extended to the adjacent features for creating rib feature

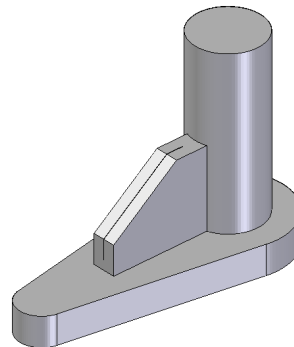


Figure 8-15 Rib feature extended to the adjacent features

No Extend

This button is chosen when you do not want to extend the rib to the adjacent faces. Figure 8-16 shows the profile and the resulting rib created by choosing the **Extend Profile** button and Figure 8-17 shows the rib feature created by choosing the **No Extend** button.

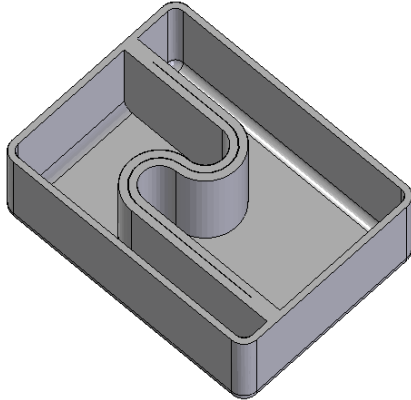


Figure 8-16 Rib feature extended to the adjacent faces using the **Extend Profile** button

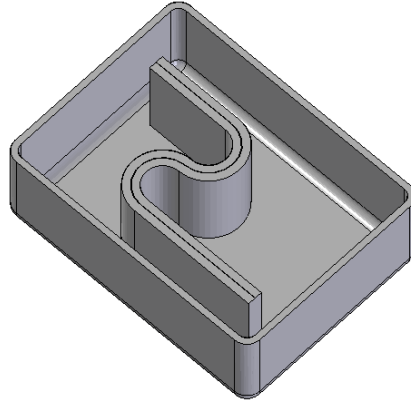


Figure 8-17 Rib feature not extended to the adjacent faces using the **No Extend** button

Extend to Next

This button is chosen when you want to extend the rib to the next features in the direction that you specified in the **Direction** step. Figure 8-18 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 specified in the **Direction** step. The depth of the rib can be specified in the **Depth** edit box that will be displayed on the right of the **Thickness** edit box on choosing the **Finite Depth** button. Figure 8-19 shows the rib feature created using this option.

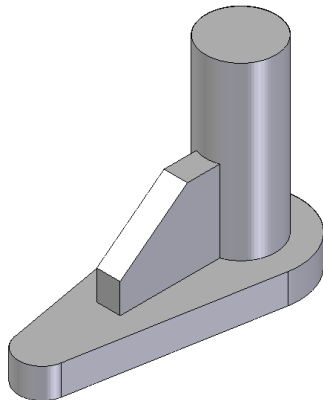


Figure 8-18 Rib feature created by extending the profile to the next features

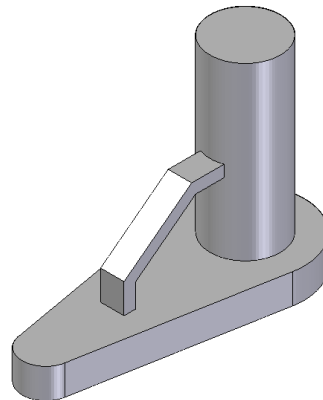


Figure 8-19 Rib feature created up to a finite depth

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.

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

Toolbar: Features > Thin Wall



By adding the thin wall feature, you will be able to scoop out the material from a model and make it hollow from inside. The resulting model will be a 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 8-20 shows a model with the thin wall feature added and the front face removed.

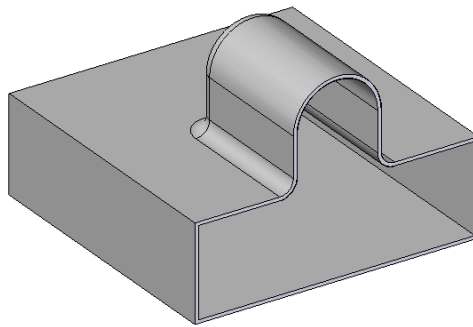


Figure 8-20 Model with the thin wall feature added

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 ribbon 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** drop-down 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 8-21 shows a thin wall model with the front and left side faces removed.

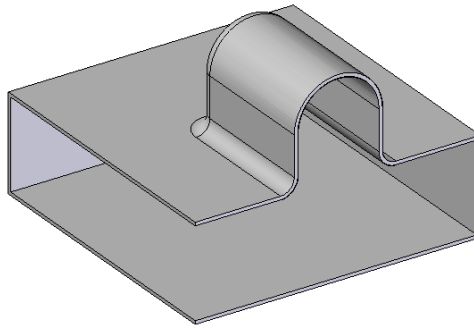


Figure 8-21 Thin wall model with the front and left faces removed

Unique Thickness

This step is not necessarily required to create the thin wall feature, and therefore, it will not be invoked automatically. You need to choose the button of this step to invoke it. You can use this step to select faces to which a different wall thickness will be applied. After selecting the face or faces, specify the unique thickness in the **Unique Thickness** edit box 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 until you have selected all the faces to which you want to add different wall thicknesses. Figure 8-22 shows a thin wall model with a unique thickness added at the left and bottom faces.

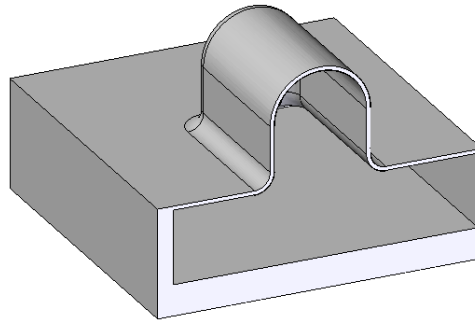


Figure 8-22 Thin wall model with different wall thicknesses

ADDING THIN WALL TO A PARTICULAR REGION

Toolbar: Features > 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 8-23. You can do this using the **Thin Region** tool.

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 can select the faces of the region to add the thin wall feature and specify the common thickness. Note that while selecting the faces, you need to make sure that they result in a closed volume. For example, to create the thin wall region shown in Figure 8-23, you need to select the top curved face, side tangent faces, back face, and the front face of the region, as shown in Figure 8-24.

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 8-23, you need to remove the front face.



Note

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

Capping Faces Step

Capping face can be considered as the face that defines the termination of the thin region. It is not necessary to select a capping surface. You can select any face of a model or an existing surface to define the capping faces. You can also define an offset value from the capping face

