

Chapter 18

Sheet Metal Design

Learning Objectives

After completing this chapter, you will be able to:

- *Create base flange.*
- *Understand the FeatureManager design tree of a sheet metal component.*
- *Create edge flange.*
- *Create tabs.*
- *Create sketched bends.*
- *Create miter flange.*
- *Create closed corners.*
- *Create hems.*
- *Create jog bend.*
- *Break corners.*
- *Create cuts on the flat faces of sheet metal components.*
- *Create lofted bends.*
- *Create the flat pattern of sheet metal components.*
- *Create sheet metal components from a flat sheet.*
- *Create sheet metal components from a flat part.*
- *Create a sheet metal component by designing it as a part.*
- *Design a sheet metal part from a shelled solid model.*
- *Create cuts in a sheet metal component across the bends.*
- *Create cylindrical and conical sheet metal components.*
- *Generate the drawing views of the flat pattern of the sheet metal components.*

SHEET METAL DESIGN

In SolidWorks, you can design the sheet metal components using various tools available for manipulating the sheet metal components in the **Part** mode. Generally, the solid models of the sheet metal components are created to generate the flat pattern of the sheet, study the design of the dies and punches, and study the process plan for designing the tools needed for manufacturing the sheet metal components. In a tool room or a machine shop, the most important thing that you need before designing the press tool, bending tool or any other tool for creating a sheet metal component is the flat pattern layout of a component. Figure 16-1 shows the model of a sheet metal component and Figure 16-2 shows its flat pattern layout. A flat pattern layout displays the flattened view of the sheet metal component, refer to Figure 16-2.

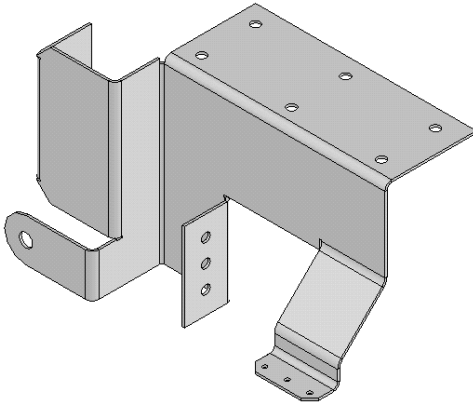


Figure 16-1 Solid model of a sheet metal component

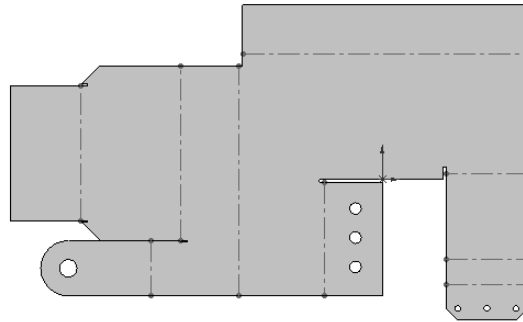


Figure 16-2 Flat pattern layout of the sheet metal component

As discussed earlier, the sheet metal components can also be created in the **Part** mode of SolidWorks. To create a sheet metal component, start a new SolidWorks document in the **Part** mode and then invoke the **Sheet Metal CommandManager**. If this **CommandManager** is not available by default, invoke this it by right clicking on the tab of a **CommandManager** and choosing **Sheet Metal** from the shortcut menu. All tools that are used to design a sheet metal component are available in this **CommandManager**. You can also invoke these tools from the **Sheet Metal** toolbar. The different methods to create sheet metal components are discussed in this chapter.

DESIGNING THE SHEET METAL COMPONENTS BY CREATING THE BASE FLANGE

The most widely used method of designing a sheet metal component is by first creating the base flange. In this method, first you will create the base flange and then add the sheet metal feature on the base flange to obtain the required sheet metal component. In this method, all the parameters related to the sheet metal such as the bending radius, the bend allowance, and the relief are defined while creating the base flange. Various tools used to create the sheet metal components are discussed next.

Creating the Base Flange

CommandManager:	Sheet Metal > Base Flange/Tab
SolidWorks menus:	Insert > Sheet Metal > Base Flange
Toolbar:	Sheet Metal > Base Flange/Tab



To create a sheet metal component, you first need to create a base feature or a base sheet. This base sheet is known as the base flange. You can create a base flange from a closed sketch or an open sketch. To create a base flange, draw the sketch of the base flange and then choose the **Base Flange/Tab** button from the **Sheet Metal CommandManager**; the **Base Flange PropertyManager** will be displayed. Also, the preview of the base flange will be displayed with the default values. Figure 16-3 shows the **Base Flange PropertyManager** for an open sketch. The rollouts in the **Base Flange PropertyManager** are discussed next.



Note

*The parameters that you define in the **Base Flange PropertyManager** are used as the default parameters throughout the current document. However, you can modify these values using the **PropertyManagers** of other tools.*

Direction 1 Rollout

This rollout is displayed only if the sketch of the base flange is open. The options in this rollout are used to define the feature termination in the first direction.

Direction 2 Rollout

The options in the **Direction 2** rollout are used to define the feature termination in the second direction. This rollout is displayed only if the sketch for the base flange is open.

Sheet Metal Gauges Rollout

This rollout enables you to use the gauge table to create the sheet metal parts. Select the **Use gauge table** check box; the **Select Table** drop-down list will be displayed. Select any of the default gauge tables from this drop-down list. You can also choose the **Browse** button and select a user-defined gauge table.

Sheet Metal Parameters Rollout

The options in the **Sheet Metal Parameters** rollout are used to define the thickness and the bend radius of the sheet. These options are discussed next.

Thickness

The **Thickness** spinner in the **Sheet Metal Parameters** rollout is used to define the thickness of the sheet.

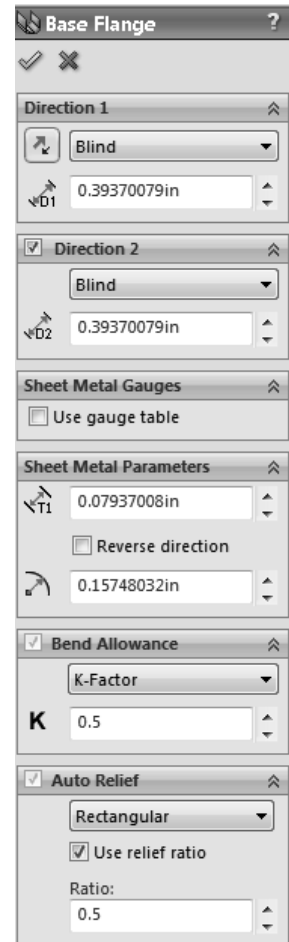


Figure 16-3 The **Base Flange PropertyManager**

Reverse direction

The **Reverse direction** check box is used to flip the direction of material addition while adding the thickness to the base flange.

Bend Radius

The **Bend Radius** spinner is used to specify the bend radius of the base flange. If the sketch of the base flange is closed, the **Bend Radius** spinner will not be available in the **Sheet Metal Parameters** rollout.

Bend Allowance Rollout

The options in the **Bend Allowance Type** drop-down list of this rollout are used to specify the bend allowance for all bends in a sheet metal component. These options are discussed next.

Bend Table

The **Bend Table** option is selected to specify the bending allowance by using the bend tables. On selecting this option, the **Bend Table** drop-down list will be displayed below the **Bend Allowance Type** drop-down list. In SolidWorks, various bend tables are provided to calculate the bending radius. The **BASE BEND TABLE** option is selected by default in the **Bend Table** drop-down list. The other bend tables available in this list are **METRIC BASE BEND TABLE**, **KFACTOR BASE BEND TABLE**, and **SAMPLE**. Choose the **Browse** button available below this drop-down list to browse the location of the folder, if you have saved a user-defined bending table file that is created in Microsoft Excel.

K-Factor

The **K-Factor** option is used to define the K-Factor. The K-Factor is defined as the ratio of the location of the neutral sheet to the thickness of the sheet. On selecting this option, the **K-Factor** spinner will be displayed. Specify the K-Factor value in this spinner.

Bend Allowance

The **Bend Allowance** option is used to define the bend allowance by specifying a numeric value. When you select this option, the **Bend Allowance** spinner will be displayed. You can specify the bend allowance value in this spinner.

Bend Deduction

The **Bend Deduction** option is used to define the bend deduction. When you select this option, the **Bend Deduction** spinner will be displayed. Specify the bend deduction value in this spinner.

Auto Relief Rollout

The **Auto Relief** rollout is used to define the relief in the sheet metal component. The reliefs are provided in the sheet metal components to avoid tearing of the sheet while bending. The options in this rollout are discussed next. You will learn about the types of reliefs in detail later in this chapter.

Auto Relief Type

The **Auto Relief Type** drop-down list is used to define the type of relief that you need to specify to the base flange. The types of reliefs available in this drop-down list are **Rectangle**,

Tear, and **Obround**. If you select the **Rectangle** or **Obround** type of relief, the **Relief Ratio** spinner will be displayed. You can use this spinner to define the relief ratio.

The base flange can be created using an open sketch with a single sketched entity, as shown in Figure 16-4. Figure 16-5 shows the resulting base flange.

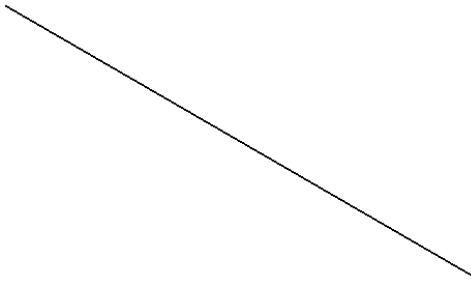


Figure 16-4 Open sketch with a single entity

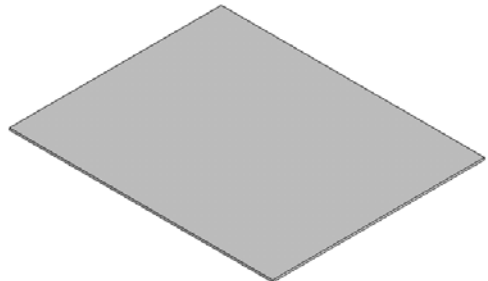


Figure 16-5 Resulting base flange

You can also create the base flange using an open sketch with multiple sketched entities, as shown in Figure 16-6. Figure 16-7 shows the resulting base flange with the bending radius applied automatically to the edges.

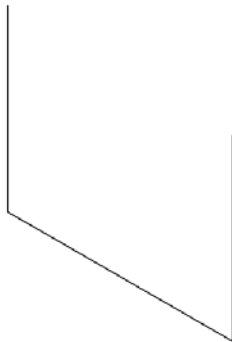


Figure 16-6 Open sketch with multiple entities

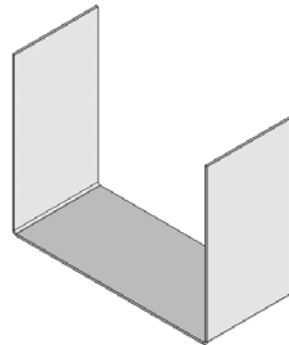


Figure 16-7 Resulting base flange

Figure 16-8 shows a closed sketch and Figure 16-9 shows the resulting base flange.

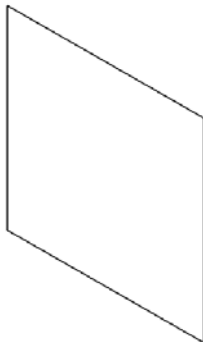


Figure 16-8 Closed sketch

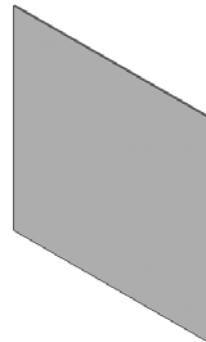


Figure 16-9 Resulting base flange

Understanding the FeatureManager design tree of a Sheet Metal Component

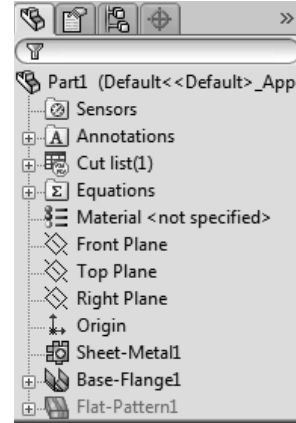
After creating the base flange, you will notice that some nodes are displayed in the **FeatureManager design tree**, as shown in Figure 16-10. The nodes are discussed next.

Cut list(1)

In SolidWorks 2010, you can create multiple sheet metal parts in single component. You can also create a sheet metal component as a combination of sheet metal parts and weldments. Therefore, when you create sheet metal parts, they are listed in the **Cut list** node. The number that is displayed shows the number of multiple bodies in the component. The **Cut list** node is similar to the **Solid Bodies** node in the **Part** environment.

Sheet-Metal1 Node

The **Sheet-Metal1** node contains all information about the sheet metal parameters such as bend, bend allowance, and auto relief parameters that are specified while creating the base flange. The values assigned to these parameters are automatically applied to all other sheet metal features that will be added to the base flange. At any stage of the design, you can edit these parameters. To do so, select **Sheet-Metal1** in the **FeatureManager design tree**; a pop-up toolbar will be displayed. Choose **Edit Feature** from the pop-up toolbar; the **Sheet-Metal1 PropertyManager** will be displayed. You can modify the default sheet metal parameters using this PropertyManager.



*Figure 16-10 Various nodes displayed in the **FeatureManager design tree** after creating the base flange*

Base-Flange1 Node

The **Base-Flange1** node is displayed after creating the base flange. You can change the thickness of the sheet by editing this feature. You can also edit the sketch of the base flange using this feature.

Flat-Pattern1 Node

The **Flat-Pattern1** node is also displayed after creating the base flange. This feature is used to create the flat pattern of the bent sheet metal component. By default, this feature is suppressed. You will learn more about the flat patterns later in this chapter.

Creating the Edge Flange

CommandManager:	Sheet Metal > Edge Flange
SolidWorks menus:	Insert > Sheet Metal > Edge Flange
Toolbar:	Sheet Metal > Edge Flange



Edge flange is a bent sheet metal wall created at an angle at the edge of an existing base flange or an existing flange. To create an edge flange, choose the **Edge Flange** button from the **Sheet Metal CommandManager**; the **Edge-Flange PropertyManager**

will be displayed, as shown in Figure 16-11 and you will be prompted to select a linear edge of a planar face to create the edge flange.

Next, you need to select the edge along which the flange will be created, as shown in Figure 16-12. As soon as you select the edge, the preview of the edge flange with the drag handle will be displayed in the drawing area, as shown in Figure 16-13. The length of the resulting flange will change dynamically as you move the cursor. Next, you need to specify the parameters of the edge flange in the **Edge-Flange PropertyManager**. The rollouts in the **Edge-Flange PropertyManager** are discussed next.

Flange Parameters Rollout

The options in the **Flange Parameters** rollout are used to define the edge reference to be used for creating the edge flange, the bending radius, and the profile of the edge flange. These options are discussed next.

Edge

The **Edge** selection box is used to select the edges to create the edge flange.

Edit Flange Profile

The **Edit Flange Profile** button is chosen to edit the profile of the edge flange. By default, the edge flange is created along the entire length of the selected edge. To edit the profile of the edge flange, choose the **Edit Flange Profile** button; the **Profile Sketch** dialog box will be displayed informing you that the sketch is valid.

Also, the sketching environment will be invoked in the background. Edit the sketch of the profile of the edge flange using the sketching tools. You will also notice that while

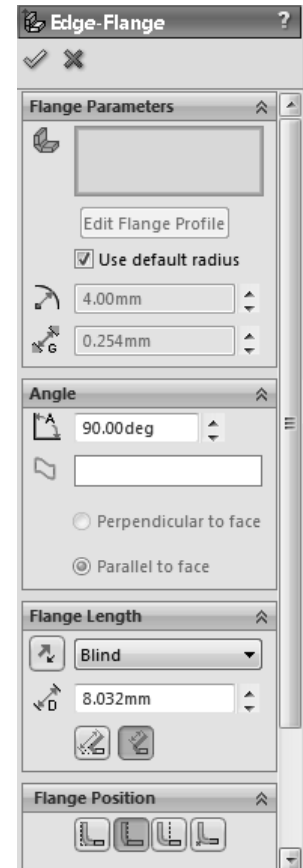


Figure 16-11 The *Edge-Flange PropertyManager*

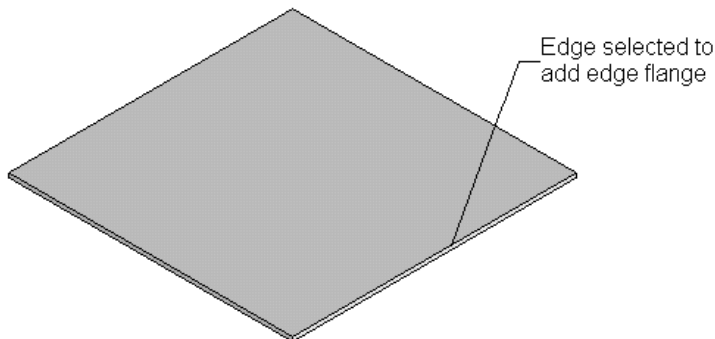


Figure 16-12 Edge selected to add the edge flange

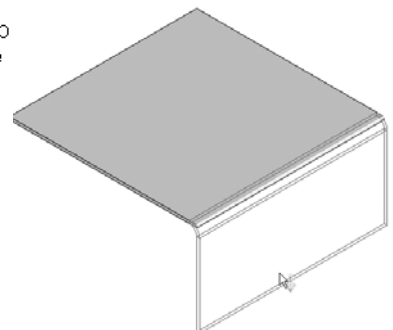


Figure 16-13 Preview of the edge flange with the drag handle

editing the sketch of the edge flange, the **Profile Sketch** dialog box informs you whether the sketch is valid for creating the edge flange or not. If the status of the sketch is shown valid in the **Profile Sketch** dialog box, the preview of the flange will be displayed in the drawing area. After editing the profile, choose the **Finish** button from the **Profile Sketch** dialog box; the flange will be created and the **Edge-Flange PropertyManager** will be automatically closed. Note that if you want to modify the other parameters of the flange, choose the **Back** button from the **Profile Sketch** dialog box. Figure 16-14 shows the edge flange created along the entire length of the selected edge. Figure 16-15 shows the edited sketch of the edge flange and Figure 16-16 shows the resulting edge flange.

Angle Rollout

The **Angle** rollout is used to define the angle of the flange. The default angle of the flange is 90-degree. You can define any other angle of the flange by using the **Flange Angle** spinner. The angle of the edge flange can be greater than 0-degree and less than 180-degree. You can also select a face and specify whether the resulting flange will be parallel or normal to it. Figure 16-17 shows an edge flange created at an angle of 45-degree. Figure 16-18 shows an edge flange created at an angle of 135-degree.