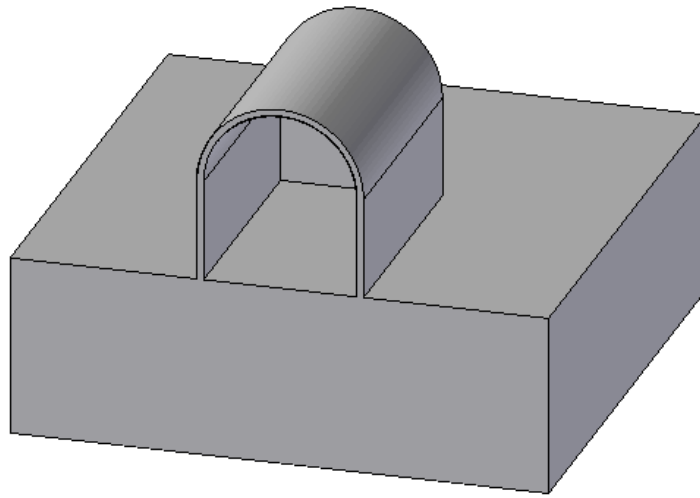


Figure 8-23 Model with the thin wall feature added to a particular region

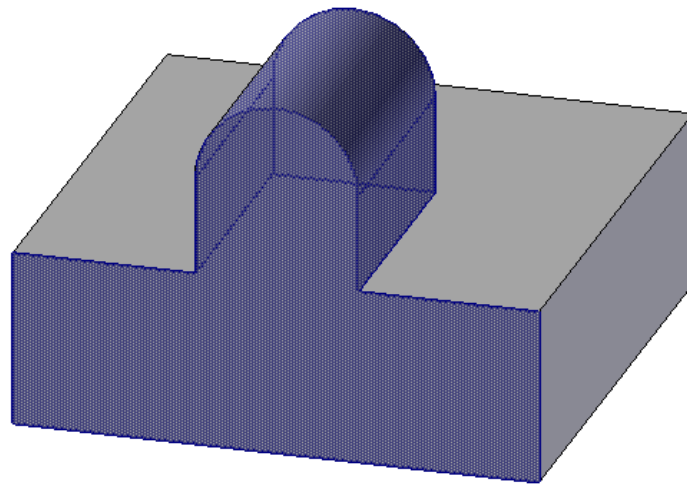


Figure 8-24 Selecting the faces of the model to create a thin wall region

using the **Offset** edit box that is displayed in the ribbon bar under this step. Figure 8-25 shows a surface used as the capping face and Figure 8-26 shows the same surface used as the capping face, but with an offset of 2. You will learn more about surfaces in the later chapters.

Figure 8-27 shows the preview of the model with the top planar face used as the capping face with an offset of 5.

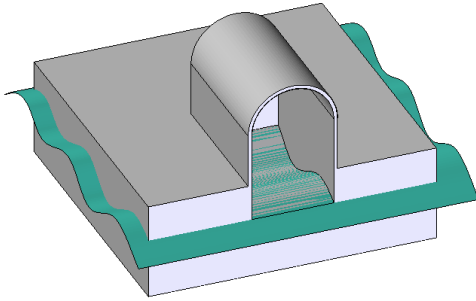


Figure 8-25 Surface used as the capping face

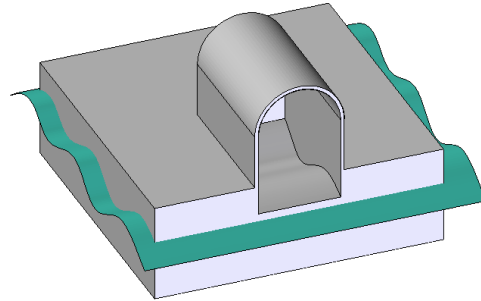


Figure 8-26 Surface used as the capping face with an offset of 2

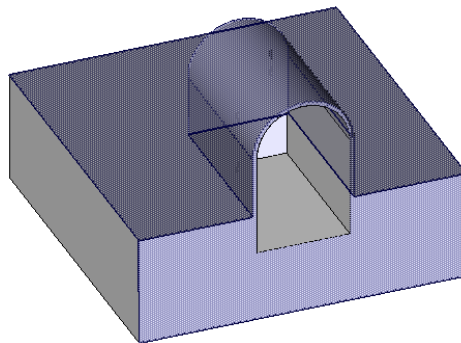


Figure 8-27 Top face used as the capping face with an offset of 5

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

Toolbar: Features > Rib > Lip



The **Lip** tool enables you to add a lip to the model by adding the 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.

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 a red rectangle is displayed at the start of the edge. Move the cursor around the start of the edge to specify various locations 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 8-28 through 8-31 show different positions of the rectangle and the resulting lip features.

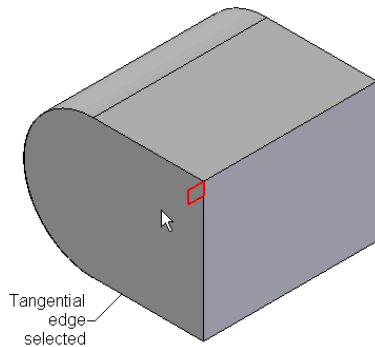


Figure 8-28 Location of the rectangle

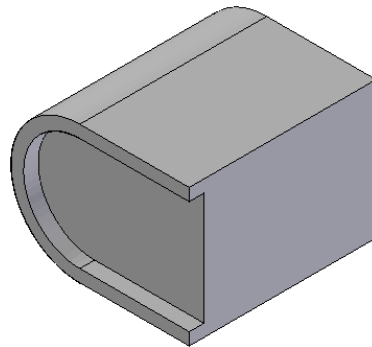


Figure 8-29 Resulting lip feature

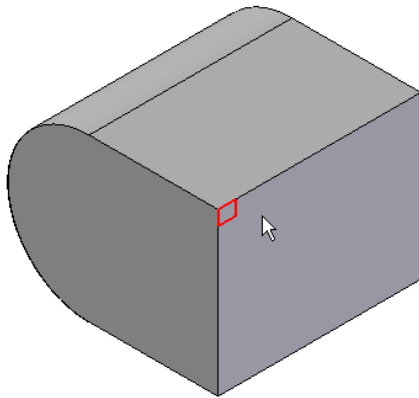


Figure 8-30 Location of the rectangle

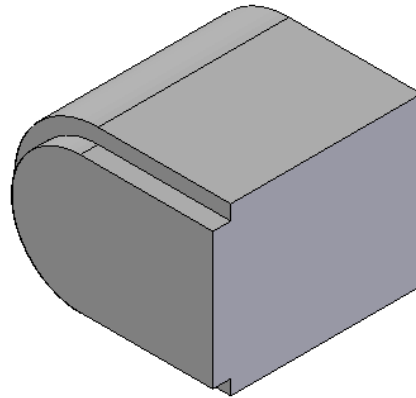


Figure 8-31 Resulting lip feature

CREATING WEB NETWORKS

Toolbar: Features > Rib > Web Network



The **Web Network** tool enables you to create a network of web using open entities, as shown in Figures 8-32 and 8-33.

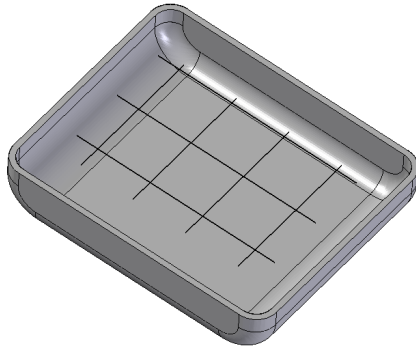


Figure 8-32 Thin wall model and a network of lines

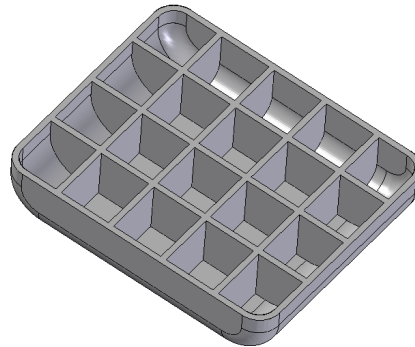


Figure 8-33 Resulting web network

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

Plane or 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 ribbon bar of this step are the same as those in the **Rib** tool. Figure 8-34 shows a web network in which the webs are not extended. Figure 8-35 shows a web network in which the webs are extended, but defined up to a finite depth.

Web Network Step

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



Tip. To select multiple individual entities, as shown in Figure 8-32, select the **Single** option from the **Select** drop-down list and then drag a box around the entities to select them.

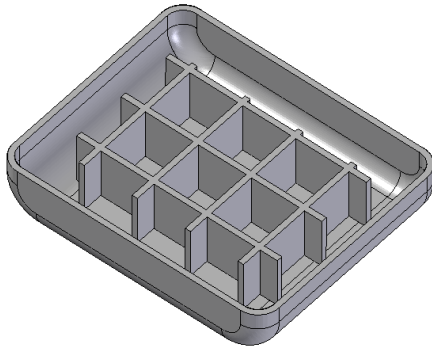


Figure 8-34 Web network with webs not extended

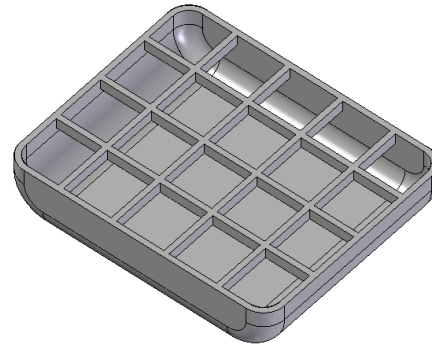


Figure 8-35 Webs defined up to a finite depth

CREATING VENTS

Toolbar: Features > Rib > 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 8-36 shows a model and a profile that defines the boundary. Note that in the vent, all the vertical lines are selected as ribs and all the horizontal lines are selected as spars. Figure 8-37 shows the resulting model with the vent.

To create the vent, invoke the **Vent** tool; the **Vent Options** dialog box will be displayed, as shown in Figure 8-38.

Vent Options Dialog Box Options

The options available 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 a setting, set the parameters in this dialog box, enter a name for the set parameter in the **Saved settings** edit box, and then choose the **Save** button. The saved settings will also be displayed on choosing the down arrow on the right of the **Vent Options** button in the ribbon bar. 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 the ribs and spars. Figure 8-39 shows a vent with the thickness of ribs and spars as 2. Figure 8-40 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.

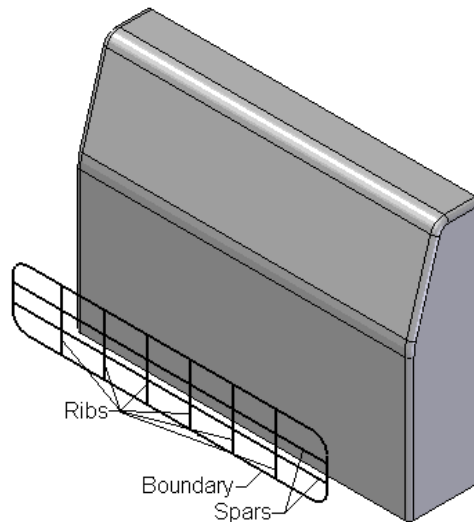


Figure 8-36 Parameters related to the vent

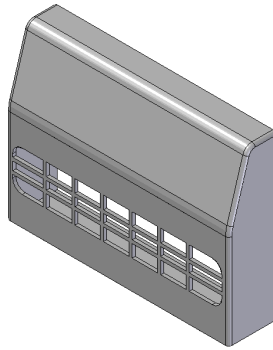


Figure 8-37 Resulting model with the vent

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 8-41 shows the ribs and spars with no extension and Figure 8-42 shows the 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 8-43 shows the ribs and spars starting at some offset from the face on which the profile is projected.



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.

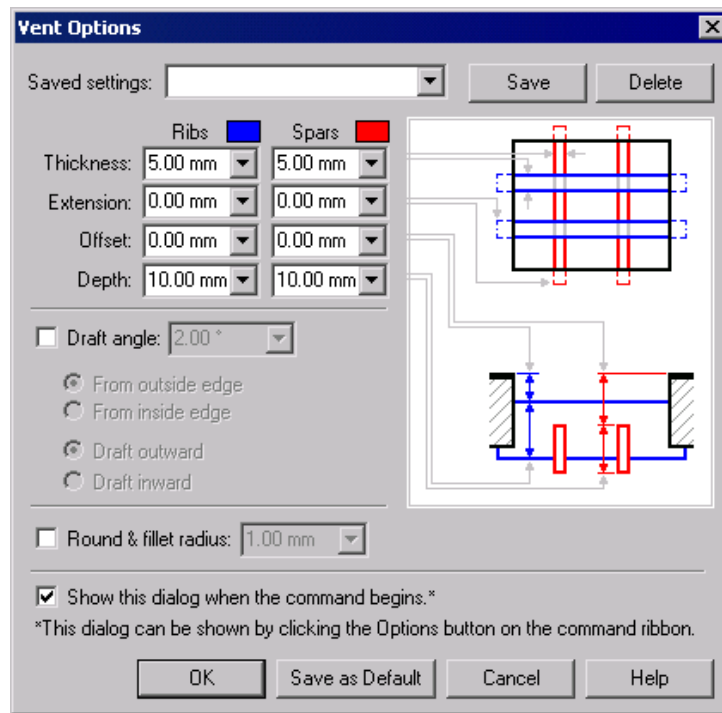


Figure 8-38 The Vent Options dialog box

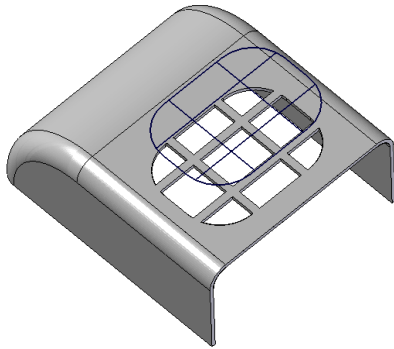


Figure 8-39 Ribs and spars thickness as 2

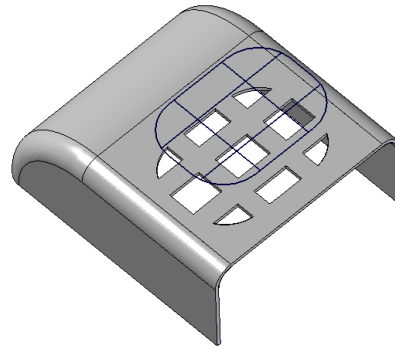


Figure 8-40 Ribs and spars thickness as 5

Depth Ribs/Spars

These edit boxes are used to specify the depth of the ribs and spars. You can have the same or different depth values for each of them. Figures 8-44 and 8-45 show vent features with different depth values for the ribs and spars.

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

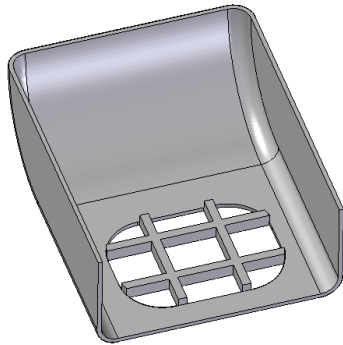


Figure 8-41 Ribs and spars not extended

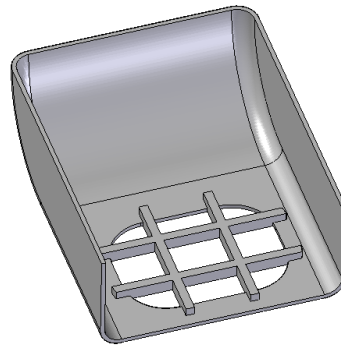


Figure 8-42 Ribs and spars extended beyond the boundary

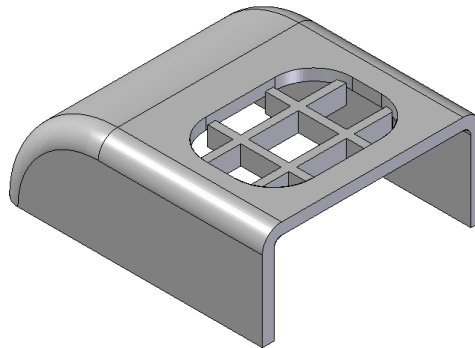


Figure 8-43 Ribs and spars starting at an offset from the top face

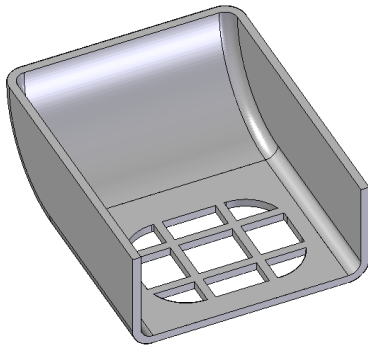


Figure 8-44 Ribs and spars depth = 2

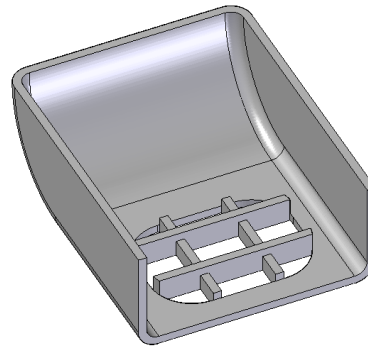


Figure 8-45 Ribs depth = 4, spars depth = 6

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 8-46 shows a vent with fillets and rounds.