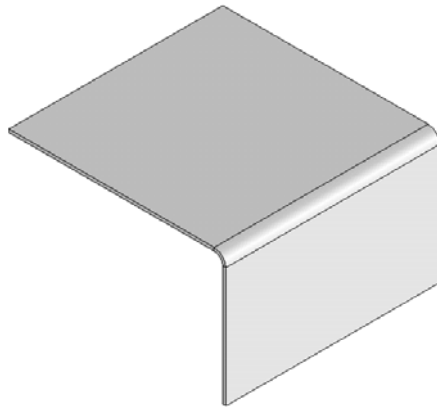


Figure 16-14 Edge flange created along the entire length of the edge

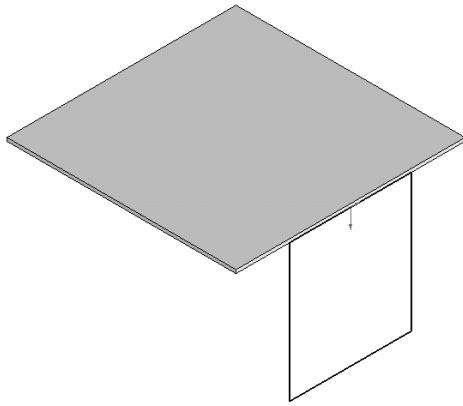


Figure 16-15 Edited sketch of the edge flange

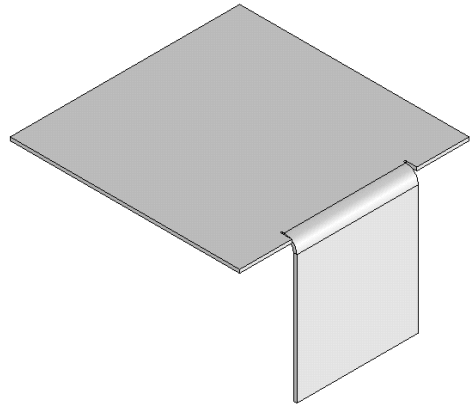


Figure 16-16 Resulting edge flange

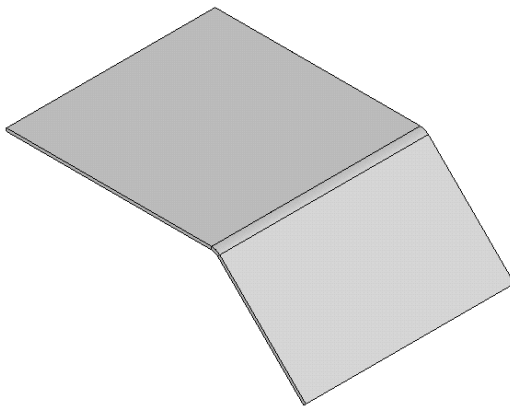


Figure 16-17 Edge flange created at an angle of 45-degree

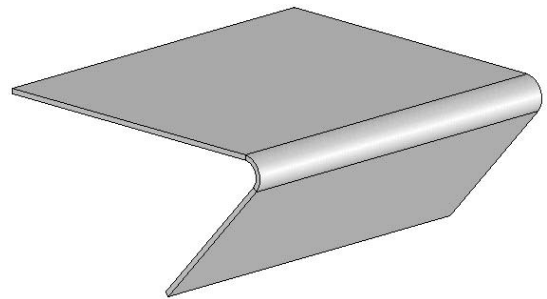


Figure 16-18 Edge flange created at an angle of 135-degree

Flange Length Rollout

The **Flange Length** rollout is used to define the length of the flange. In other words, the options for feature termination are available in this rollout. These options are the same as discussed earlier. The other two options provided in this rollout are discussed next.

Outer Virtual Sharp

The **Outer Virtual Sharp** button is used to define the length of the flange from the outer virtual sharp. The outer virtual sharp is an imaginary vertex created by extending the tangent lines virtually from the outer radius of the bend, as shown in Figure 16-19.

Inner Virtual Sharp

The **Inner Virtual Sharp** button is chosen by default and is used to define the length of the flange from the inner virtual sharp. The inner virtual sharp is an imaginary vertex created by extending the tangent lines virtually from the inner radius of the bend, as shown in Figure 16-19.

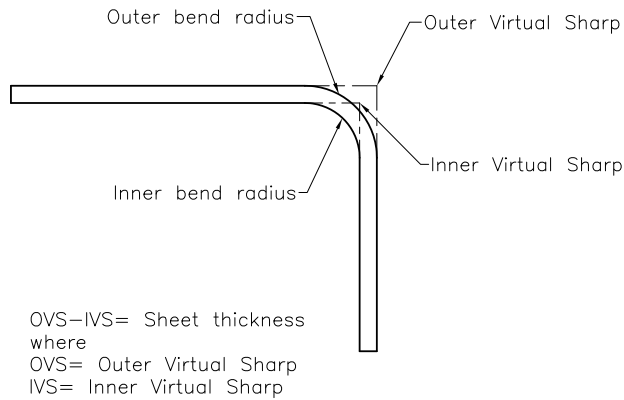


Figure 16-19 Outer Virtual Sharp and Inner Virtual Sharp

Flange Position Rollout

The **Flange Position** rollout is used to define the position of the flange on an edge. The options in this rollout are discussed next.

Material Inside

The **Material Inside** button is used to create the edge flange in such a way that the material of the flange after the bend lies inside the maximum limit of sheet. Figure 16-20 shows the edge flange created with the **Material Inside** button chosen.

Material Outside

The **Material Outside** button is chosen by default and the edge flange is created such that the material of the flange after the bend lies outside the maximum limit of the sheet. Figure 16-21 shows the edge flange created with the **Material Outside** button chosen.

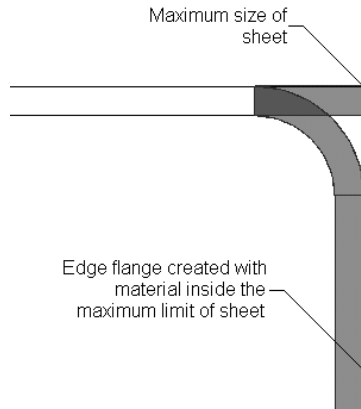


Figure 16-20 Edge flange created with the **Material Inside** button chosen

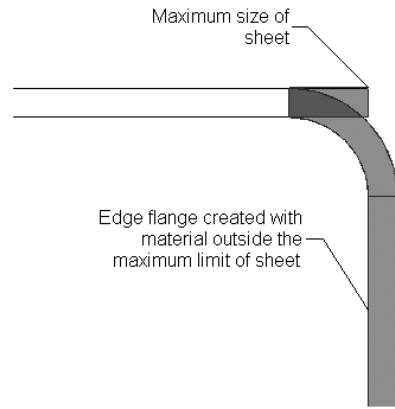


Figure 16-21 Edge flange created with the **Material Outside** button chosen

Bend Outside

The **Bend Outside** button is used to create an edge flange such that the bending of the sheet starts from outside the maximum limit of the sheet, as shown in Figure 16-22.

Bend from Virtual Sharp

The **Bend from Virtual Sharp** button is used to create an edge flange with the bending of the sheet starting from the virtual sharp. The position of the flange depends on whether you choose the **Outer Virtual Sharp** button or the **Inner Virtual Sharp** button from the **Flange Length** rollout. Figure 16-23 shows the edge flange created with the **Inner Virtual Sharp** and **Bend from Virtual Sharp** buttons chosen.

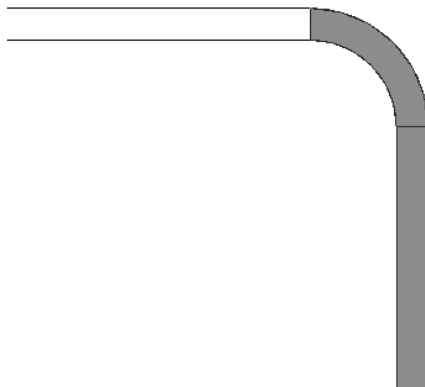


Figure 16-22 Edge flange created with the **Bend Outside** button chosen

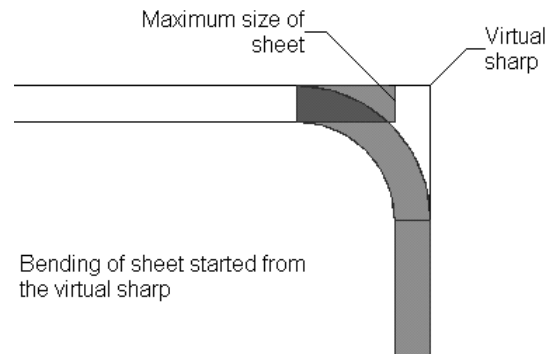


Figure 16-23 Edge flange created with the **Bend from Virtual Sharp** button chosen

Trim side bends

Select the **Trim side bends** check box to trim extra materials in the bends surrounding the current edge flange. By default, this check box is not selected. Figure 16-24 shows the

edge flange created with the **Trim side bends** check box cleared. Figure 16-25 shows the edge flange created with the **Trim side bends** check box selected.

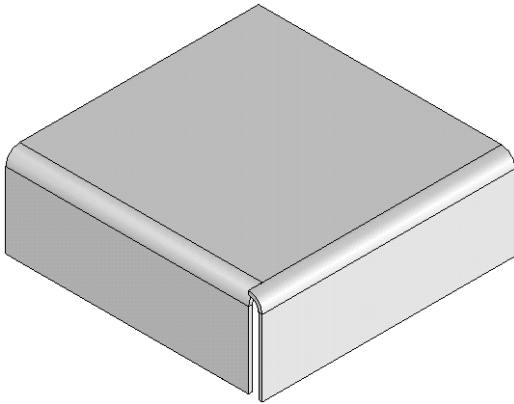


Figure 16-24 Edge flange created with the **Trim side bends** check box cleared

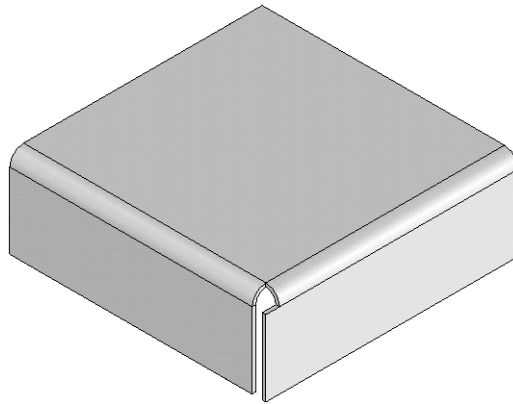


Figure 16-25 Edge flange created with the **Trim side bends** check box selected

Offset

The **Offset** check box is available only when you create an edge flange using the **Material Inside**, **Material Outside**, or **Bend Outside** options. This check box is used to create an edge flange at an offset distance from the selected edge reference. On selecting the **Offset** check box, the **Offset End Condition** drop-down list and the **Offset Distance** spinner will be displayed. Specify the offset distance using this spinner. Figure 16-26 shows the edge flange created with the **Offset** check box cleared. Figure 16-27 shows the edge flange created with the **Offset** check box selected and the offset distance specified in the **Offset Distance** spinner.

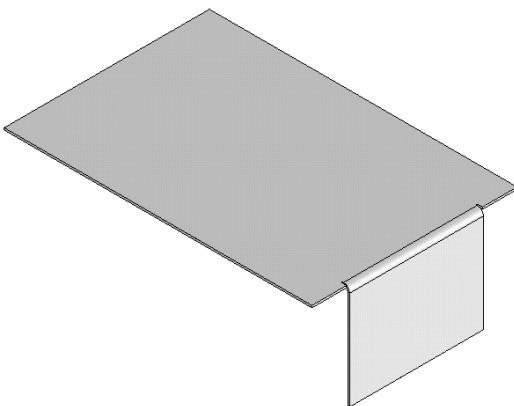


Figure 16-26 Edge flange created with the **Offset** check box cleared

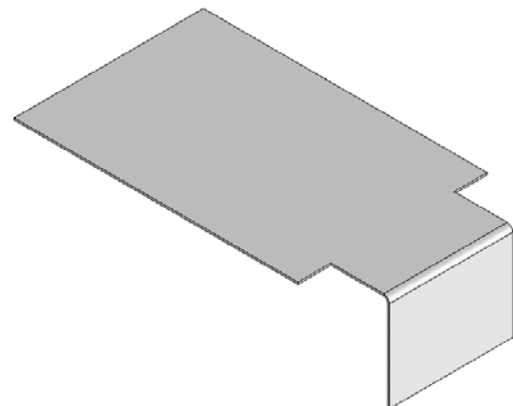


Figure 16-27 Edge flange created with the **Offset** check box selected

Custom Bend Allowance Rollout

The **Custom Bend Allowance** rollout is used to define the bend allowance other than the default bend allowance that you defined while creating the base flange. To apply the custom

bend allowance, expand this rollout by selecting the **Custom Bend Allowance** check box. Then use the options in this rollout to define the bend allowance for the current bend as discussed earlier.

Custom Relief Type Rollout

The **Custom Relief Type** rollout is used to define the type of relief other than the default relief type that you defined while creating the base flange. To apply the custom relief, expand this rollout by selecting the check box in the title bar of the **Custom Relief Type** rollout, as shown in Figure 16-28.

The types of reliefs that can be defined for a sheet metal component are discussed next.

Obround Relief

The **Obround** option is used to provide the obround relief such that the edges of the relief merging with the sheet are rounded. The **Use relief ratio** check box is selected by default. Therefore, you can modify the value of the relief ratio, by setting the value in the **Relief Ratio** spinner. If you clear the **Use relief ratio** check box, the **Relief Width** and **Relief Depth** spinners will be displayed, as shown in Figure 16-29. You can modify the relief width and relief depth individually by using these two spinners.

Figure 16-30 shows the edge flange created by providing the obround relief with the default relief ratio. Figure 16-31 shows the edge flange created by providing obround relief after modifying the relief ratio.

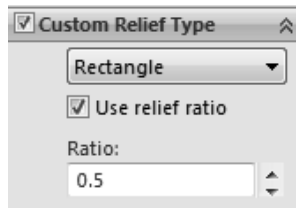


Figure 16-28 The **Custom Relief Type** rollout

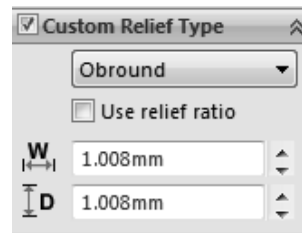


Figure 16-29 The **Relief Width** and **Relief Depth** spinners displayed in the **Custom Relief Type** rollout

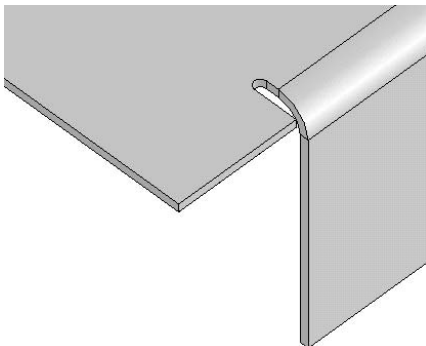


Figure 16-30 Edge flange created with the default relief ratio

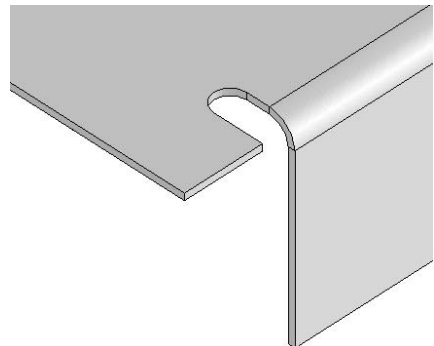


Figure 16-31 Edge flange created after modifying the relief ratio

Rectangle Relief

The **Rectangle** option is selected, by default. This option specifies the rectangular relief to the sheet metal components. The options for defining the rectangular relief are the same as discussed above. Figure 16-32 shows an edge flange created by providing the rectangular relief with the default relief ratio. Figure 16-33 shows an edge flange created by providing rectangular relief after modifying the relief ratio.

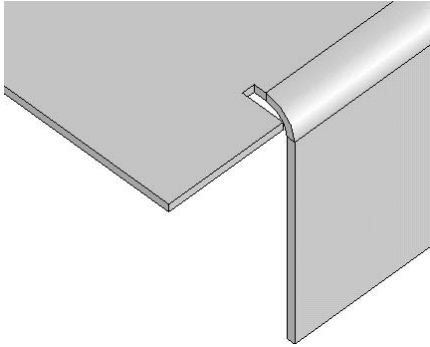


Figure 16-32 Edge flange created by providing the rectangular relief with default relief ratio

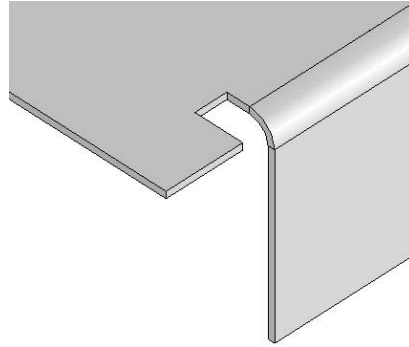


Figure 16-33 Edge flange created by providing the rectangular relief after modifying the relief ratio

Tear Relief

You can provide the tear relief to an edge flange by using the **Tear** option. The tear relief will tear the sheet in order to accommodate the bending of the sheet. When you select the **Tear** option from the **Relief Type** drop-down list, all the other options are replaced by the **Rip** and **Extend** buttons, as shown in Figure 16-34.

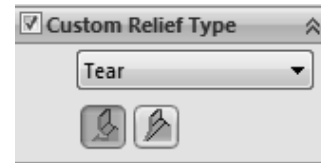


Figure 16-34 The **Custom Relief Type** rollout with the **Tear** option selected from the **Relief Type** drop-down list

The **Rip** button is chosen by default. This option rips or tears the sheet to accommodate the bending of the sheet, as shown in Figure 16-35. When the **Extend** button is chosen, the outer faces of the bend will be extended to the outer faces of the sheet on which you create the edge flange, as shown in Figure 16-36.

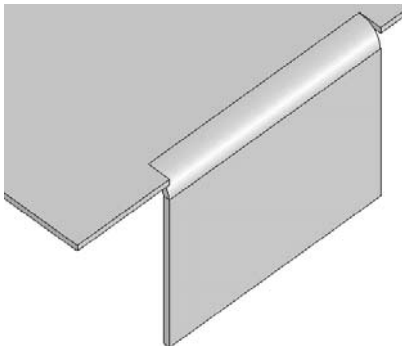


Figure 16-35 Tear relief with the **Rip** button chosen

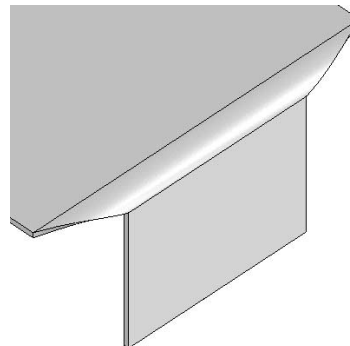


Figure 16-36 Tear relief with the **Extend** button chosen



Tip. You can edit the sketch of an edge flange after creating it. To do so, select the edge flange feature from the **FeatureManager design tree** and invoke the pop-up toolbar. Next, choose **Edit Sketch** from it; the sketching environment will be invoked. Now, edit the sketch and exit the sketching environment.