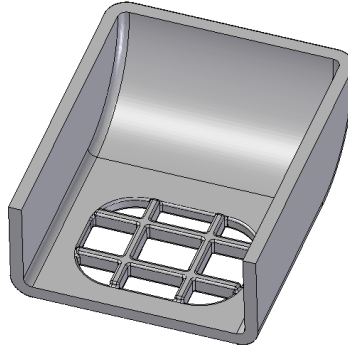


Figure 8-46 Vent with fillets and rounds

After specifying the parameters in the **Vent Options** dialog box, this 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** drop-down list to select a chain of entities or an individual entity.

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 ribbon bar of this step to define the extent.

CREATING MOUNTING BOSSES

Toolbar: Features > Rib > Mounting Boss



The **Mounting Boss** tool enables you to create mounting boss features, which are used in the plastic components to accommodate fasteners. Figure 8-47 shows a model with four mounting boss features.

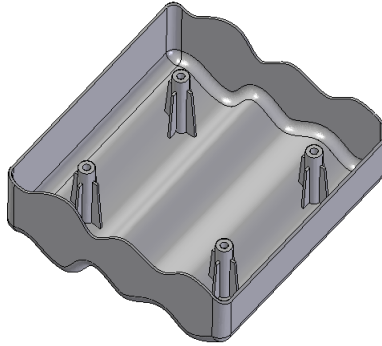


Figure 8-47 Model with four mounting bosses

To create mounting boss features, invoke the **Mounting Boss** tool; the **Mounting Boss** ribbon bar will be displayed. It is recommended that before proceeding further, you should set the parameters of the mounting boss features in the **Mounting Boss Options** dialog box, as shown in Figure 8-48. This dialog box can be invoked on choosing the **Mounting Boss Options** button in the ribbon bar.

Mounting Boss Options Dialog Box

The options available 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 a setting, set the parameters in this dialog box, enter a name in the **Saved settings** edit box, and then choose the **Save** button. The saved settings will also be displayed on choosing the down arrow on the right of the **Mounting Boss Options** button in the ribbon bar. You can delete the unwanted settings by selecting them from this drop-down list and choosing the **Delete** button.

Settings Area

The options available in this area are used to set the parameters of the mounting boss feature. All these options have a gray arrow on the right that leads to a parameter in the preview window. The preview window explains the use of the options available in the **Settings** area. These options are discussed next.

Boss diameter

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

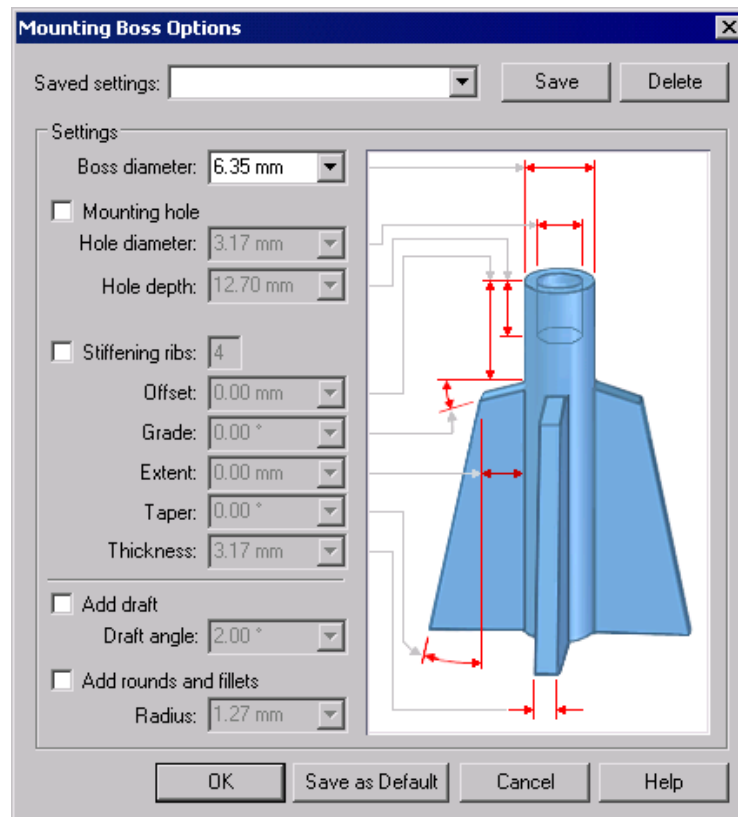


Figure 8-48 The Mounting Boss Options dialog box

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 available below this check box will be activated and you can specify the diameter and the depth of the hole in these edit boxes.

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.

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 now you can enter the draft angle in this edit box. Figure 8-49 shows a model with mounting bosses without the draft and Figure 8-50 shows the mounting bosses with the draft.

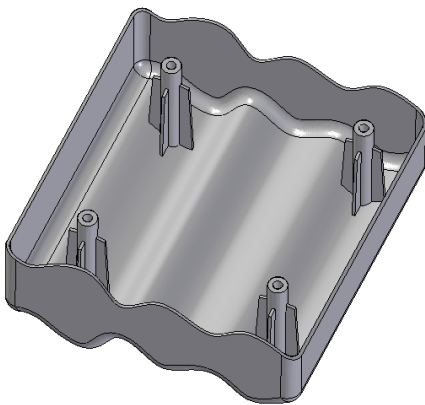


Figure 8-49 Mounting bosses without draft

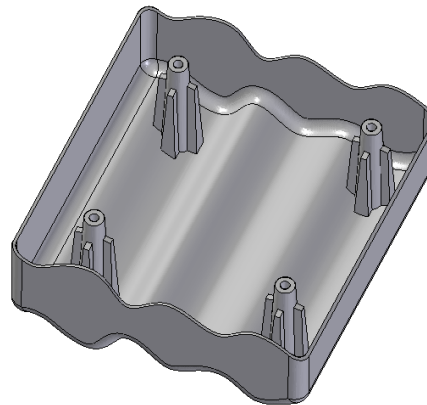


Figure 8-50 Mounting bosses with draft

All 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, this tool works in the following three steps:

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.

In this step, you need to place the profiles of the mounting bosses on a parallel plane and so

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 be automatically invoked when you select the parallel plane. In this step, you can place the profiles of the mounting boss on the selected plane. The sketching environment is invoked in this step and the **Mounting Boss Location** button is chosen in the **Features and Relationships** toolbar. Also, the profile of the mounting boss is attached to the cursor. You can click in the model to place the mounting boss.

You can also modify the mounting boss options by choosing the **Mounting Boss Options** button, which will be available in the ribbon bar when you choose the **Mounting Boss Location** button from the **Features and Relationships** toolbar. Figure 8-51 shows the profiles for four mounting bosses placed on a parallel plane.