Creating Tabs

CommandManager: Sheet Metal > Base Flange/Tab
Toolbar: Sheet Metal > Base Flange/Tab



A tab feature is created by adding material to the walls of the sheet metal component. To create a tab, select a face to use as the sketching plane and create the sketch of the tab. Remember that the sketch must be closed. Now, choose the **Base Flange/Tab** button from the **Sheet Metal CommandManager**; a tab will be created and the thickness of the tab will be automatically adjusted according to the thickness of the sheet. In SolidWorks 2010, you can create a new tab added as a separate body. To do so, clear the **Merge result** check box in the **Sheet Metal Parameters** rollout. When a new tab is created by clearing the **Merge result** check box, you can notice a new node, **Sheet-Metal(2)**, is added to the **FeatureManager design tree** and the name of the new tab is added below it. Also, the name of the new tab will be listed in the **Cut list** node. Figure 16-37 shows the sketch for creating a tab and Figure 16-38 shows the resulting tab.

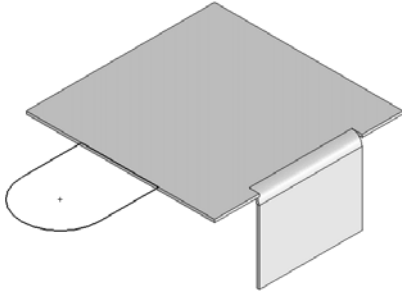


Figure 16-37 Sketch for creating a tab

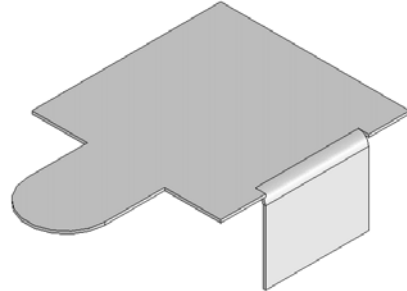


Figure 16-38 Resulting tab

**Note**

If you create a tab feature and select it from the **FeatureManager design tree**, the **Edit Feature** option will not be available in the pop-up toolbar. So, you cannot edit a tab feature. However, you can edit the sketch of the tab feature.

Creating the Sketched Bend

CommandManager: Sheet Metal > Sketched Bend
SolidWorks menus: Insert > Sheet Metal > Sketched Bend
Toolbar: Sheet Metal > Sketched Bend



The **Sketched Bend** tool is used to create a bend by using a sketch as the bending line. To create a sketched bend, select the face of the sheet on which you need to create a bend line and invoke the sketching environment. Draw a line to define the bend line by using the **Line** tool. Now, choose the **Sketched Bend** button from the **Sheet Metal CommandManager**; the **Sketched Bend PropertyManager** will be displayed, as shown in Figure 16-39. Also, you will be prompted to specify the planar face to be fixed while creating the bend. Select the side of the sheet that will be fixed when you create the bend; a black sphere will be displayed on the selected point and the **Reverse Direction** arrow will also be displayed to reverse the direction of the bend creation. You will notice that the **Bend Centerline** button is chosen by default in the **Bend position** area. Therefore, the sheet is bent equally on both sides of the bend line. The other options in the **Sketched Bend PropertyManager** are the same as those discussed earlier. After setting all parameters, choose the **OK** button from the **Sketched Bend PropertyManager**. Figure 16-40 shows the sketch to be used as the bending line and the side of the face to be fixed while bending. Figure 16-41 shows the resulting bend.

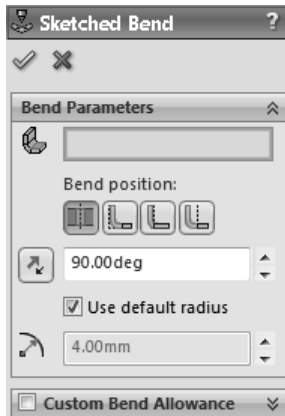


Figure 16-39 The **Sketched Bend PropertyManager**

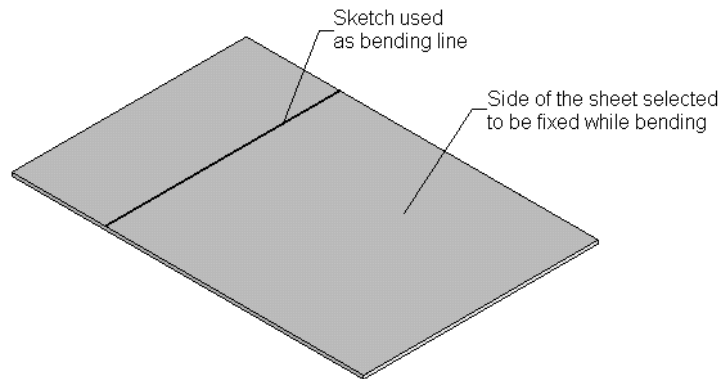


Figure 16-40 The face to be fixed and the bending line

**Note**

You can also create more than one sketch line for creating multiple bends by using a single sketch bend feature. But, make sure that the bend sketches do not intersect each other. Figure 16-42 shows the sheet and the two bend lines and Figure 16-43 shows the resulting bends.

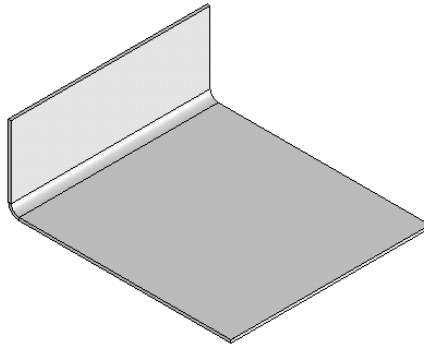


Figure 16-41 Resulting sketched bend

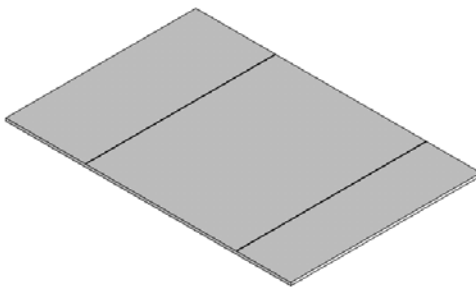


Figure 16-42 Two bend lines

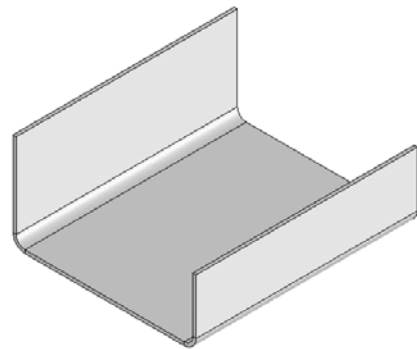


Figure 16-43 Resulting sketched bend

Creating the Miter Flange

CommandManager:	Sheet Metal > Miter Flange
SolidWorks menus:	Insert > Sheet Metal > Miter Flange
Toolbar:	Sheet Metal > Miter Flange



The **Miter Flange** tool is used to create a series of flanges along the edges of a sheet metal component. The profile of the miter flange is defined by the sketch created on a sketching plane normal to the direction of extrusion of the flange. To create a miter flange, select the sketching plane and invoke the sketching environment. Create a sketch for the miter flange and then choose the **Miter Flange** button from the **Sheet Metal** toolbar; the **Miter Flange PropertyManager** will be displayed, as shown in Figure 16-44.

The preview of the flange with the default settings will be displayed in the drawing area and you will be prompted to select the linear edge(s) to attach the miter flange. Figure 16-45 shows the sketch for creating the miter flange and Figure 16-46 shows the preview of the miter flange. You can select the other continuous edges, if required, as shown in Figure 16-47. After selecting the edges, choose the **OK** button from the **Miter Flange PropertyManager**; the miter flange is created, as shown in Figure 16-48.

The **Gap distance** area in the **Miter Flange PropertyManager** is discussed next.

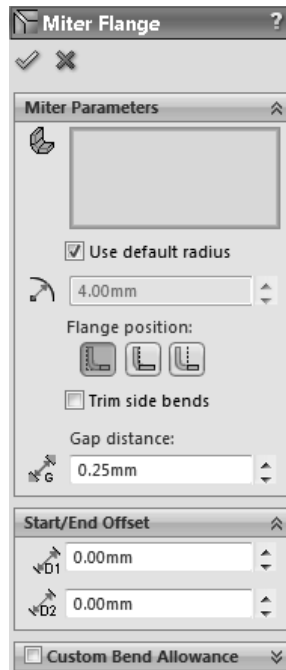


Figure 16-44 The Miter Flange PropertyManager

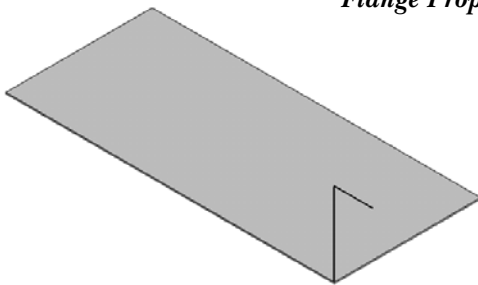


Figure 16-45 Sketch for creating the miter flange

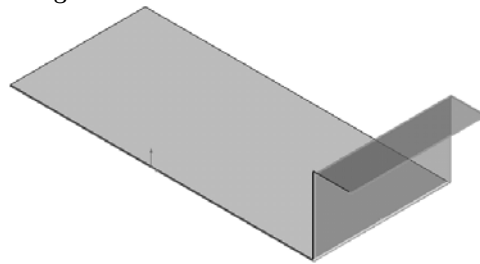


Figure 16-46 Preview of the miter flange

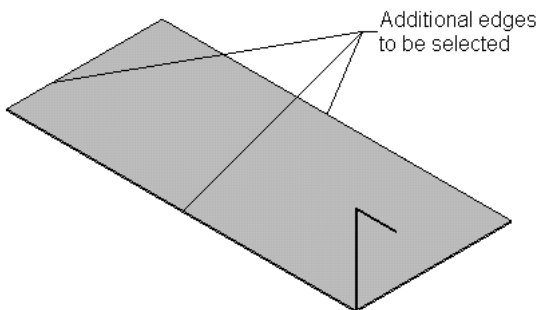


Figure 16-47 Sketch and the additional edges to be selected

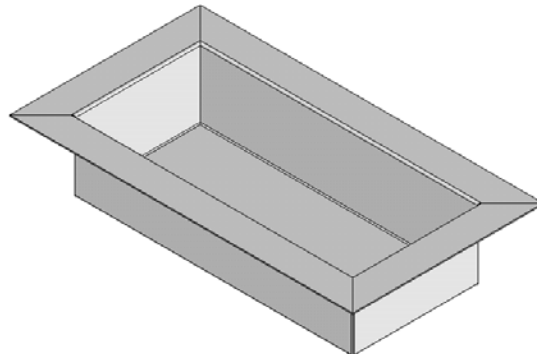


Figure 16-48 Resulting miter flange

Gap distance Area

The spinner in this area is used to define the rip distance between two consecutive flanges. Set the value in the **Rip Gap** spinner to modify the distance value of the rip. While creating a miter flange, if the feature creation is aborted due to default rip distance, the **Rebuild Errors** message box will be displayed and it will prompt you to enter a larger distance value. Figure 16-49 shows the miter flange created using the default distance value and Figure 16-50 shows the miter flange created using the modified rip distance.

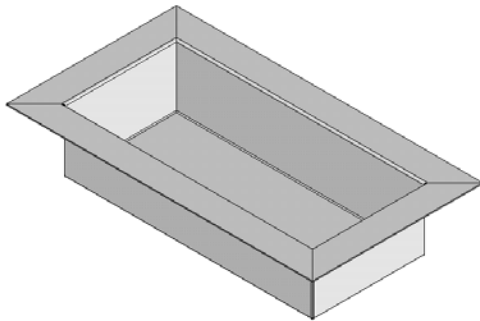


Figure 16-49 Miter flange with the default rip distance

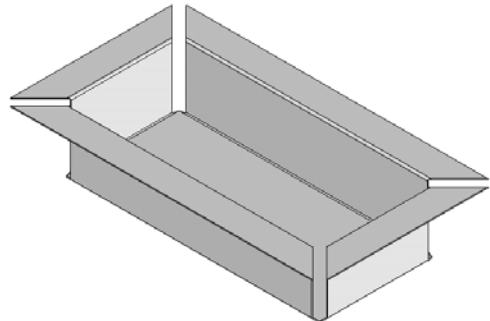


Figure 16-50 Miter flange with the modified rip distance

Start/End Offset Rollout

You can specify the start and end offset distances of the miter flange by using the options in the **Start/End Offset** rollout. The **Start Offset Distance** spinner is used to specify the offset distance from the start face of the miter flange. The **End Offset Distance** spinner is used to specify the offset distance from the end face of the miter flange. If the start and end offset distances are applied to the miter flange created on the continuous edges of the base flange, the start offset distance will be applied to the first edge and the end offset distance will be applied to the edge selected at last. Figure 16-51 shows the miter flange created on a single edge with the start and end offsets. Figure 16-52 shows the offsets applied to the miter flange created by selecting all the edges of the base flange.

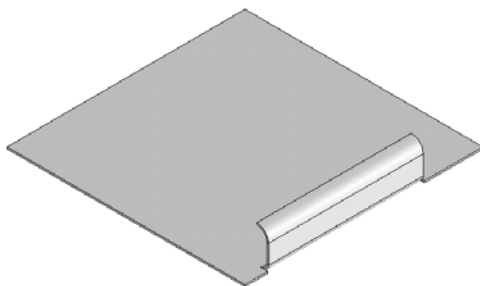


Figure 16-51 Offset distances applied to the miter flange created by selecting a single edge

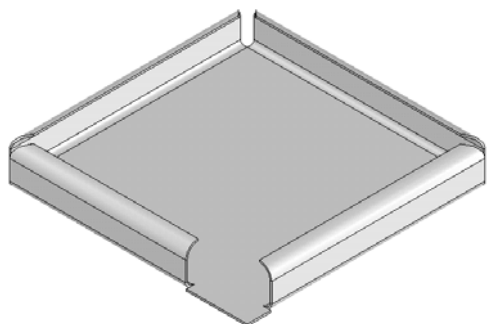



Figure 16-52 Offset distances applied to the miter flange created by selecting all edges



Tip. While creating a miter flange, if an edge is tangent to the selected edge, then  symbol will be displayed. Click on this symbol; the edges that are tangent to the selected edge will be automatically selected.

Creating Closed Corners

CommandManager: Sheet Metal > Corner > Closed Corner
SolidWorks menus: Insert > Sheet Metal > Closed Corner
Toolbar: Sheet Metal > Corner > Closed Corner



In SolidWorks, when you create walls using the **Edge Flange** tool, there may be a gap between the corners due to relief. You can close this gap and create a closed corner. To do so, choose **Corners > Closed Corner** from the **Sheet Metal CommandManager**; the **Closed Corner PropertyManager** will be displayed, as shown in Figure 16-53, and you will be prompted to select the planar corner face(s) to extend for creating a closed corner. Select the face or the edge that you need to extend, as shown in Figure 16-54; the selected face will be highlighted in green. Note that both the flanges must be normal to each other for creating the closed corners. You cannot close the faces of the flanges, if any one of them is not at 90-degree. After selecting the faces, select the type of corner that you need to create by choosing the buttons in the **Corner type** area of the **Faces to Extend** rollout; the preview of the closed corner will be displayed in the drawing area. Choose the **OK** button from the **Closed Corner PropertyManager**. Figures 16-55 through 16-57 show the preview of the closed corners created using the **Butt**, **Overlap**, and **Underlap** options.

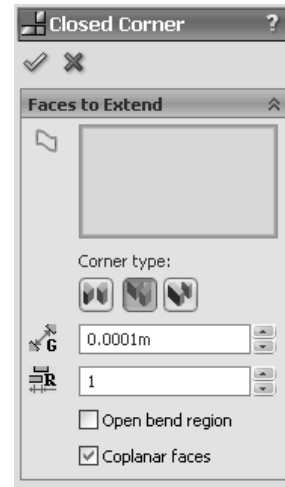


Figure 16-53 The Closed Corner PropertyManager

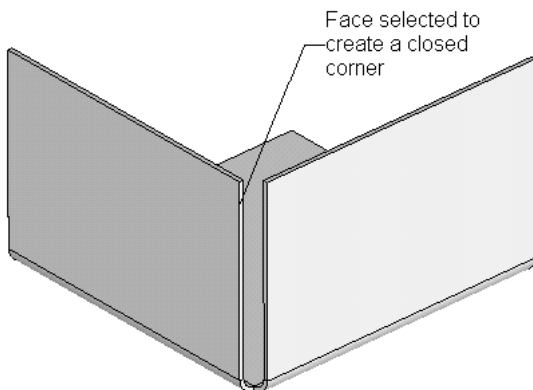


Figure 16-54 Face selected to create a closed corner

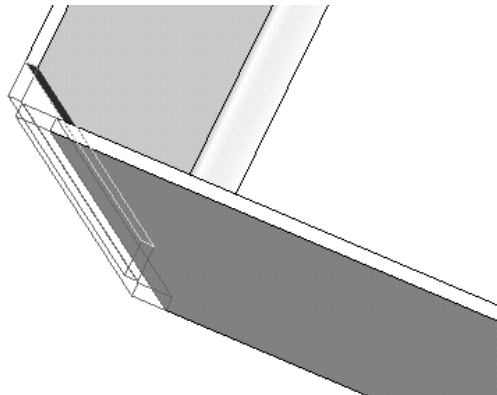


Figure 16-55 Closed corner created with the **Butt** button chosen

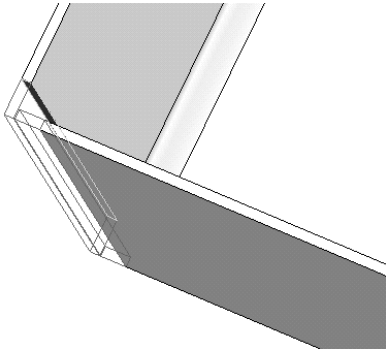


Figure 16-56 Closed corner created with the **Overlap** button chosen

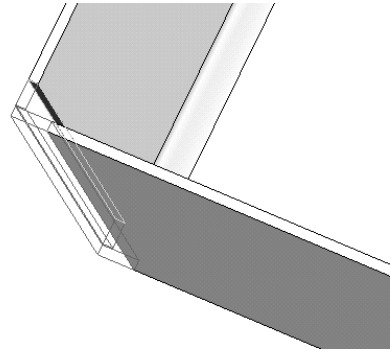


Figure 16-57 Closed corner created with the **Underlap** button chosen

Creating Hems

CommandManager:	Sheet Metal > Hem
SolidWorks menus:	Insert > Sheet Metal > Hem
Toolbar:	Sheet Metal > Hem



Hems are generally used to bend a small area of sheet in order to eliminate the sharp edges in a sheet metal component. Hems are also used to join two sheet metal components. To create a hem, choose the **Hem** button from the **Sheet Metal CommandManager**; the **Hem PropertyManager** will be displayed, as shown in Figure 16-58.

You will be prompted to select an edge on a planar face to create a hem feature. Select the edge on a planar face; the preview of the hem with the default settings will be displayed in the drawing area. The rollouts in the **Hem PropertyManager** are discussed next.

Edges Rollout

The options in the **Edges** rollout are used to specify the edges on which hem will be created. As you select the edges, the names of the edges will be listed in the **Edges** selection box and the preview of the hem will be displayed. The **Reverse Direction** button is chosen to reverse the direction of the hem.

By default, the **Material Inside** button is chosen in the **Edges** rollout. So, the hem is created such that the material of the hem after the bend lies inside the maximum limit of sheet. If you choose the **Bend Outside** button, then the hem will be created with the bend starting from the maximum limit of the sheet.

Type and Size Rollout

The **Type and Size** rollout is used to define the type and size of the hem. The types of hem that you can create using the options in this rollout are discussed next.

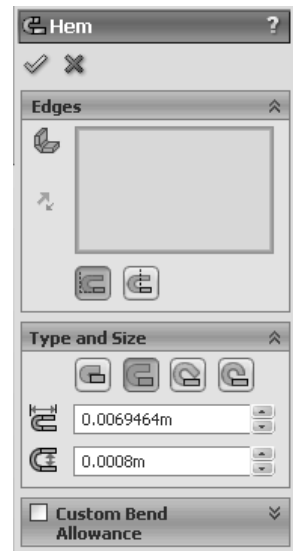


Figure 16-58 The **Hem PropertyManager**

Closed Hem

The closed hem is a hem that has no gap between the inner face of the hem and the face adjacent to the edge on which the hem is created. To create a closed hem, choose the **Closed** button and set the length of the closed hem by using the **Length** spinner. If you select more than one edge to create the hem, the **Miter Gap** rollout will be displayed. You can specify the rip gap in this rollout. Figure 16-59 shows the edge selected to create a closed hem and Figure 16-60 shows the resulting closed hem.

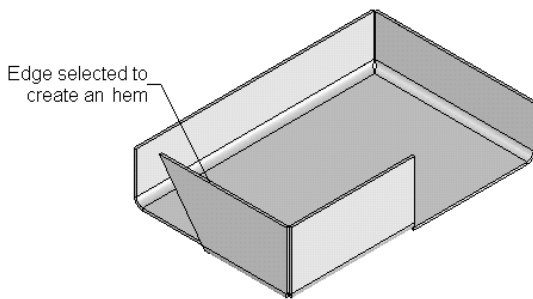


Figure 16-59 Edge selected to create a closed hem

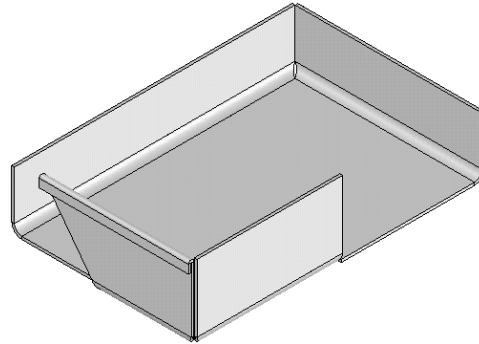


Figure 16-60 Resulting closed hem

Open Hem

The open hem is a hem with a gap between the inner face of the hem and the face adjacent to the edge on which the hem is created. To create an open hem, choose the **Open** button; the **Length** and **Gap Distance** spinners will be displayed. You can specify the value of the length and the gap distance in these spinners. Figure 16-61 shows the edge to be selected to create an open hem and Figure 16-62 shows the resulting open hem.

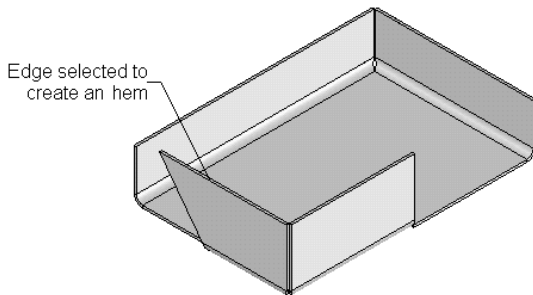


Figure 16-61 Edge selected to create an open hem

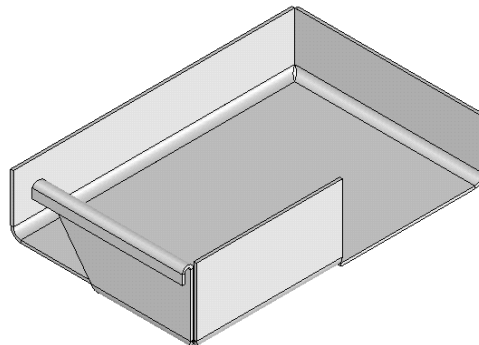


Figure 16-62 Resulting open hem

Tear Drop Hem

By default, the **Tear Drop** button is chosen when you invoke the **Hem PropertyManager**. So, a tear drop shaped hem will be created on the selected edge. Set the angle and the radius of the tear drop in the **Angle** and **Radius** spinners, respectively. Figure 16-63 shows a tear drop hem created on a sheet metal component.

Rolled Hem

The **Rolled** button is used to create the rolled shaped hem. When you chosen this button, the **Angle** and **Radius** spinners are displayed to set the respective values.

Figure 16-64 shows the rolled hem created on a sheet metal component.

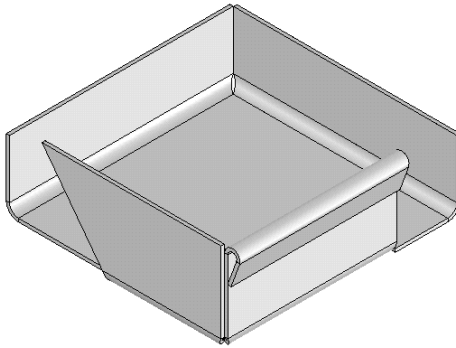


Figure 16-63 Tear drop hem

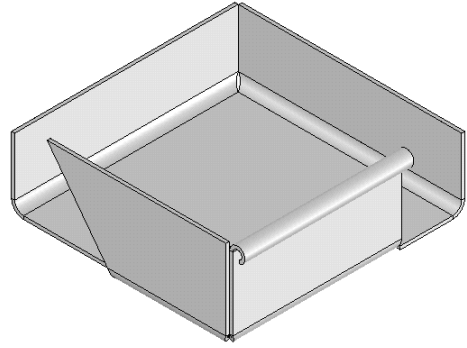


Figure 16-64 Rolled hem

Creating the Jog Bend

CommandManager: Sheet Metal > Jog
SolidWorks menus: Insert > Sheet Metal > Jog
Toolbar: Sheet Metal > Jog



In SolidWorks you can create two bends using a bend line. The bend line is sketched on the face of the sheet metal component on which you need to create the jog bend. The first bend will be in the same plane as that of the bend line and the second bend will be at an offset distance. Note that the sketch of the bend line must lie inside the face of the sheet metal. After creating the bend line, choose the **Jog** button from the **Sheet Metal CommandManager**; the **Jog PropertyManager** will be displayed, as shown in Figure 16-65 and you will be prompted to select the planar face to be fixed while creating the bend. Select the side of the face to be fixed; the preview of the jog bend with the default values will be displayed in the drawing area. Figures 16-66 and 16-68 show the bend line and the faces to be fixed. Figures 16-67 and 16-69 show the respective jog bends. The rollouts in the **Jog PropertyManager** are discussed next.

Selections Rollout

The options in the **Selections** rollout are used to define the face to be fixed while bending and to define the radius of the bend. The name of the selected face that needs to be fixed while bending will be displayed in the **Fixed Face** selection box. By default, the **Use default radius** check box will be selected. If you need to define the bending radius other than the default radius, clear this check box and set the value of the radius in the **Bend Radius** spinner.

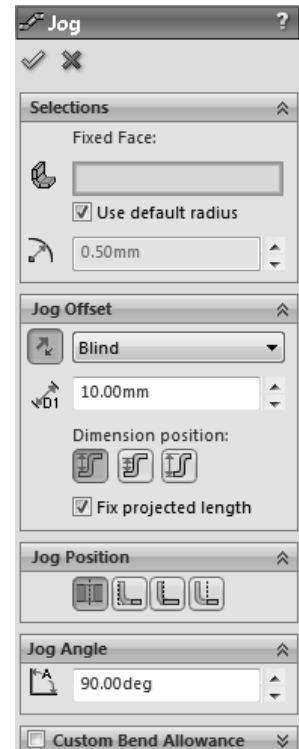


Figure 16-65 The Jog PropertyManager

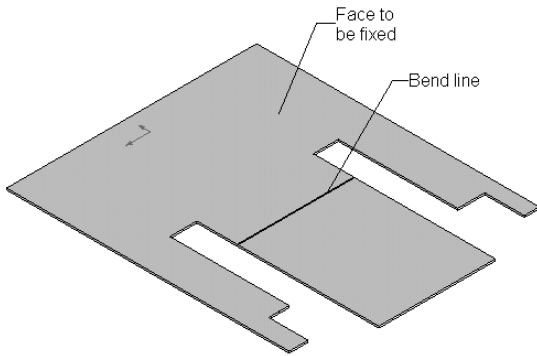


Figure 16-66 Bend line and face to fix

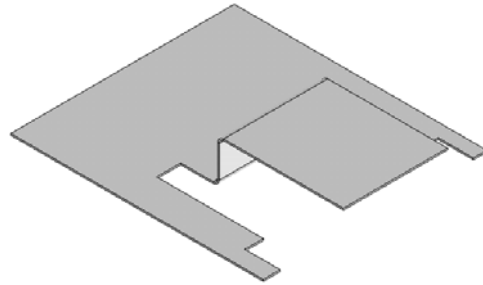


Figure 16-67 Resulting jog bend

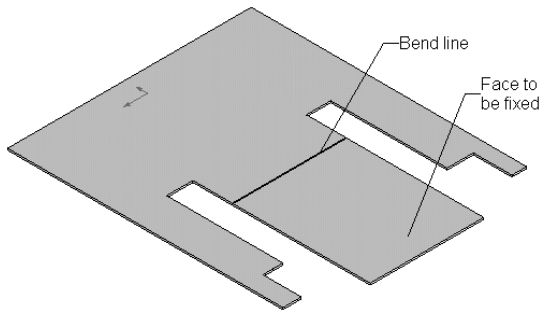


Figure 16-68 Bend line and face to fix

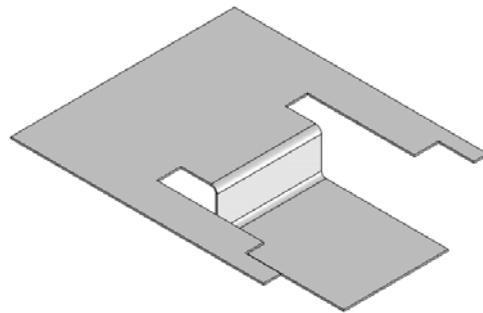


Figure 16-69 Resulting jog bend

Jog Offset Rollout

The **Jog Offset** rollout is used to define various parameters of the jog. You can define the feature termination option for the jog by using this rollout. You can also create a jog with 0 mm blind depth. The options in this rollout are discussed next.

Dimension position Area

The **Dimension position** area of the **Jog Offset** rollout is used to define the position from where the dimension of the jog offset will be calculated. The buttons in this area are used to specify the offset by calculating the inside offset, outside offset, and overall dimension, refer to Figure 16-70.

Fix projected length

The **Fix projected length** check box is selected by default and is used to maintain the length of the bent sheet equal to the projected length of the original sheet after adding a jog bend. If you clear this check box, the overall length of the sheet will be maintained equal to the original sheet even after adding the jog bend. Figure 16-71 shows the bend line that will be used to create a jog bend. Figure 16-72 shows the preview of the jog bend created with **Fix projected length** check box selected. Figure 16-73 shows the preview of the jog bend created with the **Fix projected length** check box cleared.

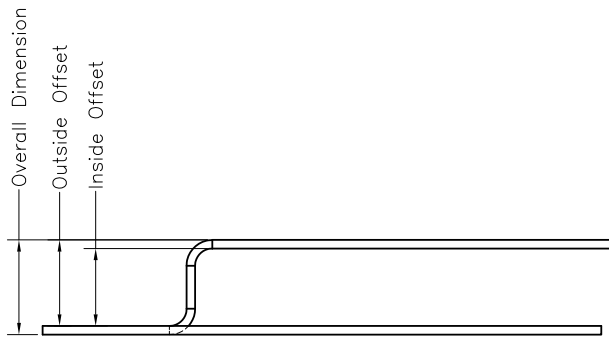


Figure 16-70 Inside offset, outside offset, and overall dimension

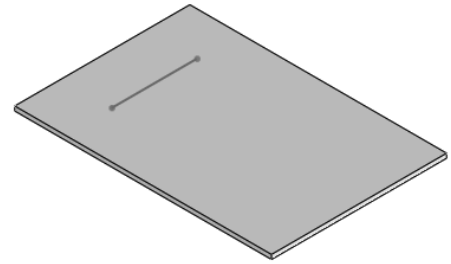


Figure 16-71 Bend line to create the jog bend



Figure 16-72 Jog bend created with the **Fixed projected length** check box selected



Figure 16-73 Jog bend created with the **Fixed projected length** check box cleared

Jog Position Rollout

The **Jog Position** rollout is used to define the position of bending. The options in this rollout are similar to those discussed earlier.

Jog Angle Rollout

The **Jog Angle** rollout is used to define the angle of the jog bend. The default value of the jog angle is 90-degree. You can set the value of the angle at which you need to create the jog bend in the **Jog Angle** spinner. Figure 16-74 shows the jog bend created at an angle of 135-degree.

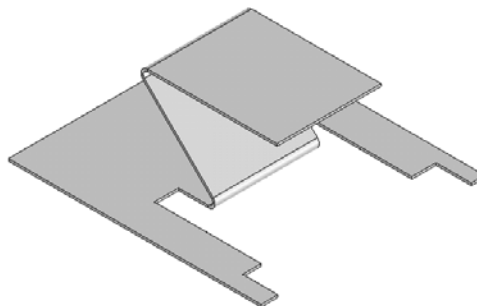


Figure 16-74 Jog bend created at an angle of 135-degree

Breaking the Corners

CommandManager	Sheet Metal > Corners > Break-Corner/Corner-Trim
SolidWorks menus:	Insert > Sheet Metal > Break-Corner
Toolbar:	Sheet Metal > Corners > Break-Corner/Corner-Trim



In SolidWorks, you are provided with an option to break the edges of the sheet metal components to create chamfer or fillet. The edges of the sheet metal components are chamfered or filleted using the **Break-Corner/Corner-Trim** tool. To invoke this tool, choose **Corners > Break-Corner/Corner-Trim** from the **Sheet Metal CommandManager**; the **Break Corner PropertyManager** will be displayed, as shown in Figure 16-75. Also, you will be prompted to select corner edge(s) or flange face(s).

Select the faces or the edges that you need to break; the preview of the corner break is displayed in the drawing area with the default settings. As the **Chamfer** button is chosen by default in the **Break Corner Options** rollout, the corner break created by default is a chamfer. You can set the value of the chamfer by using the **Distance** spinner. If you need to create a corner break as fillet, choose the **Fillet** button in the **Break Corner Options** rollout; the **Distance** spinner will be replaced by the **Radius** spinner. After setting all parameters, choose the **OK** button from the **Break Corner PropertyManager**. Figure 16-76 shows the sheet metal component with the chamfers and fillets added using the **Break Corner PropertyManager**.

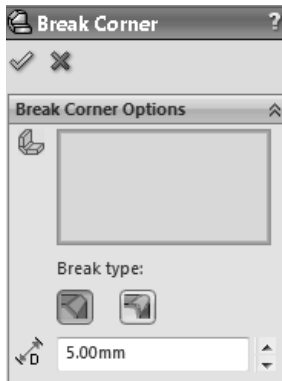


Figure 16-75 The **Break Corner PropertyManager**

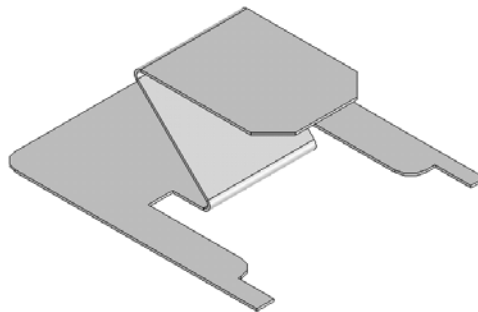


Figure 16-76 Chamfers and fillets added to the sheet metal component



Note

When you invoke the **Corner** tool after creating a flat pattern of the sheet metal component, some more options will be displayed in the **Break Corner PropertyManager**. These options are discussed later in this chapter.

Creating Cuts on the Planar Faces of the Sheet Metal Components

CommandManager:	Sheet Metal > Extruded Cut
SolidWorks menus:	Insert > Cut > Extrude
Toolbar:	Sheet Metal > Extruded Cut



Creating cuts in the sheet metal components is similar to creating cuts in the solid models. To create cuts on the planar faces of the sheet metal components, select a face or a plane as the sketching plane and invoke the sketching environment. Draw a sketch for creating the cut feature and choose the **Extruded Cut** button from the **Sheet Metal CommandManager**; the **Extrude PropertyManager** will be displayed. Set the options for feature termination in the **Extrude PropertyManager**. You will observe that some additional options are displayed in the **Distance 1** rollout of the **Extrude PropertyManager**. These additional options are discussed next.

Link to thickness

The **Link to thickness** check box is used to set the value of the feature termination according to the thickness of the sheet. On selecting this check box, the cut feature will be terminated at the blind distance equal to the sheet thickness, irrespective of the feature termination option selected.

Flip side to cut

This check box is used to reverse the side of the sheet metal part to cut.

Normal cut

The **Normal cut** check box is selected by default and is used for the bent sheet metal components. When the profile of the cut feature is on different plane and if this check box is selected, the cut feature will be created normal to the sheet thickness. However, if you clear this check box, the cut feature will be created normal to the sketching plane. Figure 16-77 shows the side view of a sheet metal component in which a cut feature is created with the **Normal cut** check box selected and a cut feature created with the **Normal cut** check box cleared.

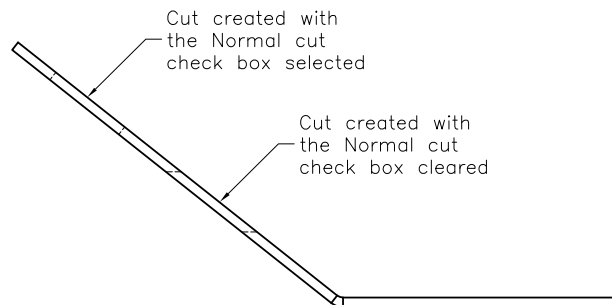


Figure 16-77 Cut created with the **Normal cut** check box selected and cleared

Creating Lofted Bends

CommandManager: Sheet Metal > Lofted-Bend
SolidWorks menus: Insert > Sheet Metal > Lofted-Bend
Toolbar: Sheet Metal > Lofted-Bend



The lofted bends are created by defining a transition of sheet between two open sections placed apart at some offset distance. To create the lofted bends, create the open sections. Remember that the sections should not have vertices. If the sections have vertices, replace them with fillets. Now, choose the **Lofted-Bend** button from the **Sheet Metal CommandManager**; the **Lofted Bends PropertyManager** will be displayed and you are prompted to select two profiles. Select two profiles to create the lofted bends; the preview of the lofted bend will be displayed in the drawing area. Set the thickness of the sheet using the **Thickness** spinner and choose the **OK** button from the **Lofted Bends PropertyManager**. Figure 16-78 shows the open sections that you need to select for creating a lofted bend. Figure 16-79 shows the resulting lofted bend.