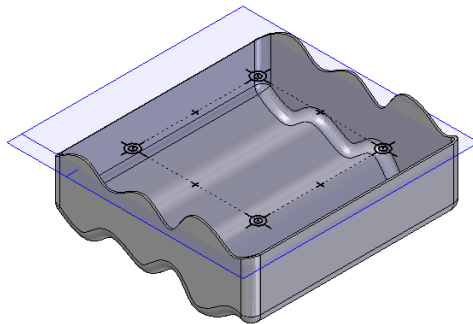


Figure 8-51 Profiles for four mounting bosses

Extent Step

This step will be automatically invoked when you choose the **Finish** button from the sketching environment in the **Mounting Boss** step. In this step, you can specify the side for creating the 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 **Mounting Boss 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, you can change the sequence in which the features were created in the model. For example, in the model shown in Figure 8-52, the 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 by you is the one shown in Figure 8-53.

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.

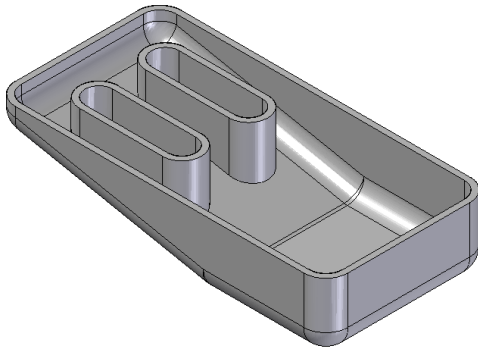


Figure 8-52 Thin wall created around cavities

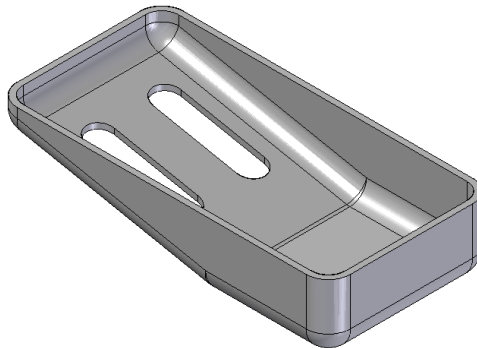


Figure 8-53 Original model required

In Solid Edge, the features are reordered using the **EdgeBar**. Select the feature in the **EdgeBar** and drag it above or below the other features. If a feature cannot be dragged above a feature in the **EdgeBar**, 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 **EdgeBar** while you reorder it. Figure 8-54 shows the thin wall feature being dragged above the cutout feature to reorder it before the cutout feature. When you reorder the thin wall feature before the cutout, you will get the model, as shown in Figure 8-53.

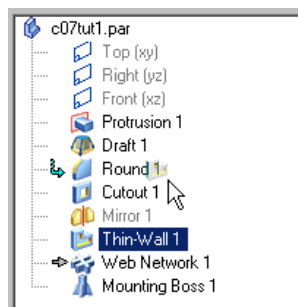
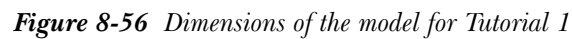


Figure 8-54 Reordering features in the **EdgeBar**

Tutorial 1

|Solid Edge|c08|c08tut1.par

Evaluation Chapter. Do not copy. Visit www.cadclm.com for details



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 8-57.
- b. Add the remaining protrusion features to the base feature, refer to Figure 8-58.
- c. Add holes to the model, refer to Figure 8-59.
- d. Create the rib feature, refer to Figure 8-62.
- e. Save the model and close the file.

Creating the Base Feature

1. Start Solid Edge in the **Part** environment and then select the top plane as the sketching plane for the base feature, as shown in Figure 8-57.

Adding the Remaining Protrusion Features and Holes

1. Add the remaining protrusion features to the model, as shown in Figure 8-58.
2. Create holes in the model, as shown in Figure 8-59.

Creating the Rib Feature

Next, you need to create the rib feature. The profile for this rib is a single line. It is extruded symmetrically on both sides of the sketching plane.

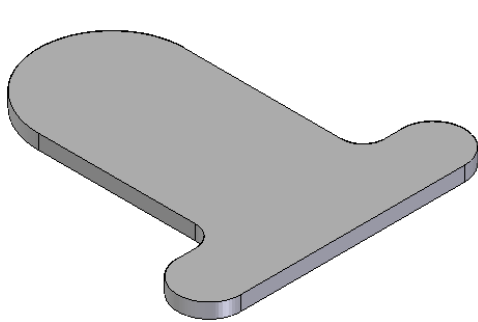


Figure 8-57 Base feature of the model

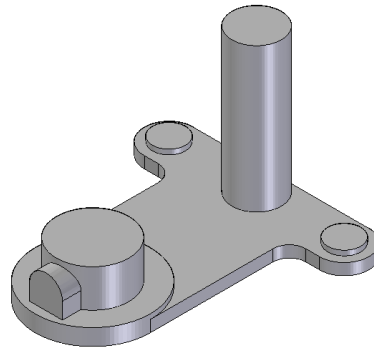


Figure 8-58 Model after adding the remaining protrusion features

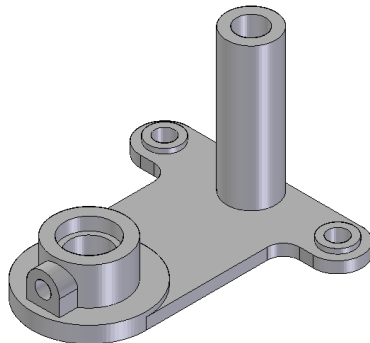


Figure 8-59 Model after adding holes

1. Choose the **Rib** button from the **Features** toolbar; the **Plane or Sketch Step** is activated and you are prompted to click on a planar face or a reference plane.
2. Select or create a reference plane passing through the center of the circular features in the middle of the model.
3. Draw a single line for the rib feature and add the required relationships and dimensions to it, as shown in Figure 8-60.
4. 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.
5. Enter the value **8** in the **Thickness** edit box and specify the direction, as shown in Figure 8-61.

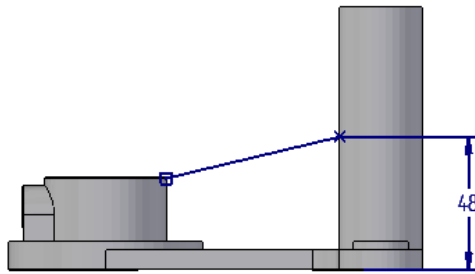


Figure 8-60 Profile for the rib feature

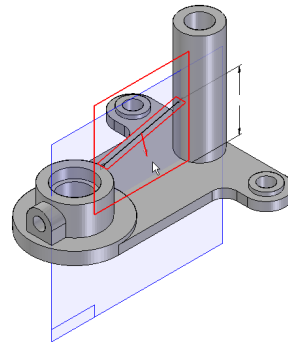


Figure 8-61 Specifying the direction of the rib

As soon 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.

6. Move the cursor close to the profile such that the preview of the rib feature is displayed symmetrically in both directions of the sketch. Click at this stage.
7. Choose **Finish** from the ribbon bar to create the rib feature. The final model, after creating the rib feature, is shown in Figure 8-62.