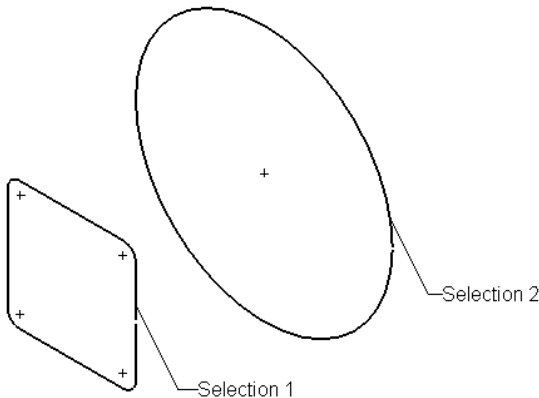


Figure 16-78 Sections for creating lofted bend

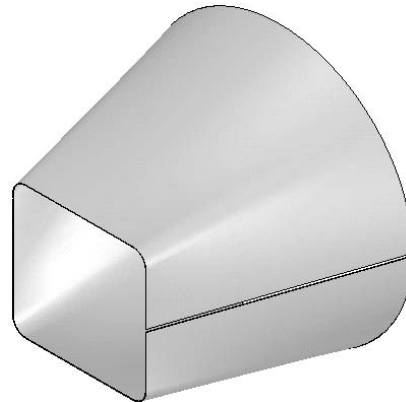


Figure 16-79 Resulting lofted bend

Creating a Flat Pattern View of the Sheet Metal Components

CommandManager: Sheet Metal > Flatten
SolidWorks menus: Insert > Sheet Metal > Flatten
Toolbar: Sheet Metal > Flatten



The flat pattern view of a sheet metal component is extensively used in the tool room or the machine shop to define the size of the raw sheet, and also the shape of the sheet that you need before bending. It is also used for process planning to start the manufacturing of the tool that will create the sheet metal component. Before creating the flat pattern, you can set the option for the flat pattern. To set the options for the flat pattern, select the **Flat-Pattern1** feature from the **FeatureManager design tree** and invoke the pop-up toolbar. Next, choose the **Edit Feature** option from this toolbar; the **Flat-Pattern PropertyManager** will be invoked, as shown in Figure 16-80.

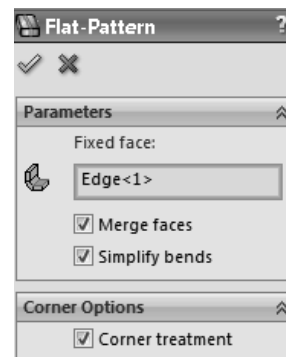


Figure 16-80 The Flat-Pattern PropertyManager

The rollouts in the **Flat-Pattern PropertyManager** are discussed next.

Parameters Rollout

The options in the **Parameters** rollout are used to define various parameters to create the flat pattern of the sheet metal component. The options in this rollout are discussed next.

Fixed face

The **Fixed face** display area is used to specify the face that will be fixed while opening the sheet to create the flat pattern. Select a face; the face will be highlighted in different color and its name will be displayed in the **Fixed face** display area. You can select any face from the drawing area that needs to be fixed while creating the flat pattern.

Merge faces

The **Merge faces** check box is used to merge the flat faces and the bending faces while creating the flat pattern. This check box is selected by default. If you clear this check box, the flat faces and bend faces will not be merged. Figure 16-81 shows the flat pattern of a sheet metal component with the **Merge faces** check box selected. Figure 16-82 shows the flat pattern of a sheet metal component with the **Merge faces** check box cleared.

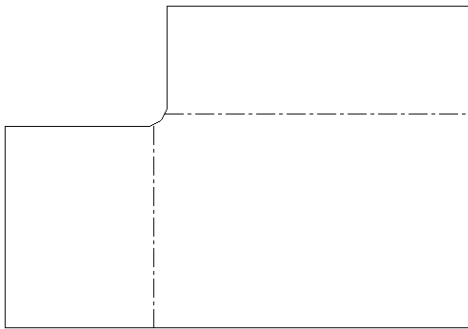


Figure 16-81 Flat pattern with the **Merge faces** check box selected

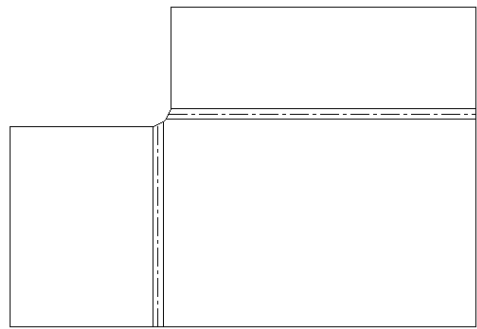


Figure 16-82 Flat pattern with the **Merge faces** check box cleared

Simplify bends

The **Simplify bends** check box is selected by default and is used to straighten the curved edges of the sheet metal component in the flat pattern. If you clear this check box, the curved edge will not be straightened in the flat pattern.

Corner Options

The **Corner Options** rollout is used to set the option to dress up the corners of the flattened sheet metal component. The option available in this rollout is discussed next.

Corner Treatment

The **Corner Treatment** check box is selected by default and is used to automatically apply the corner treatment to the flattened sheet. This option removes or adds the material at the corners of the sheet. If you clear this check box, the corner treatment is not applied to the flattened sheet. Figure 16-83 shows the flat pattern of a sheet metal component with the **Corner Treatment** check box selected. Figure 16-84 shows the flat pattern of a sheet metal component with the **Corner Treatment** check box cleared.

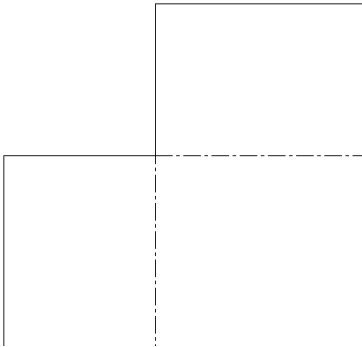


Figure 16-83 Flat pattern with the **Corner Treatment** check box selected

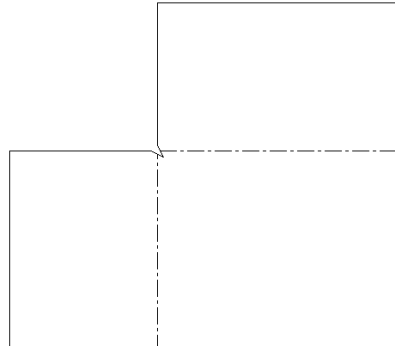


Figure 16-84 Flat pattern with the **Corner Treatment** check box cleared

After setting all options, choose the **OK** button from the **Flat-Pattern PropertyManager**. Now to flatten the sheet metal component, choose the **Flatten** button from the **Sheet Metal** toolbar. You can also select the **Flat-Pattern1** feature from the **FeatureManager design tree** and invoke the pop-up toolbar. Next, choose the **Unsuppress** option from this toolbar to flatten the sheet metal component. The sheet metal component is flattened by selecting the base flange as the face to be fixed. When there are multiple parts, in SolidWorks 2010, you can also choose the **Flatten** option by expanding the **Cut list** node and right-clicking on the name of the corresponding part.

CREATING SHEET METAL COMPONENTS FROM A FLAT SHEET

In SolidWorks, you can create a sheet metal component by first creating the flat pattern of the sheet and then adding bends to it to get the required shape of the component. Consider an example of the sheet metal component shown in Figure 16-85. The flat pattern of this component is shown in Figure 16-86.

To create this component from a flat sheet, you first need to create the base flange similar to the flat pattern by invoking the **Base Flange/Tab** tool, as shown in Figure 16-87. Next, create

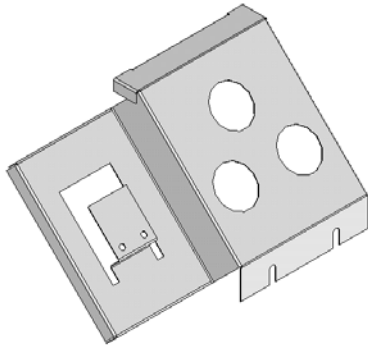


Figure 16-85 Sheet metal component

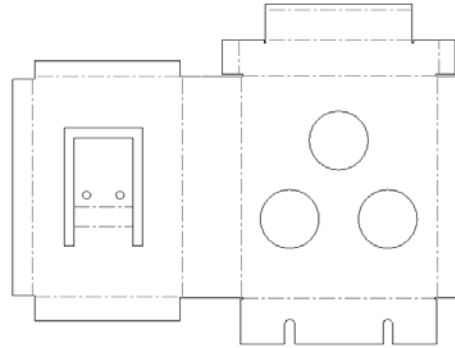


Figure 16-86 Flat pattern of the sheet metal component

the bend lines in a single sketch, as shown in Figure 16-88. Now, choose the **Sketched Bend** tool to bend the sheet metal component along the sketch created as the bending lines, as shown in Figure 16-89. You can also add other sheet metal features to complete the component.

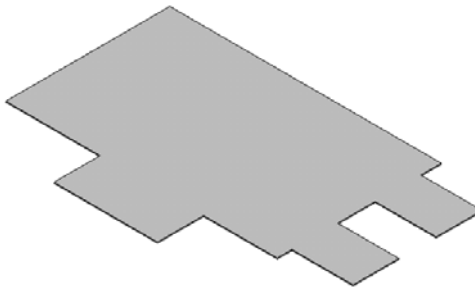


Figure 16-87 Base flange

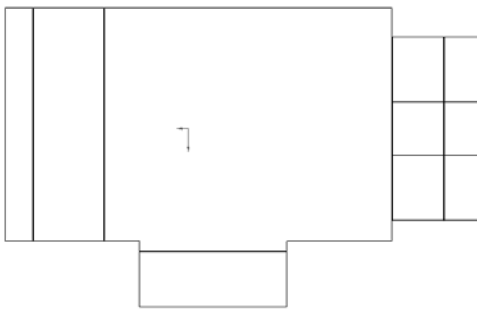


Figure 16-88 Bend lines

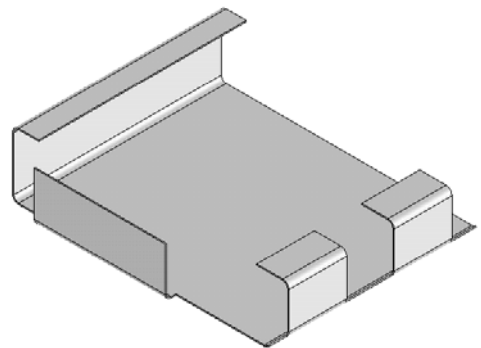


Figure 16-89 Sheet metal component with bend

CREATING A SHEET METAL COMPONENT FROM A FLAT PART

You can create a sheet metal component by creating the flat state of the sheet as a solid part in the **Part** mode and then converting the solid part into sheet metal. To create a sheet metal component by using this method, create a closed sketch that will define the flat state of the sheet. Extrude the sketch using the blind depth equal to the thickness of the flat sheet using the **Extruded Boss/Base** tool. Next, convert the flat part into a sheet metal component in flattened state. The procedure to convert a flat part into a sheet metal component is discussed next.

Converting a Part or a Flat Part into Sheet Metal by Adding Bends

CommandManager: Sheet Metal > Insert Bends
SolidWorks menus: Insert > Sheet Metal > Insert Bends
Toolbar: Sheet Metal > Insert Bends



To convert a part into a sheet metal component, you need to add bends to the part. The bends are added to the part using the **Insert Bends** tool. To do so, create a solid part and choose the **Insert Bends** button from the **Sheet Metal CommandManager**; the **Bends PropertyManager** will be displayed and you will be prompted to select the fixed face or edge and set the bend parameters. The partial view of the **Bends PropertyManager** is shown in Figure 16-90.

Select the top face of the flat part, as shown in Figure 16-91; the selected face will be highlighted in different color and the name of the face will be displayed in the **Fixed Face or Edge** selection box. Set the parameters of the bend allowance and the relief in the **Bend Allowance** and **Auto Relief** rollouts, respectively. After specifying all these parameters, choose the **OK** button from the **Bends PropertyManager**; the **SolidWorks** message box will be displayed, as shown in Figure 16-92. It will inform you that no bends were found. Choose the **OK** button from the **SolidWorks** message box.

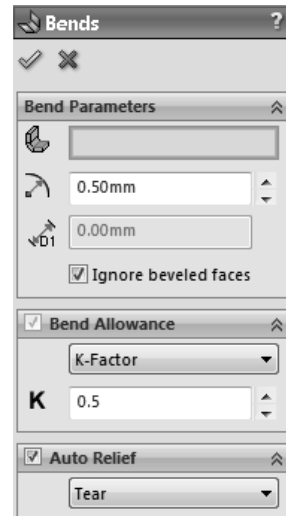


Figure 16-90 Partial view of the Bends PropertyManager

After choosing the **OK** button from the **SolidWorks** message box, you will notice that some new nodes are added to the **FeatureManager design tree**. The usage of these nodes is discussed later in this chapter. Note that although no bends are added to the flat part, you can observe that the flat part is converted into a sheet metal part. Now, you can add all features of a sheet metal component to this part.

Adding Bends to the Flattened Sheet Metal Component

After converting the solid part into a sheet metal component, you can add bends to the sheet metal component. There are two methods to add bends to the flattened sheet metal

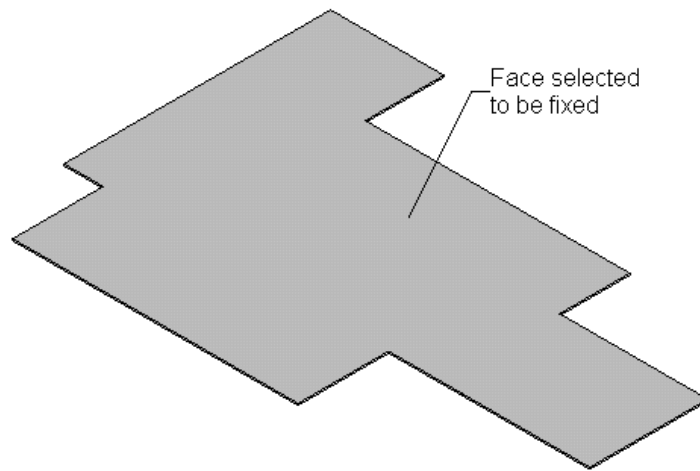


Figure 16-91 Face selected to be fixed

components that are extracted from a flat part. The first method of creating sketched bends has been discussed earlier in this chapter and the second method is discussed next.

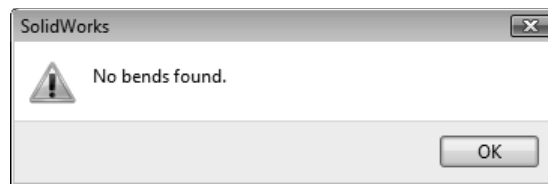


Figure 16-92 The SolidWorks message box

Creating Sketched Bends Using the Process Bends Method

The Process Bends method is used to create sketched bends in the sheet metal components extracted from a part. These bends are also called Flat Bends. All bends created using the bending lines are placed in the **Process-Bends1** node in the **FeatureManager design tree**. To create the bends using this method, expand the **Process-Bends1** node and select the **Flat-Sketch1** node. Next, choose the **Edit Sketch** option from the pop-up toolbar; the sketching environment will be invoked. Create the sketch of the bending lines and exit the sketching environment; the sheet metal component will be bent along the bend lines. If you expand the **Process-Bends1** node again, you will observe that all bend features that you have added using the process bends are displayed. Figure 16-93 shows the sketches that will be used as bend lines to create bends. Figure 16-94 shows the resulting bent sheet metal component.



Tip. If you need to edit the radius of the bends individually, expand the **Process-Bends1** node in the **FeatureManager design tree**. Select the bend that you need to modify and invoke the pop-up toolbar. Next, choose the **Edit Feature** option from this toolbar; the **FlatBend PropertyManager** will be displayed, which can be used to edit the parameters of the bend.

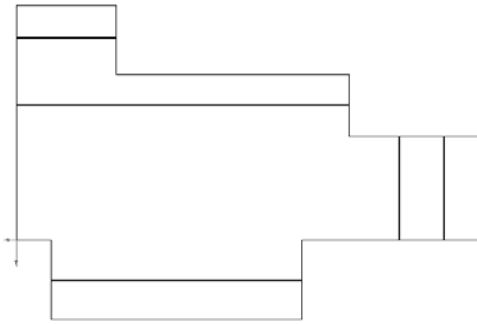


Figure 16-93 Bend lines

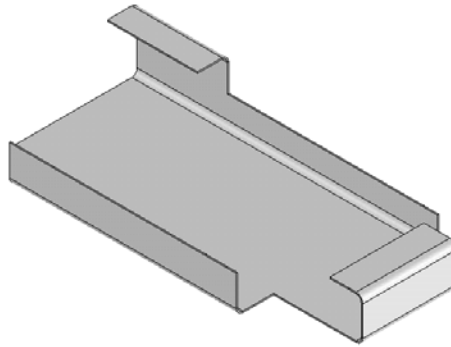


Figure 16-94 Resulting bending

Unbending the Sheet Metal Part Using the No Bends Tool

CommandManager: Sheet Metal > No Bends

Toolbar: Sheet Metal > No Bends



The **No Bends** tool is used to straighten the bends in the sheet metal component that are created from a solid part and roll it back to the stage when it did not have any bends. Choose the **No Bends** button from the **Sheet Metal** toolbar; the **FeatureManager design tree** roll backs to the stage where no bends were added. It is a toggle tool. You can also roll back a sheet metal component by rolling up the rollback bar above the **Flatten-Bends1** feature from the **FeatureManager design tree**.

After invoking the **No Bends** tool, if you add an extruded feature to the part such that the depth of the extruded feature is equal to the sheet thickness, it will automatically be converted into a flange when you resume the sheet metal part. Consider the model shown in Figure 16-95. This figure shows an extruded feature added to an unbent sheet metal part. Now, if you choose the **No Bends** button from the **Sheet Metal** toolbar, the flange with the default settings for bending and relief will be created, as shown in Figure 16-96. If you want to specify custom bending and relief, expand the **Flatten-Bends1** node in the **FeatureManager design tree** and select **SharpBend1**. Next, invoke the pop-up toolbar and choose the **Edit Feature** option from it. You can define the custom parameters of bend radius, bend allowance, and relief by using the **SharpBend PropertyManager**.

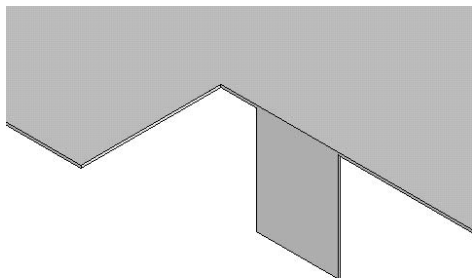


Figure 16-95 Model with an extruded feature

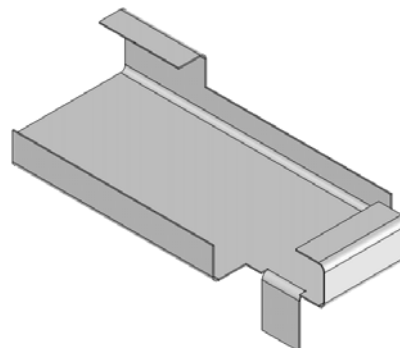


Figure 16-96 Flange with the default settings for bending and relief



Tip. Remember that if the original sketch of the sheet metal part had multiple lines at an angle to each other such as the L or U sections, they will not be unbent.

CREATING A SHEET METAL COMPONENT BY DESIGNING IT AS A PART

SolidWorks provides you with an option to first design the entire part in the part mode and then convert it into a sheet metal component. Consider the example of a sheet metal component shown in Figure 16-97. To create this component, create the design of the sheet metal component by using the part modeling tools, as shown in Figure 16-98.

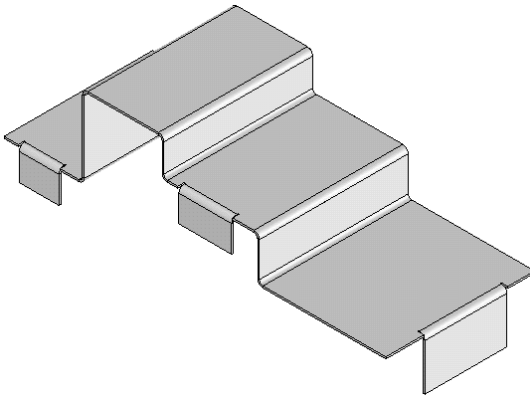


Figure 16-97 A sheet metal component

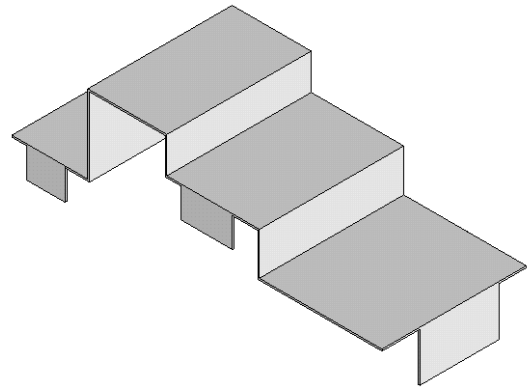


Figure 16-98 Component designed as a part

After designing it as a part, invoke the **Insert Bends** tool from the **Sheet Metal** toolbar; the **Bends PropertyManager** will be displayed. Select the face that will be fixed and specify the sheet metal parameters. Choose the **OK** button from the **Bends PropertyManager**; the part file will be converted into a sheet metal part. If some reliefs are added to the sheet metal component, the **SolidWorks** message box will be displayed and you will be informed that auto relief cuts were made for one or more bends. Choose the **OK** button from this message box; the sheet metal component will be created. Figure 16-99 shows the flat pattern of the sheet metal component shown in Figure 16-97.

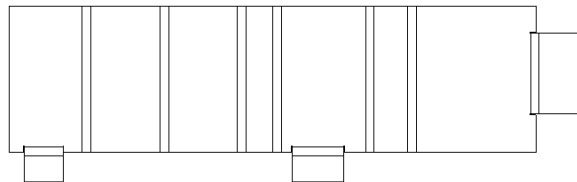


Figure 16-99 Flat pattern of the sheet metal component

Understanding the Types of Bends

Now, you need to learn various types of bends that are added to sheet metal components when you create the sheet metal components by converting a part into sheet metal. The types of bends that are added to the sheet metal components during this conversion are discussed next.

Sharp Bends

If you create a part with sharp edges and convert it into a sheet metal component, the bends added to the sheet metal component are recognized as sharp bends. The sharp bends are placed in the **Flatten-Bends1** feature in the **FeatureManager design tree**. Figure 16-100 shows a part created with sharp edges. Figure 16-101 shows the part converted into the sheet metal component.

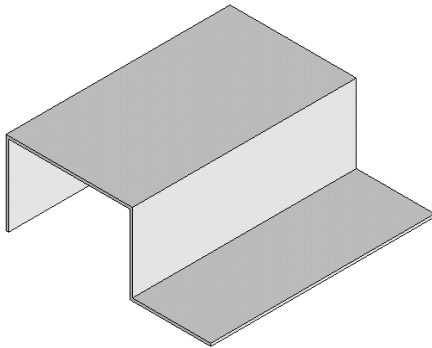


Figure 16-100 Part created with sharp edges

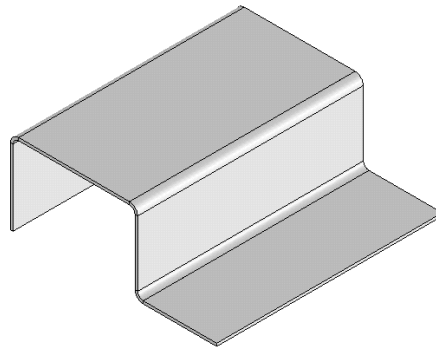


Figure 16-101 Part converted into the sheet metal component

Round Bends

If you create a part with rounded edges and convert it into a sheet metal component, the bends added to the sheet metal component are recognized as rounded bends. The rounded bends are placed in the **Flatten-Bends1** feature in the **FeatureManager design tree**. Figure 16-102 shows a part created with rounded edges. Figure 16-103 shows the part converted into the sheet metal component.

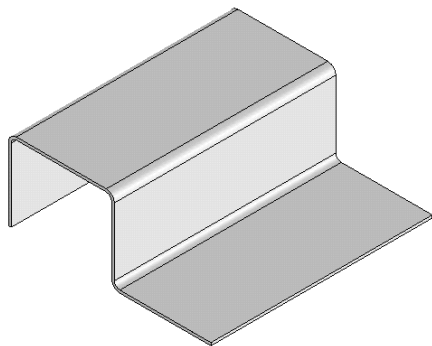


Figure 16-102 Part with rounded edges

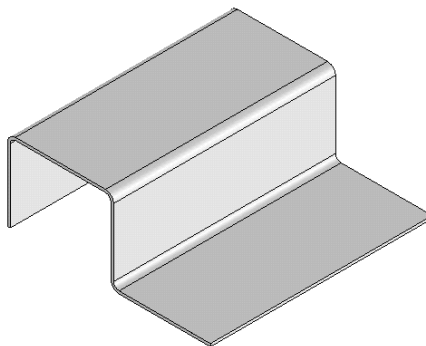


Figure 16-103 Part converted into the sheet metal component

Flat Bends

The flat bends are the bends that are created by bending the flattened sheet. The flat bends are placed in the **Process-Bends1** feature in the **FeatureManager design tree**. The procedure to create these type of bends has been discussed earlier.

CONVERTING A SOLID BODY INTO A SHEET METAL PART

CommandManager:	Sheet Metal > Convert to Sheet Metal
SolidWorks menus:	Insert > Sheet Metal > Convert To Sheet Metal
Toolbar:	Sheet Metal > Convert to Sheet Metal

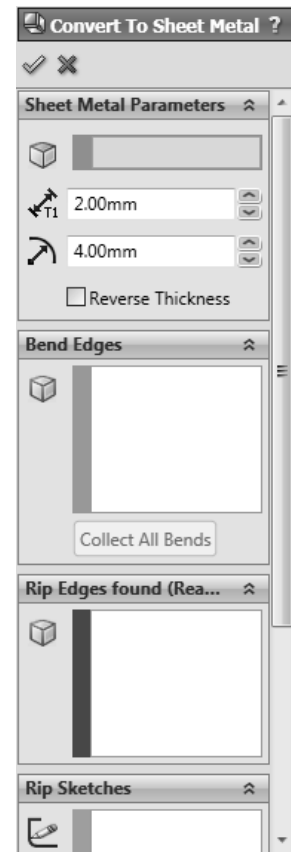
You can convert a solid body into a sheet metal part using the **Convert to Sheet Metal** tool. To do so, create a solid body in the **Part** mode and then choose the **Convert to Sheet Metal** button from the **Sheet Metal CommandManager**; the **Convert To Sheet Metal PropertyManager** will be displayed, as shown in Figure 16-104. Next, select the face of the solid body that will be fixed while opening the sheet to create the flat pattern, as shown in Figure 16-105; the selected face will be highlighted in green and its name will be displayed in the **Select a fixed entity** selection box in the **Sheet Metal Parameters** rollout. Set the sheet thickness and the radius of the bend using the **Sheet thickness** and **Default radius for bends** spinners. You can flip the direction of the sheet metal thickness by selecting the **Reverse Thickness** check box in this rollout.

Next, select the edges of the solid body, as shown in Figure 16-106; the name of the selected edges will be displayed in the **Select edges/faces that represent bends** selection box of the **Bend Edges** rollout. Note that the corresponding rib edges of the bend edges are selected automatically and their names will be displayed in the **Automatically found rib edges** selection box. After setting the required parameters, choose the **OK** button from the **Convert To Sheet Metal PropertyManager**; the solid body will be converted into a sheet metal component. Figure 16-107 shows the solid body to be converted into a sheet metal component and Figure 16-108 shows the flat pattern of the resulting sheet metal component.

The other rollouts in this PropertyManager are discussed next.

Rip Sketches Rollout

This rollout is used to define the required rips. The **Select a sketch to add a rip** selection box in this rollout is used to select a sketch from the drawing area to define the required rip. To do



*Figure 16-104 Partial view of the **Convert To Sheet Metal PropertyManager***

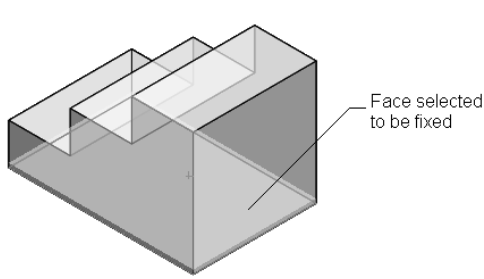


Figure 16-105 The face selected to be fixed

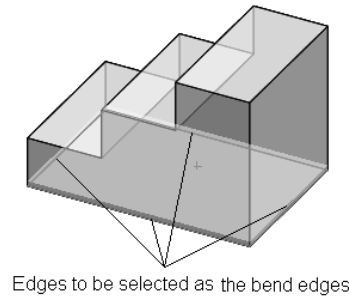


Figure 16-106 Edges selected

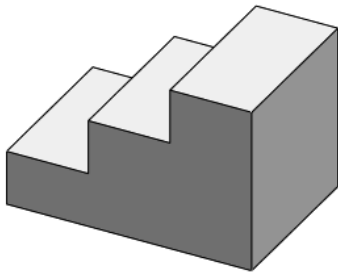


Figure 16-107 The solid part to be converted into a Sheet Metal component

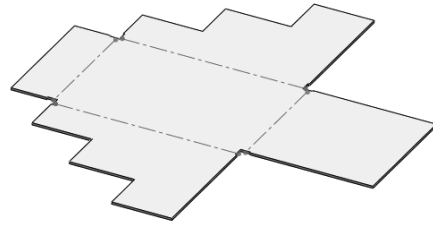


Figure 16-108 The flat pattern of the Sheet Metal component

so, you first need to create a sketch of the rip in the sketching environment. You can specify the gap between the rips by using the **Default gap for all rips** spinner in this rollout.

Auto Relief Rollout

The options in the **Auto Relief** rollout of this PropertyManager are same as discussed earlier in the **Base Flange PropertyManager**.

Choose the **OK** button after setting all parameters; the selected solid part will be converted into sheet metal part. Note that if there are multiple bodies in solid part, then you need to convert each body to sheet metal part separately.

DESIGNING A SHEET METAL PART FROM A SOLID SHELLED MODEL

In SolidWorks you can design a sheet metal part as a solid model and then shell the model. Remember that while shelling the model, you need to remove at least one face. After shelling the model, you need to rip the edges of the thin solid model. The ripping is done in order

to cut the sheet so that it can be opened easily while creating the flat pattern. The procedure to rip the edges of a solid part is discussed next.

Ripping the Edges

CommandManager: Sheet Metal > Rip
SolidWorks menus: Insert > Sheet Metal > Rip
Toolbar: Sheet Metal > Rip



The **Rip** tool is used to add a gap between the edges of a shelled solid part before converting it into a sheet metal component. To rip the edges, choose the **Rip** button from the **Sheet Metal CommandManager**; the **Rip PropertyManager** will be displayed, as shown in Figure 16-109. Also, you will be prompted to set the rip gap and select the edge(s) to rip. Select the internal edges that you need to rip; the direction arrows will be displayed on the selected edge; the name of the selected edge will be displayed in the **Edge to Rip** selection box. The two arrows displayed on the selected edge indicates that the ripping will be done on both the sides of the selected edge. Use the **Change Direction** button to toggle between the two directions. The **Rip Gap** spinner is used to set the value of the rip gap.

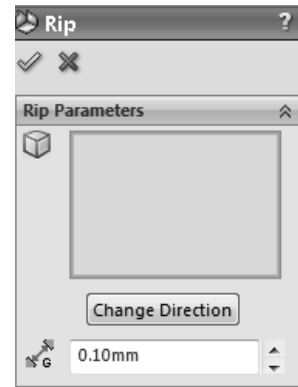


Figure 16-109 The **Rip PropertyManager**

Figure 16-110 shows the edge selected to create the rip in both directions. Figure 16-111 shows the resulting rip.

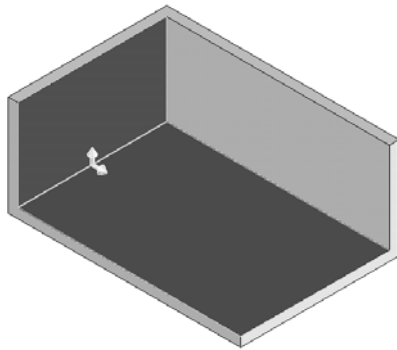


Figure 16-110 Edge selected to create the rip

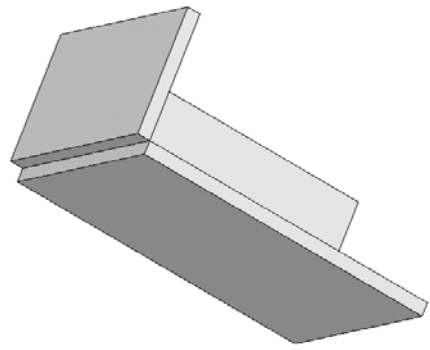


Figure 16-111 Resulting rip

After ripping the edges, choose the **Insert Bends** button to invoke the **Bends PropertyManager**. Next, select the fixed face, set the sheet metal parameters and then choose the **OK** button. Figure 16-112 shows the shelled solid model and Figure 16-113 shows the model after ripping and converting it into a sheet metal component. Figure 16-114 shows the flat pattern of the sheet metal component.

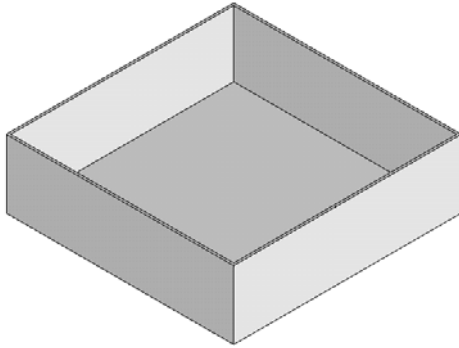


Figure 16-112 Shelled solid model

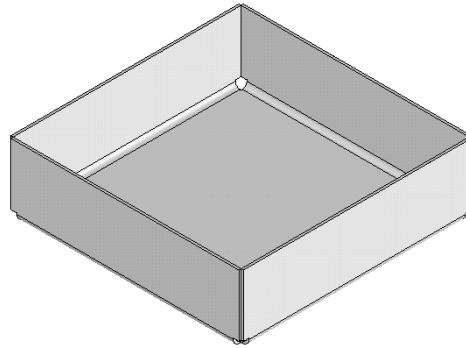


Figure 16-113 Model after ripping and converting it into a sheet metal component

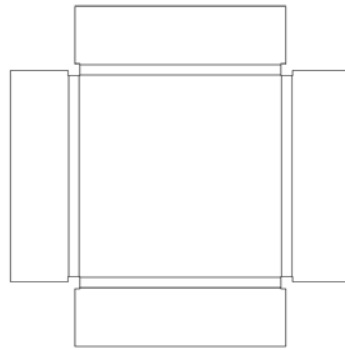


Figure 16-114 Flat pattern of the sheet metal component



Tip. You can also rip the edges by using the **Rip Parameters** rollout available in the **Bends PropertyManager**.

CREATING CUTS IN SHEET METAL COMPONENTS ACROSS THE BENDS

In this section, you will learn to create cuts across the bends, as shown in Figure 16-115. The methods of creating cuts across the bends are different for the sheet metal components created from the base flange and the sheet metal component created by converting a solid part. The methods for creating cuts for both types of sheet metal components are discussed next.

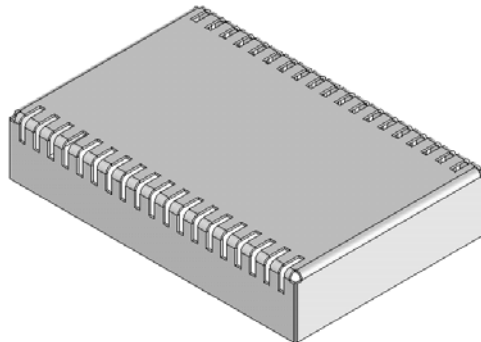


Figure 16-115 A sheet metal component with cuts across the bends

Creating Cuts in a Sheet Metal Component Created from a Solid Model

After creating the solid model, convert it into a sheet metal component by invoking the **Insert Bends** tool. Then, create the bends by editing the sketch in the **Process-Bends** node. Next create the flat pattern of the sheet metal component by invoking the **No Bends** tool. Now, select the face on which you need to create the cut and invoke the sketching environment. Create the sketch and extrude it to create the cut feature. Make sure that the cut is across the bend created by using the **Process-Bends** node. Next, invoke the **No Bends** tool again; the cut will be created across the bends.

Consider the sheet metal component shown in Figure 16-116. Invoke the **No Bends** tool and create the flat pattern of the sheet metal component, as shown in Figure 16-117. Select the top face of the flattened sheet metal component as the sketching plane and invoke the sketching environment. Create the sketch and extrude the cut by using the **Link to thickness** option from the **Extrude PropertyManager**. After creating the cut, use the **Linear Pattern** tool to create a linear pattern of the cut feature. The flattened sheet metal component after creating and patterning the cut feature is shown in Figure 16-118. Next, invoke the **No Bends** tool again to display the sheet metal component with cuts across the bends, as shown in Figure 16-119.