Saving the Model

1. Save the model with the name and location given below and then close the file.

`\\Solid Edge\\c08\\c08tut1.par`

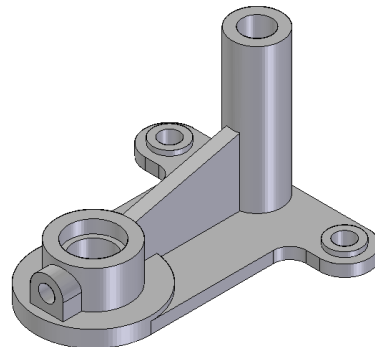


Figure 8-62 Model after creating the rib feature

Tutorial 2

In this tutorial, you will create the model of the ice tray shown in Figure 8-63. Its dimensions are given in the drawing views shown in Figure 8-64. Save the model with the name and the location given below:

`\\Solid Edge\\c08\\c08tut2.par`

(Expected time: 30 min)

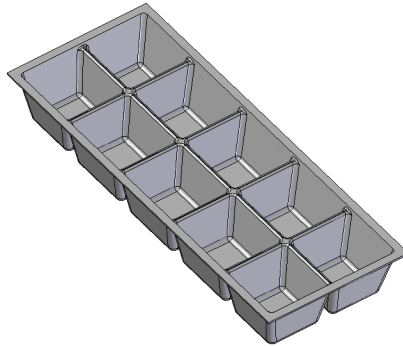


Figure 8-63 Model for Tutorial 2

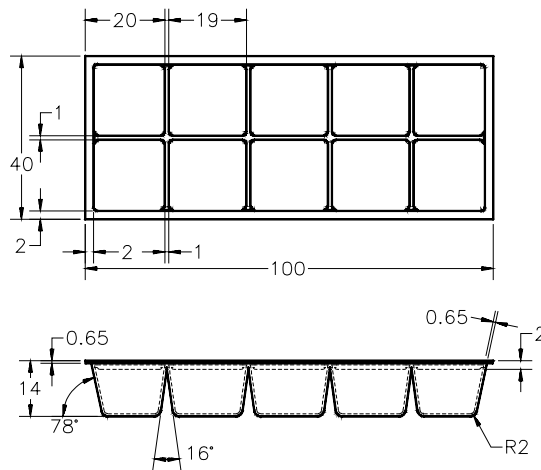


Figure 8-64 Dimensions of the model for Tutorial 2

The following steps are required to complete this tutorial:

- Start a new part file and create the base feature on the top plane, refer to Figure 8-65.
- Add a draft to the base feature, refer to Figure 8-66.
- Add a thin wall feature to the model, refer to Figure 8-67.
- Create a web network in the model, refer to Figures 8-68 and 69.
- Add rounds to the sharp edges of the model, refer to Figure 8-70.
- Add another thin wall feature, refer to Figure 8-71.
- Save the model and close the file.

Creating the Base Feature

1. Start a new part file and then create the base feature of the model, which is a box of 100X40X14 size. The base feature of the model is shown in Figure 8-65.

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 **Add Draft** button from the **Features** toolbar; the **Add Draft** ribbon bar is displayed.



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

2. Select the top planar face of the base feature as the draft plane; the **Select Faces Step** is invoked.
3. Select all the four side faces of the model to add the draft. Enter the value **12** in the **Draft Angle** edit box and press ENTER.
4. Choose **Next** from the ribbon bar to invoke the **Draft Direction** step. Move the cursor in the drawing window and click when the lower half of the line is inside the model.
5. Choose the **Finish** button to create the draft and then choose **Cancel** to exit this tool. The model, after adding the draft, is shown in Figure 8-66.

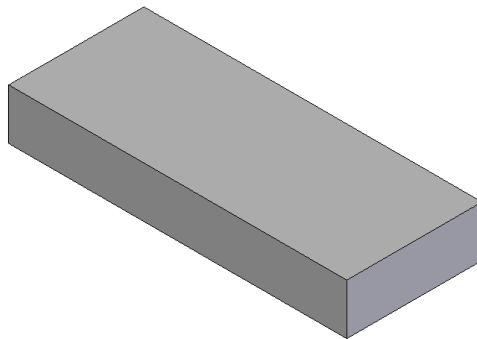


Figure 8-65 Base feature of the model

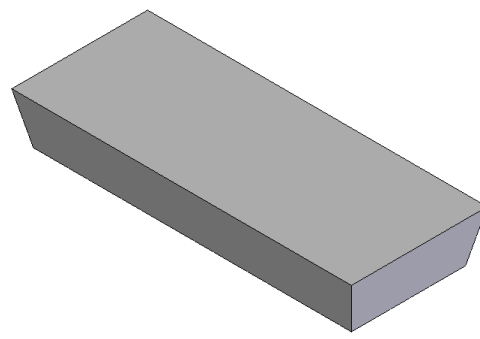


Figure 8-66 Model after adding the draft

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. You also need to remove the top face of the model while creating the thin wall.

1. Choose the **Thin Wall** button from the **Features** toolbar; the **Thin Wall** ribbon bar is displayed and the **Common Thickness Step** is activated. As a result, you are prompted to key in a common thickness value.



2. Enter the value **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 8-67.

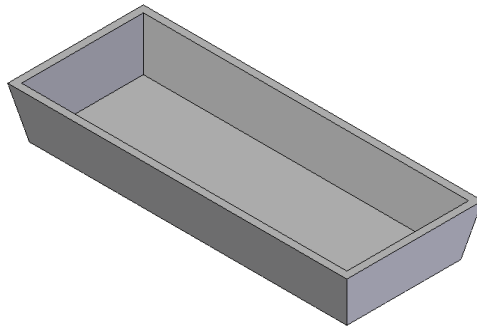



Figure 8-67 Model after adding the thin wall feature

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 **Rib** flyout in the **Features** toolbar; the **Web Network** ribbon bar is displayed and the **Plane or Sketch** Step is activated. As a result, you are prompted to click on a planar face or 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.
3. Draw the profile of the web network, as shown in Figure 8-68.
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 in the preview, only one line is selected. But it is just for 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.

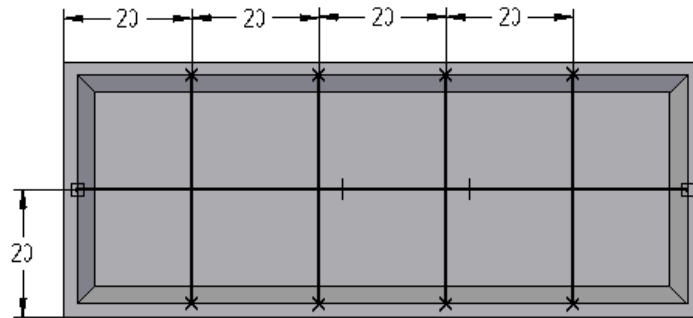




Figure 8-68 Sketch for the web network

To add a draft of 8-degrees to the web network, you need to invoke the **Web Network** step manually.


6. Choose the **Web Network** button from the ribbon bar to invoke the **Web Network** step.
7. Choose the **Draft** button and then enter **8** in the **Angle** edit box. Choose the **Flip 1** button 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 ribbon bar to complete the web network, see Figure 8-69.

Creating Rounds

1. Add a round of radius 1 to the inner edges of the cavities created by the web network. 
2. Next, add a round of radius 0.5 to the top face of the web network. The model, after adding the rounds, is shown in Figure 8-70.

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 **Features** toolbar; the **Thin Wall** ribbon bar is displayed and the **Common Thickness** step is activated. Hence, you are prompted to key in a common thickness value. 
2. Enter the value **0.65** 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.

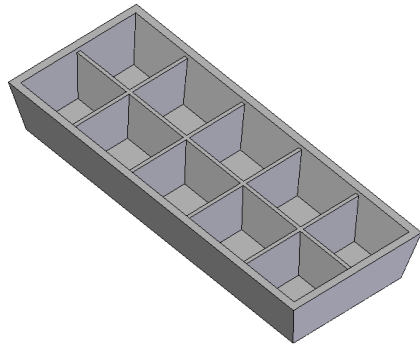


Figure 8-69 Model after creating the web network

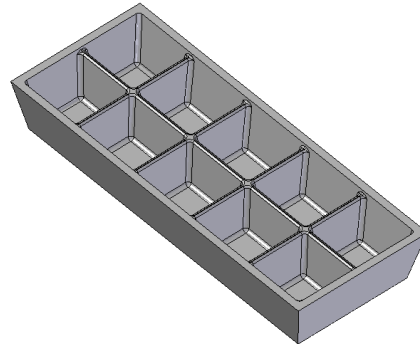


Figure 8-70 Model after adding the rounds

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 8-71.

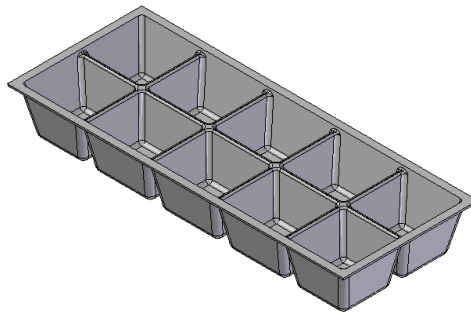


Figure 8-71 Final model of the ice tray

Saving the Model

1. Save the model with the name and location given below and then close the file.

`\\Solid Edge\\c08\\c08tut2.par`

Tutorial 3

In this tutorial, you will create the model of the cover shown in Figure 8-72. Its dimensions are given in Figure 8-73. The outer fillet in Figure 8-73 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 = 10degrees, rib extent = 1, rib taper = 10-degrees, rib thickness = 1.

Save the model with the name and the location given below:

`\\Solid Edge\\c08\\c08tut3.par`

(Expected time: 45min)

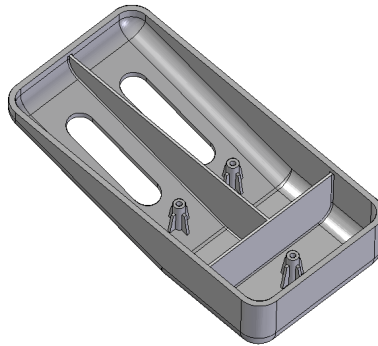


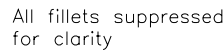
Figure 8-72 Model for Tutorial 3

The following steps are required to complete this tutorial:

- Start Solid Edge in the **Part** environment. Create the base feature on the front plane, refer to Figure 8-74.
- Add a draft to the base feature.
- Add rounds to the sharp edges of the model, refer to Figure 8-75.
- Add a thin wall feature to the model, refer to Figure 8-76.
- Create two cutouts in the model, refer to Figure 8-77.
- Create a web network in the model, refer to Figure 8-79.
- Add mounting bosses to the model, refer to Figure 8-81.
- Save the model and close the file.

Creating the Base Feature

- Start Solid Edge in the **Part** environment and then select the front plane as the sketching plane for the protrusion feature.
- Create the profile of the base feature and extrude it symmetrically. The base feature of the model is shown in Figure 8-74.



Adding Draft to the Base Feature

1. Choose the **Add Draft** button from the **Features** toolbar; the **Add Draft** ribbon bar is displayed.

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

2. Select the top planar face of the base feature as the draft plane; the **Select Face** step is invoked.
3. Select all four side faces of the model to add the draft. Enter the value **1** in the **Draft Angle** edit box and press ENTER.

4. Choose **Next** from the ribbon bar to invoke the **Draft Direction** step. Move the cursor in the drawing window and click when the lower half of the line is inside the model.
5. Choose the **Finish** button to create the draft and then choose **Cancel** to exit this tool.
6. Add a round of radius 8 to the sharp edges of the model, as shown in Figure 8-75.

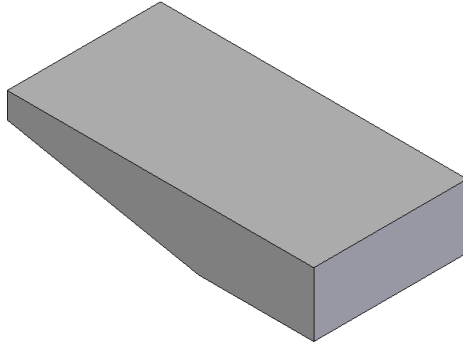


Figure 8-74 Base feature of the model

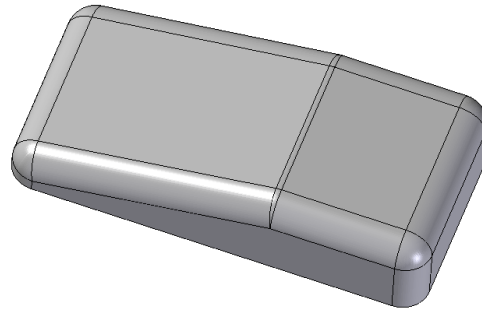



Figure 8-75 Viewing the model from the bottom after adding the draft and round

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. You also need to remove the top face of the model while creating a thin wall.

1. Choose the **Thin Wall** button from the **Features** toolbar; the **Thin Wall** ribbon bar is displayed and the **Common Thickness** step is activated. Therefore, you are prompted to key in a common thickness value. 
2. Enter the value **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 a thin wall feature. The model, after creating the thin wall feature, is shown in Figure 8-76.

Creating Cutouts

1. Create two cutouts in the model, as shown in Figure 8-77.

Creating the Web Network

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

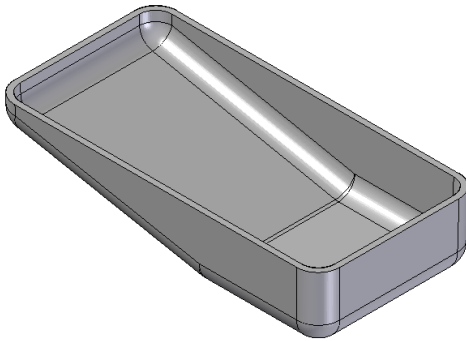


Figure 8-76 Model after creating the thin wall feature

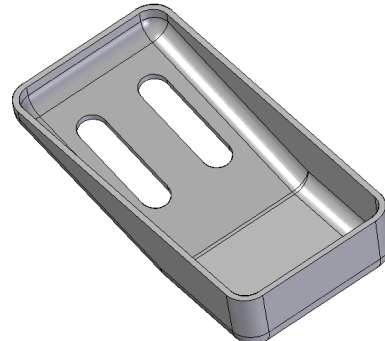



Figure 8-77 Model after creating the cutouts

1. Choose the **Web Network** button from the **Rib** flyout in the **Features** toolbar. The **Web Network** ribbon bar is displayed and the **Plane or Sketch Step** is activated. As a result, you are prompted to click on a planar face or reference plane. 
2. Define a plane parallel to the top planar face of the thin wall feature. The plane should be offset 2 units inside the model.
3. Draw the profile of the web network, as shown in Figure 8-78.

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 in the preview, only one line is selected. But this is just for 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.
6. Choose the **Finish** button from the ribbon bar to complete the web network, see Figure 8-79.

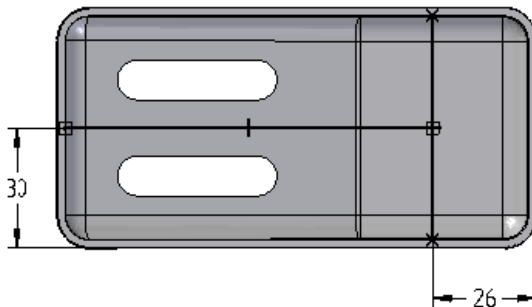


Figure 8-78 Profile for the web network

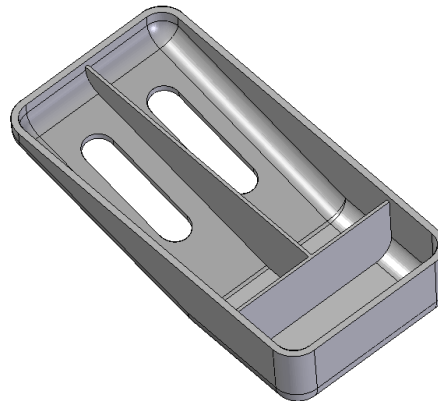


Figure 8-79 Model after creating the web network

Creating the Mounting Bosses

Next, you need to create the mounting bosses. 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** button from the **Web Network** flyout in the **Features** toolbar.