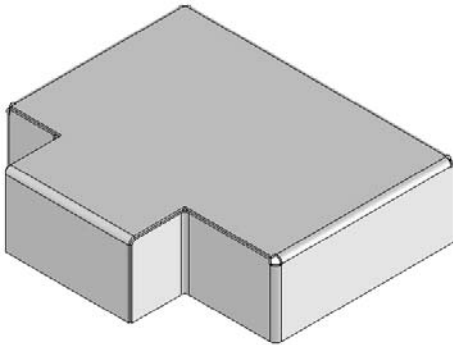


Figure 16-116 Sheet metal component

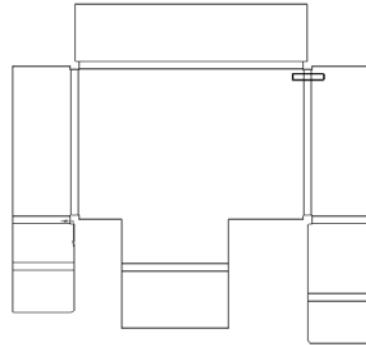


Figure 16-117 Flat pattern of the sheet metal component

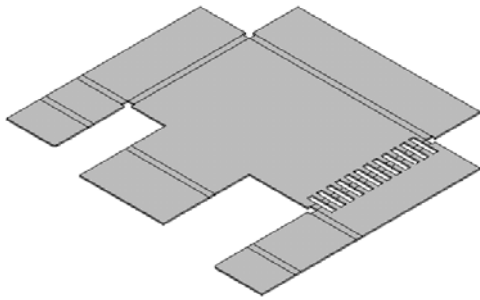


Figure 16-118 Flattened sheet after creating and patterning the cuts

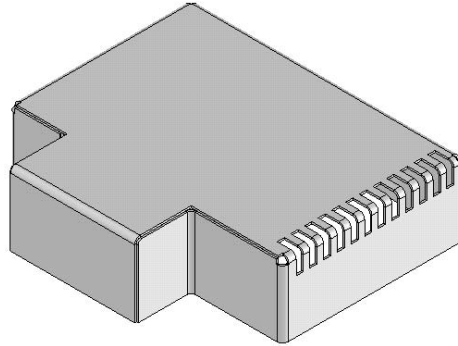


Figure 16-119 Final sheet metal component

Note



If you choose the **Flatten** button from the **Sheet Metal** toolbar and create the cuts on a flattened sheet, the cuts will not be displayed in the unflattened sheet metal component.

Creating Cuts in a Sheet Metal Component Created Using the Base Flange

A sheet metal component designed by using a base flange does not include the **Flatten-Bends1** and the **Process-Bends1** features. Therefore, for creating cuts in such a sheet metal component, you first need to unfold the sheet using the **Unfold** tool and then create the cut feature. After creating the feature, you need to fold the sheet again using the **Fold** tool. Consider the example of the sheet metal component displayed in Figure 16-120. For creating cuts in this sheet metal component, you first need to unfold the sheet. The method of unfolding the sheet is discussed next.

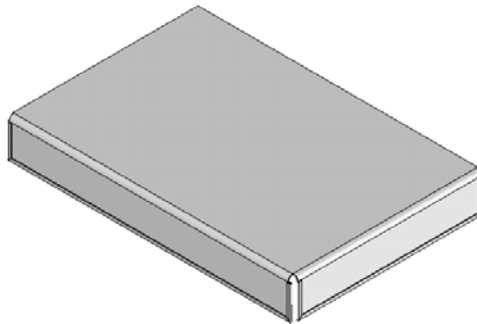


Figure 16-120 A sheet metal component

Unfolding the Sheet

CommandManager: Sheet Metal > Unfold
SolidWorks menus: Insert > Sheet Metal > Unfold
Toolbar: Sheet Metal > Unfold



To unfold a sheet, invoke the **Unfold** tool by choosing the **Unfold** button from the **Sheet Metal CommandManager**; the **Unfold PropertyManager** will be displayed, as shown in Figure 16-121. Also, you will be prompted to select a face to be fixed and the bends to be unfolded.



Figure 16-121 The *Unfold PropertyManager*

Select the face that you need to fix and the bends that you need to unfold. To unfold all bends, choose the **Collect All Bends** button in the **Selections** rollout and choose the **OK** button from the **Unfold PropertyManager**. You will notice that the sheet metal component is unfolded. Figure 16-122 shows the unfolded sheet.

After unfolding the sheet, create the required cut feature. The unfolded sheet after creating the cut feature is shown in Figure 16-123. Fold the sheet again after creating the cut feature.

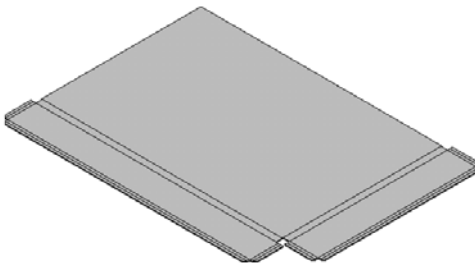


Figure 16-122 The *unfolded sheet*

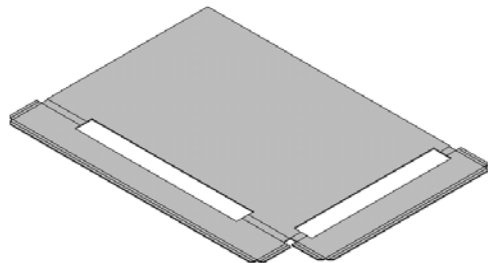


Figure 16-123 The *unfolded sheet after creating the cut feature*

Folding the Sheet

CommandManager: Sheet Metal > Fold
SolidWorks menus: Insert > Sheet Metal > Fold
Toolbar: Sheet Metal > Fold



To fold an unfolded sheet, choose the **Fold** button from the **Sheet Metal CommandManager**; the **Fold PropertyManager** will be displayed, as shown in Figure 16-124. Also, you will be prompted to select a face to be fixed and the bends to be folded.

The face of the sheet that was fixed while unfolding the sheet is selected by default when you invoke the **Fold PropertyManager**. You can also select any other face that you need to fix while folding the sheet. Choose the **Collect All Bends** button to select all bends to be folded

and then choose the **OK** button from the **Fold PropertyManager**. Figure 16-125 shows the final sheet metal component after folding it using the **Fold** tool.

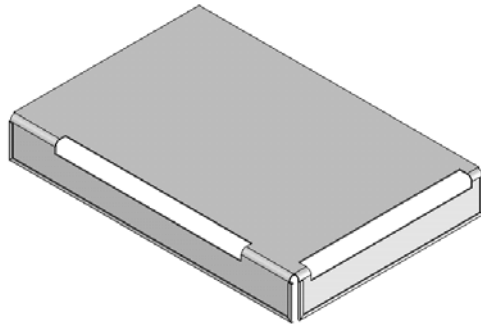
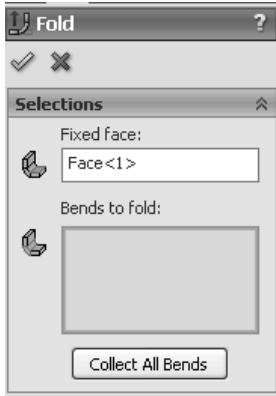


Figure 16-124 The **Fold PropertyManager** **Figure 16-125** Sheet metal component after folding



Note

If you create edge flanges throughout the length of edges of the base flange, SolidWorks may not fold the component back.

Creating Cylindrical and Conical Sheet Metal Components

SolidWorks also allows you to create cylindrical and conical sheet metal components. For creating a cylindrical or conical sheet metal component, you need to make sure that there is some gap to unfold the sheet. Create a conical or cylindrical sheet metal part and then invoke the **Bends PropertyManager** to convert the part into a sheet metal component. Now, select a linear edge of the conical or cylindrical part that will be fixed, refer to Figure 16-126. Choose the **OK** button from the **Bends PropertyManager**. Next, choose the **Flatten** button. Figure 16-127 shows the flat pattern of the conical sheet metal component.

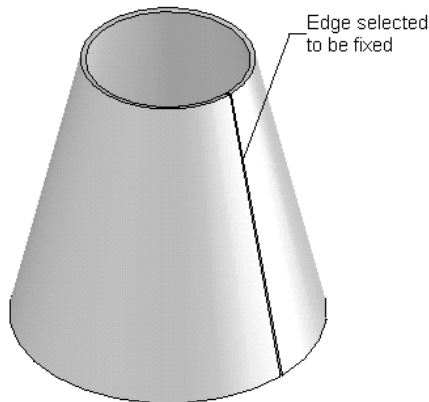


Figure 16-126 Edge selected to be fixed

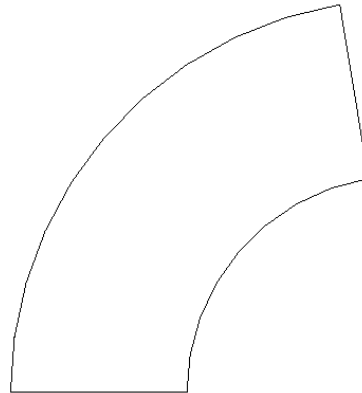


Figure 16-127 Flat pattern of the conical sheet metal component



Note

*If you create a cylindrical or conical sheet metal component by lofting, you can directly use the **Flat Pattern** option to create the flat pattern.*

GENERATING THE DRAWING VIEW OF THE FLAT PATTERN OF THE SHEET METAL COMPONENTS

After creating a sheet metal component, the next step is to generate the drawing view of the flat pattern of the sheet metal component. To generate the drawing view of the flat pattern, select the **Flat Pattern** check box from the **Orientation** rollout in the **Model View PropertyManager** and place the drawing view on the sheet. Figure 16-128 shows a sheet metal component and Figure 16-129 shows its resulting flat pattern view. Figure 16-130 shows the drawing view of the flat pattern of the sheet metal component.

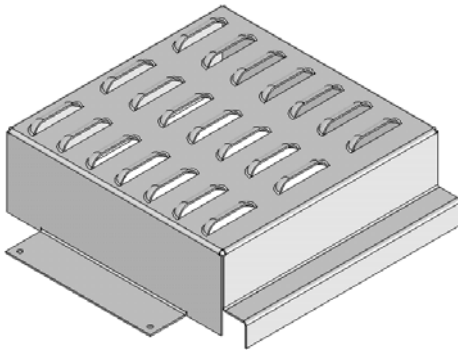


Figure 16-128 Sheet metal component

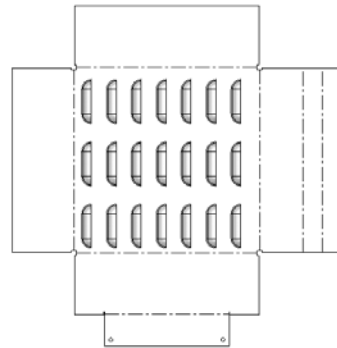


Figure 16-129 Flat pattern

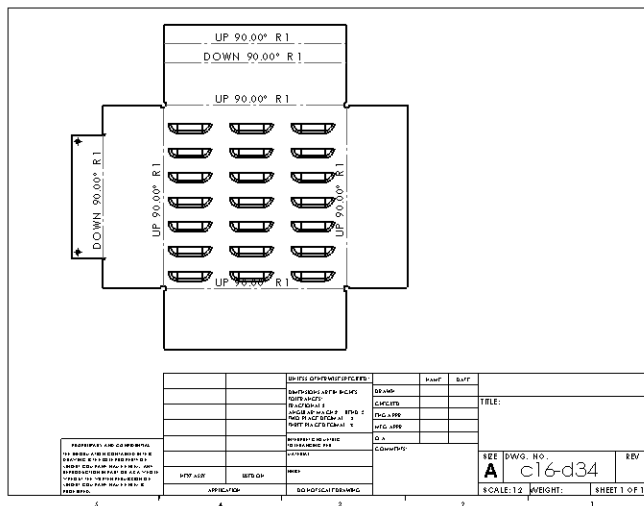


Figure 16-130 Drawing view of the flat pattern

TUTORIALS

Tutorial 1

In this tutorial, you will create the sheet metal component shown in Figure 16-131. The flat pattern of the sheet metal component and its views and dimensions are shown in Figures 16-132 and 16-133, respectively. You will first create the base flange and then the other features. After creating the model, you will create its flat pattern. **(Expected time: 45 min)**

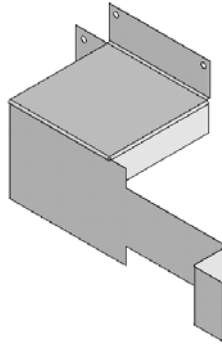


Figure 16-131 Sheet metal component

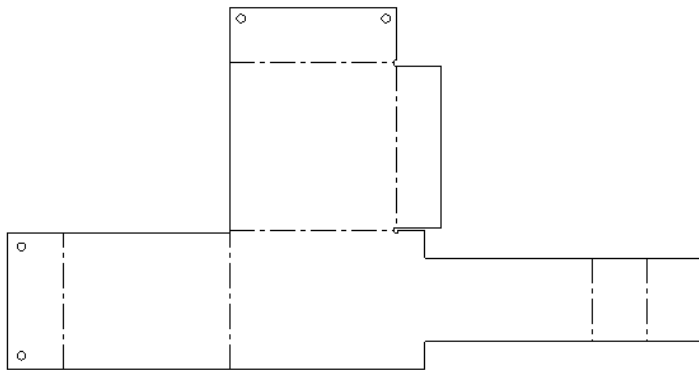


Figure 16-132 Flat pattern of the sheet metal component

The following steps are required to complete this tutorial:

- Create the base flange of the sheet metal component.
- Add other required flanges to the sheet metal component.
- Create the tab feature.
- Add a hem to the right most flange.
- Create the flat pattern of the sheet metal component.
- Save the model.

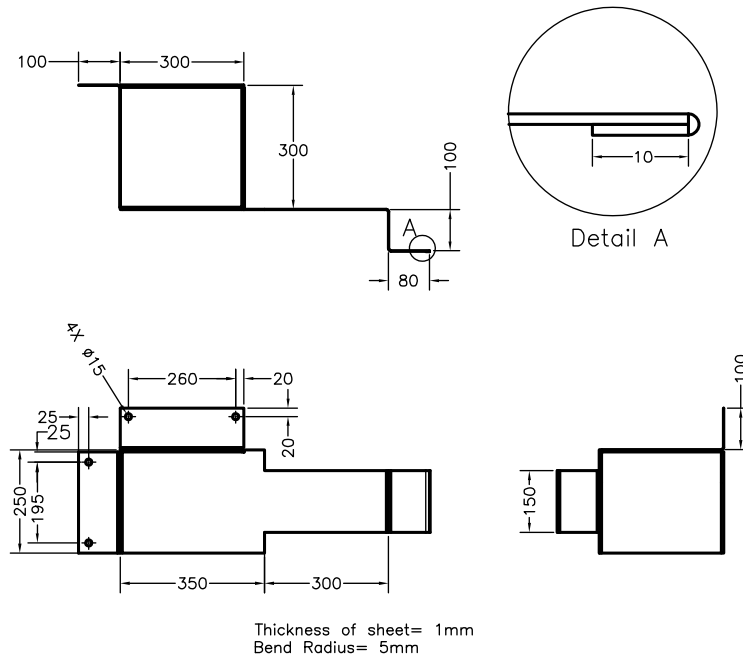


Figure 16-133 Drawing views and dimensions for Tutorial 1

Creating the Base Flange

For creating this sheet metal component, you first need to create the base flange. The base flange will be created by using a rectangular sketch drawn on the Front Plane.

1. Start a new SolidWorks document in the **Part** mode.
2. Invoke the sketching environment using the Front Plane as the sketching plane.
3. Create a rectangle of 350x250 mm as the sketch of the base flange, refer to Figure 16-134.
4. Choose the **Base Flange/Tab** button from the **Sheet Metal CommandManager**; the **Base Flange PropertyManager** is displayed.
5. Set the value of the following parameters and choose the **OK** button from the **Base Flange PropertyManager**.




Thickness: **1** K-Factor: **1** Auto Relief Type: **Rectangular** Ratio: **0.5**

Figure 16-134 shows the base flange created by using the sketch created on the Front Plane as the sketching plane.

Creating the First Edge Flange

After creating the base flange, you need to create the first edge flange using the top edge of the base flange as the reference. You will observe that the **Sheet-Metal1** feature is displayed in the **FeatureManager design tree**. Modify the default bend radius using the **Sheet-Metal1** feature before creating the first edge flange.

1. Select the **Sheet-Metal1** node from the **FeatureManager design tree** to invoke the pop-up toolbar. Choose the **Edit Feature** option from the pop-up toolbar.
2. Set the value in the **Bend Radius** spinner to **5** and choose the **OK** button from the **Sheet-Metal1 PropertyManager**.
3. Choose the **Edge Flange** button from the **Sheet Metal CommandManager**; the **Edge-Flange PropertyManager** is displayed. Also, you are prompted to select a linear edge of a planar face to create the edge flange. 
4. Select the edge of the base flange, as shown in Figure 16-135.

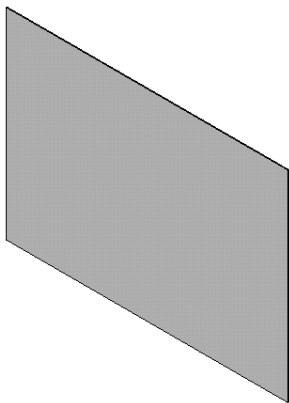


Figure 16-134 Base flange

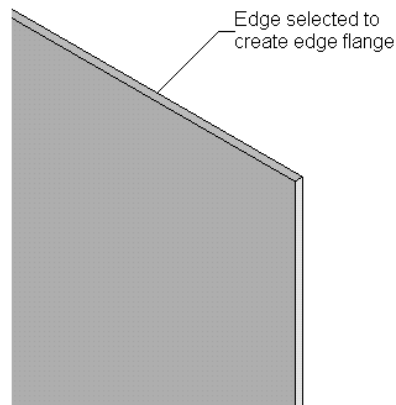


Figure 16-135 Edge selected to create edge flange

5. In the **Flange Length** rollout, set the value in the **Length** spinner to **300**. Choose the **Material Outside** button from the **Flange Position** rollout. Make sure that the flange is being created in the backward direction.

Next, you need to edit the profile of the edge flange.

6. Choose the **Edit Flange Profile** button from the **Flange Parameters** rollout. If this button is not available, clear the **Use default radius** check box and then select it again; the **Edit Flange Profile** button will be available. As soon as you choose this button, the sketching environment is invoked and the **Profile Sketch** dialog box is displayed.
7. Edit the sketch, refer to Figure 16-136. Choose the **Finish** button from the **Profile Sketch** dialog box. The model after creating the first flange is displayed, as shown in Figure 16-137.