The **Mounting Boss** ribbon bar is displayed and the **Plane Step** is activated. Therefore, you are prompted to click on a planar face or 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 soon as you define the parallel plane, the **Mounting Boss Step** is activated and the sketching environment is invoked. Also, the **Mounting Boss Location** button is chosen in the **Features** and Relationship toolbar of the sketching environment. Before you start placing the mounting boss profile, you need to modify the parameters of the mounting boss.

3. Make sure that the **Mounting Boss Location** button is chosen in the **Features** toolbar. Next, choose the **Mounting Boss Options** button from the ribbon 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 8-80. It is recommended that you select the edge of the model as the first entity and the mounting boss profile as the second entity to add the dimension.

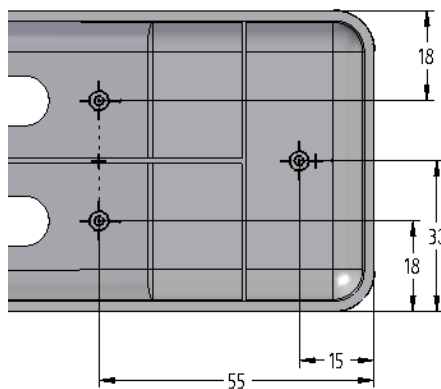


Figure 8-80 Partial view of the model with the profiles for mounting bosses

6. Exit the sketching environment and specify the direction of the feature creation downward. Choose the **Finish** button to complete the tool. The final model of the cover, after creating the mounting bosses, is shown in Figure 8-81.



Figure 8-81 Final model after creating the mounting bosses

Saving the Model

1. Save the model with the name and the location given below and then close the file.

\\Solid Edge\\c08\\c08tut3.par

Self-Evaluation Test

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

1. By adding the thin wall feature, you are allowed to scoop out the material from a model and make it hollow from inside. (T/F)
2. The **Vent** tool enables you to create a vent in an existing model by defining only its boundary. (T/F)
3. Internal threads are created in a hole 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 the joints together so that they do not fail under an increased load. (T/F)
5. The _____ features are used in the plastic components to accommodate fasteners.
6. In Solid Edge, the features are reordered using the _____.
7. _____ is a process of tapering the selected faces of a model for its easy removal from casting while manufacturing.
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 the model 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 you specified by invoking this tool.

Review Questions

Answer the following questions:

- Which of the following tools enables you to create a network of web using open entities?
(a) **Lip** (b) **Rib**
(c) **Web Network** (d) **Web**
- Which of the following tools enables you to add a taper to the selected faces of a model?
(a) **Add Draft** (b) **Taper**
(c) **Rib** (d) None
- Which of the following tools enables you to create a vent in an existing model by defining its boundary, ribs, and spars?
(a) **Add Draft** (b) **Taper**
(c) **Rib** (d) **Vent**
- Which of the following tools can be used to add a thin wall to the entire model?
(a) **Add Draft** (b) **Thin Region**
(c) **Rib** (d) **Thin Wall**
- In which of the following steps, you can place the profiles of the mounting boss on the selected plane?
(a) **Mounting Boss Step** (b) **Sketch Step**
(c) **Profile Step** (d) None
- 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 (d) None
- The **Lip** tool enables you to add a lip to the model by adding the material or by adding a groove to the model by removing the material. (T/F)
- In Solid Edge, the faces that are threaded are shaded in blue color. (T/F)
- In Solid Edge, you can add drafts using five options. (T/F)
- In the **Thin Wall** tool, you can also remove some of the faces of the model or apply different wall thicknesses to some of them. (T/F)

Exercises

Exercise 1

Create the model shown in Figure 8-82. Its dimensions and drawing views are shown in Figure 8-83. After creating the model, save it with the name and location given below:

\Solid Edge\c08\c08exr1.par

(Expected time: 30 min)

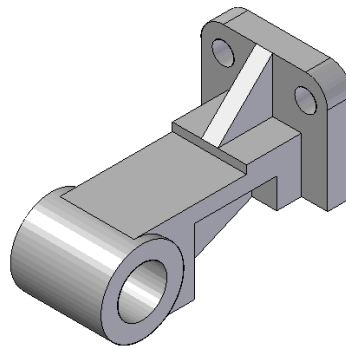


Figure 8-82 Model for Exercise 1

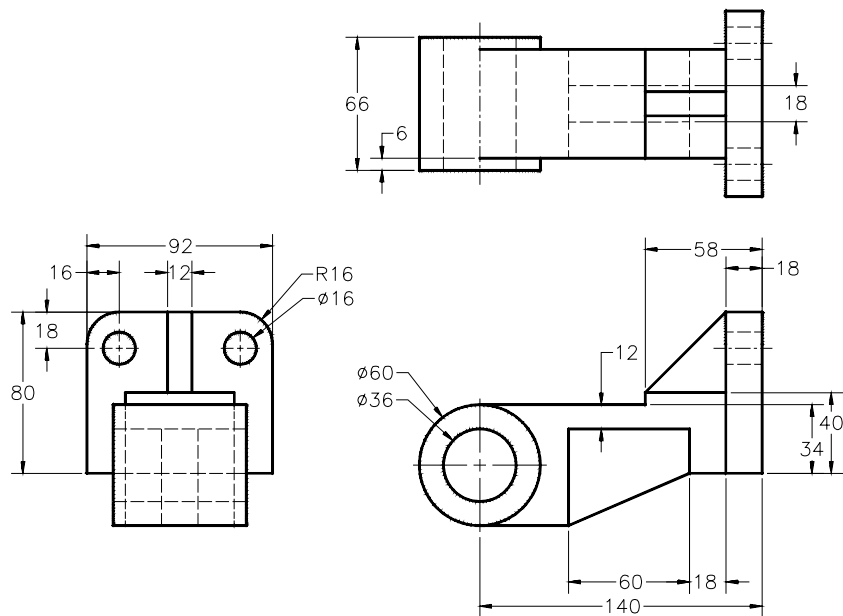
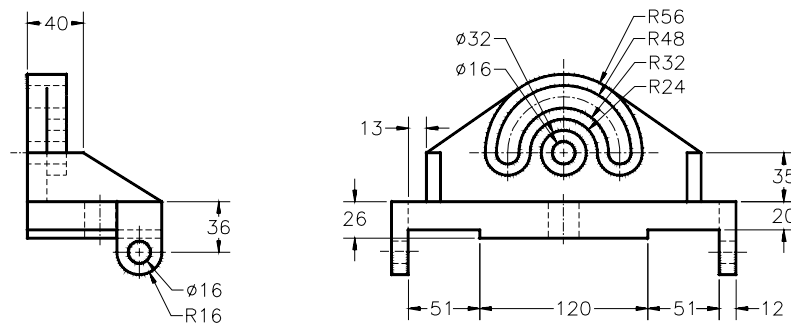


Figure 8-83 Dimensions of the model

|Solid Edge|c08|c08exr2.par

A 3D model of a mechanical part, likely a bracket or support. It features a central rectangular base with a circular hole. The base is flanked by two vertical supports. On the right support, there is a spiral or helical structure. The part is shown in a perspective view, highlighting its three-dimensional geometry.



1. T, 2. F, 3. T, 4. T, 5. mounting boss, 6. EdgeBar, 7. Adding draft, 8. three, 9. Thin Region, 10. rectangle