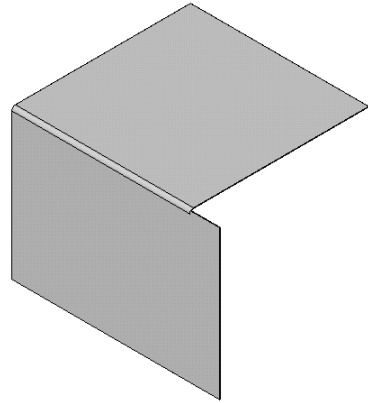
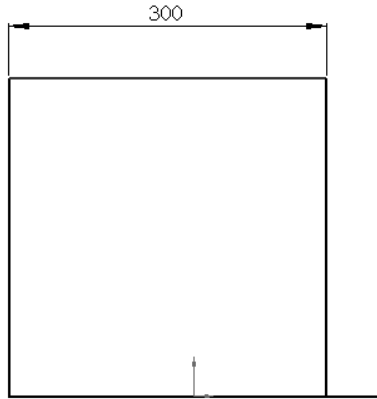


Figure 16-136 Modified sketch of the edge flange **Figure 16-137** Model after creating edge flange

Creating the Second Edge Flange

Next, you need to create the second edge flange with the holes.

1. Choose the **Edge Flange** button from the **Sheet Metal CommandManager** to invoke the **Edge-Flange PropertyManager**.
2. Select the edge, as shown in Figure 16-138; the preview of the edge flange is displayed.
3. Set the value in the **Length** spinner to **100** and choose the **OK** button from the **Edge-Flange PropertyManager**.

The model after creating the second edge flange is shown in Figure 16-139.

4. Similarly, create the edge flanges on the left side of the sheet metal component.
5. Edit the sketch of the edge flanges and draw the sketch for the holes. Refer to Figure 16-133 for dimensions. Edge-flanges with the holes are shown in Figure 16-140.

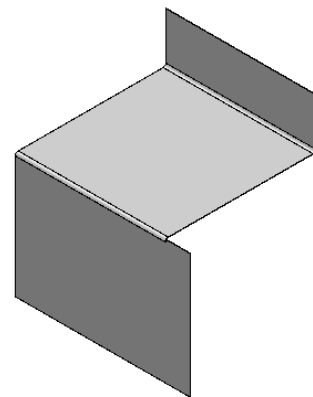
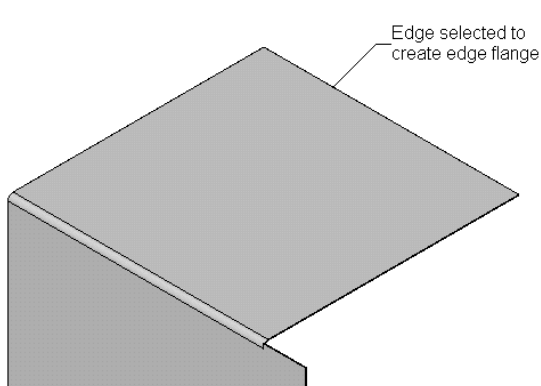


Figure 16-138 Edge selected to create the edge flange **Figure 16-139** Model after creating the second edge flange

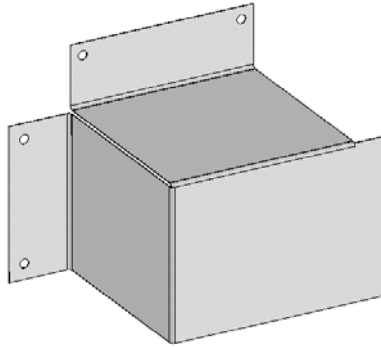


Figure 16-140 Model after creating edge flanges and holes on the left of the model

Creating the Tab and Flange Features

Next, you need to add a tab feature to the sheet metal component. A tab feature is used to add material to the base flange or any other flange feature.

1. Select the front face of the base flange as the sketching plane and invoke the sketching environment.
2. Draw the sketch of the tab feature, as shown in Figure 16-141.
3. Choose the **Base-Flange/Tab** button from the **Sheet Metal CommandManager** and make sure that **Merge Result** check box is selected and choose the **OK** button to create the tab feature. The model after creating the tab feature is displayed in Figure 16-142.

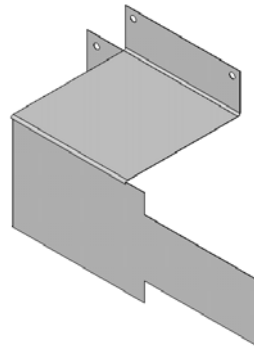
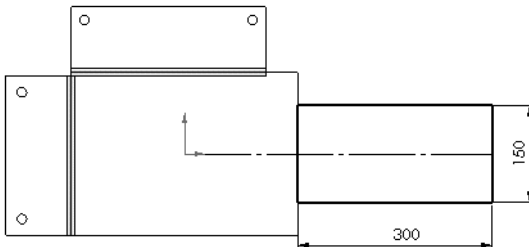


Figure 16-141 Sketch of the tab feature

Figure 16-142 Model after creating the tab fea-

4. Create three more edge flanges, two of length 100 mm and one of length 80 mm, refer to Figure 16-133. The model after creating all the edge flanges is displayed in Figure 16-143.

Creating the Hem Feature

After creating the other sheet metal features, you need to create the hem on the right most edge flange.

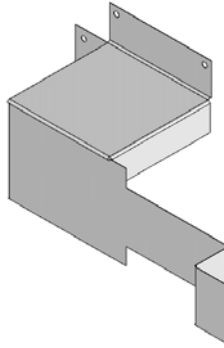



Figure 16-143 Model after creating all edge flanges

1. Choose the **Hem** button from the **Sheet Metal CommandManager**; the **Hem PropertyManager** is displayed. 
2. Choose the **Closed** button from the **Type and Size** rollout and set the value in the **Length** spinner to **10**.
3. Select the edge to create the hem, as shown in Figure 16-144; the hem is created, as shown in Figure 16-145.

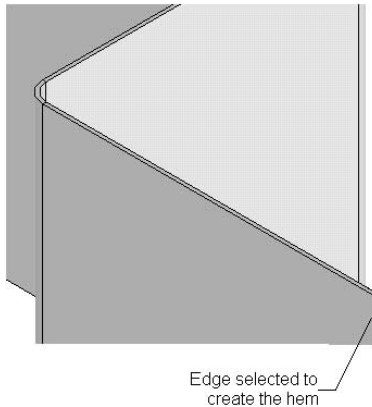


Figure 16-144 Edge selected to create the hem

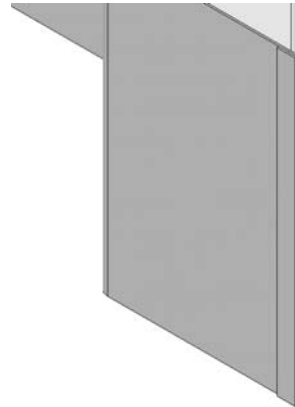



Figure 16-145 Resulting hem

Creating the Flat Pattern

After creating the sheet metal component, you need to create its flat pattern.

1. Choose the **Flatten** button from the **Sheet Metal CommandManager** to create the flat pattern of the sheet metal component. 
2. Orient the flattened model parallel to the screen.

The flattened sheet metal component is displayed in Figure 16-146.

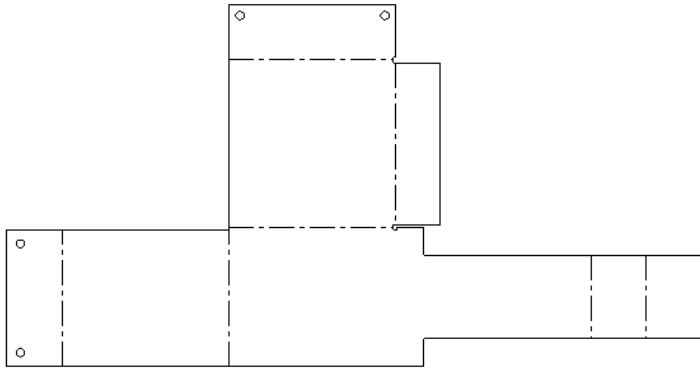


Figure 16-146 *Flattened sheet metal component*

Saving the Model

Next, you need to save the model.

1. Choose the **Save** button from the Menu Bar and save the drawing in the location and the name c16_tut01 in the location given below and close the file.

\Documents\SolidWorks\c16

Tutorial 2

In this tutorial, you will create the sheet metal component shown in Figure 16-147. The flat pattern of the sheet metal component is displayed in Figure 16-148. After creating the model, you need to generate its drawing views, as shown in Figure 16-149. The drawing views and the dimensions for the model are displayed in Figure 16-150. **(Expected time: 1 hr)**

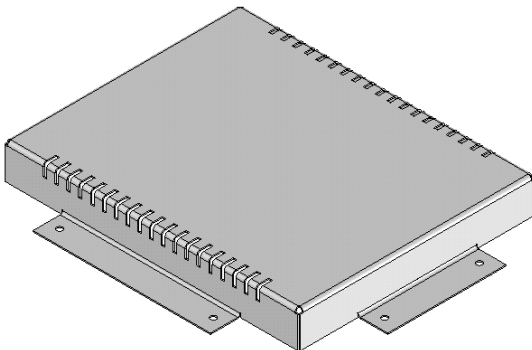


Figure 16-147 *Sheet metal component*

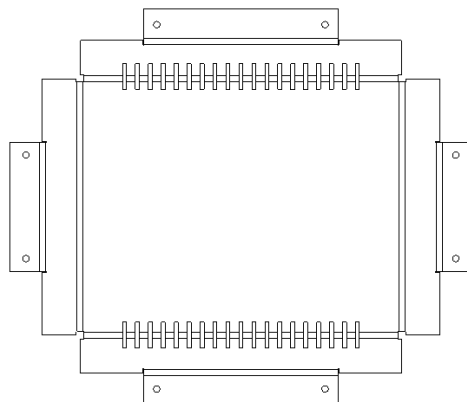


Figure 16-148 *Flattened sheet metal component*

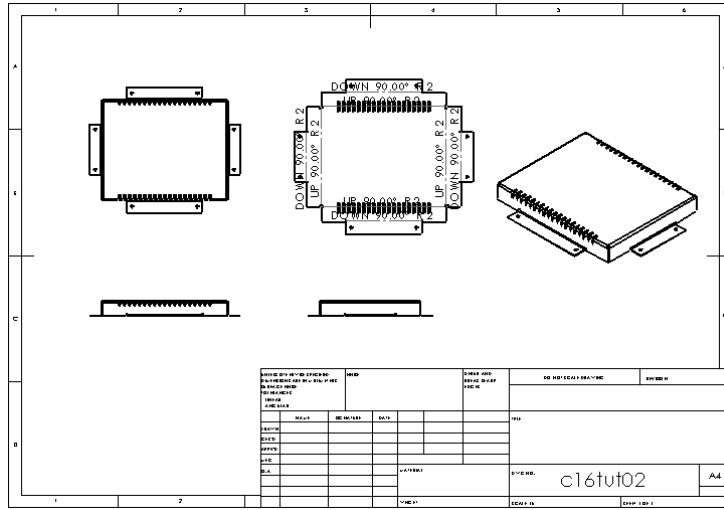
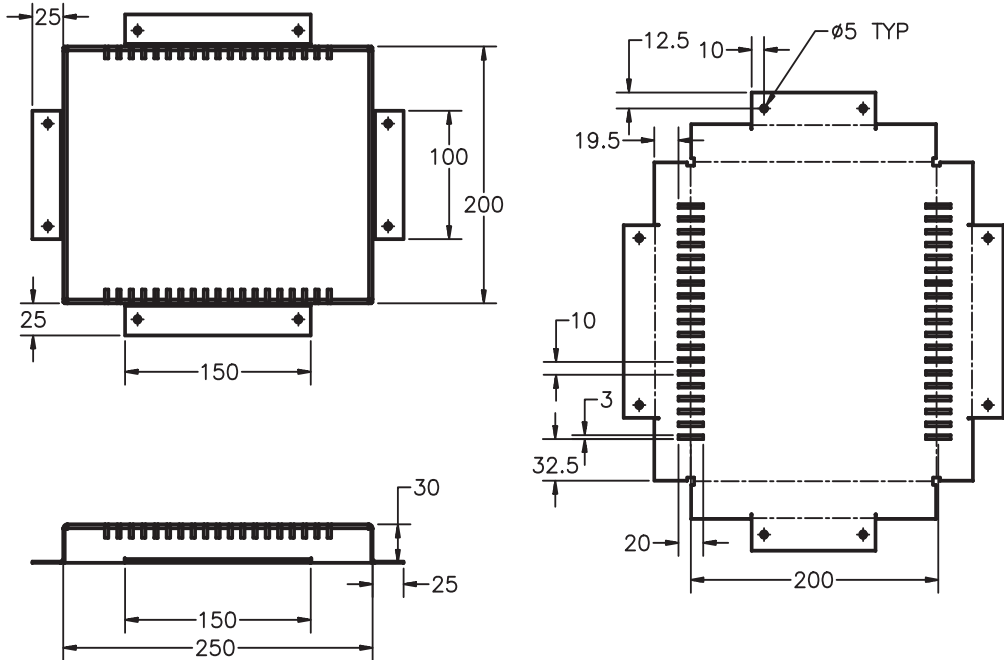


Figure 16-149 Drawing views of the model



The thickness of sheet is 1mm. Bend radius is 2mm

Figure 16-150 Views and dimensions for Tutorial 2

The following steps are required to complete this tutorial:

- a. Create the base feature by extruding the sketch created on the Top Plane.
- b. Shell the base feature.
- c. Convert the shelled solid model into sheet metal component.
- d. Roll back the model to the state where no bends were added to the sheet metal component.
- e. Add flanges to the sheet metal component.
- f. Create the flat pattern of the sheet metal component.
- g. Create slots using the cut feature and pattern them on the sides of the flatten sheet metal component.
- h. Refold the sheet metal component.
- i. Generate the drawing views of the sheet metal component.
- j. Save the model.

Creating the Base Feature

You will create this model by converting a shelled part into a sheet metal component. Therefore, you first need to create the base feature of the model by extruding a sketch created on the Top Plane.

1. Start a new SolidWorks document in the **Part** mode.
2. Invoke the sketching environment by selecting the top plane as the sketching plane.
3. Draw the sketch of the base feature that consists of a rectangle of 250x200.
4. Extrude the sketch to a distance of 30 mm; the base feature of the model is displayed, as shown in Figure 16-151.

Shelling the Base Feature

Next, you need to add the shell feature to the model by removing the bottom face of the base feature.

1. Rotate the model so that its bottom face is clearly visible.
2. Invoke the **Shell** tool and select the bottom face of the base feature as the face to remove.
3. Set the value in the **Thickness** spinner to **1** and choose the **OK** button from the **Shell1 PropertyManager**; the shell feature is created, as shown in Figure 16-152.

Converting the Shelled Model into Sheet Metal Component

After creating the base feature and shelling the model, you need to convert it into a sheet metal component using the **Insert Bends** tool.

1. Choose the **Insert Bends** button from the **Sheet Metal CommandManager** to invoke the **Bends PropertyManager**.



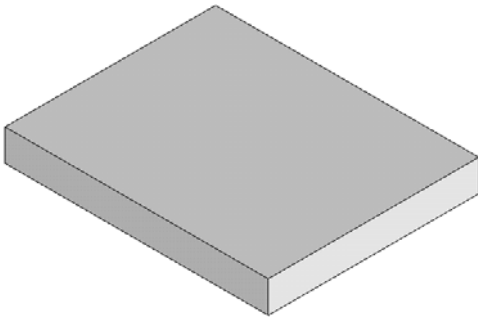


Figure 16-151 Base feature

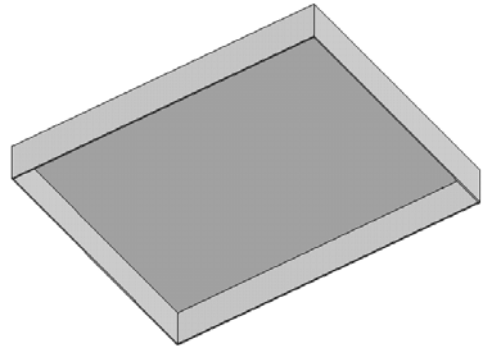


Figure 16-152 Model after shelling

2. Select the top face of the model to fix.
3. Click once in the **Edges to Rip** selection box in the **Rip Parameters** rollout to activate the selection mode.
4. Select all the inner vertical edges of the base feature as the edges to rip.
5. Set the value **2** in the **Bend Radius** spinner and **1** in the **K-Factor** spinner.
6. Choose the **OK** button from the **Bends PropertyManager**; the SolidWorks message box is displayed, which informs you that the auto reliefs are added. Choose the **OK** button from this message box.

The model after converting the solid shelled model into the sheet metal component is shown in Figure 16-153.

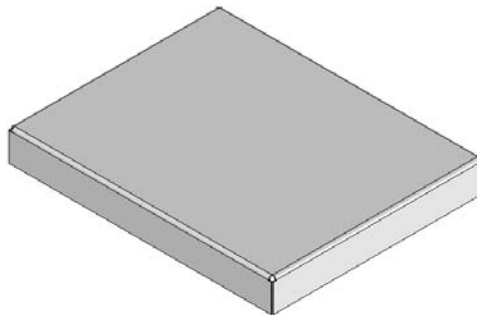
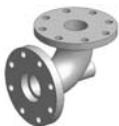



Figure 16-153 Solid shelled model after converting into sheet metal component



Tip. You can also convert a solid model into sheet metal by invoking the **Convert to sheet metal** tool.

Creating the Edge Flanges

Next, you need to add the edge flanges to the sheet metal component. Before adding the edge flanges, you need to roll back the model to the stage when there were no bends in the sheet metal component.

1. Choose the **No Bends** button from the **Sheet Metal CommandManager** to roll back the model to the stage where it had no bends. Note that the four walls of the model are not unbent. 
2. Now, select the bottom face of one of the four walls of the model as the sketching plane and invoke the sketching environment.
3. Draw the sketch of the flanges, as shown in Figure 16-154.

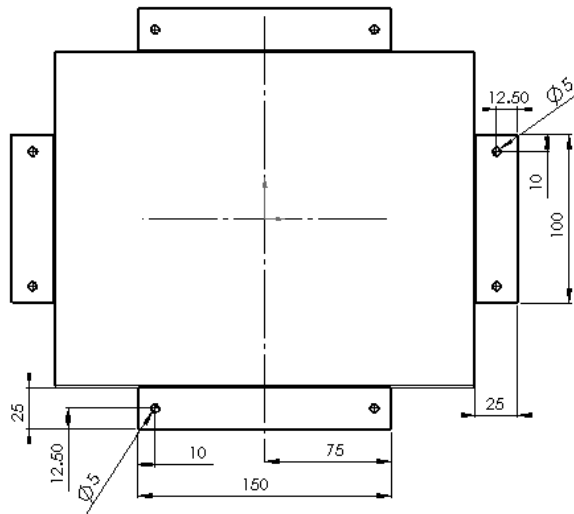


Figure 16-154 Sketch of the flanges

4. Invoke the **Extrude PropertyManager**. Select the **Link to thickness** check box and choose the **Reverse Direction** button.
5. Choose the **OK** button from the **Extrude PropertyManager**. Figure 16-155 shows the model after extruding the flanges.

After extruding the flanges, you need to roll the model back to the bending stage.


6. Again, choose the **No Bends** button from the **Sheet Metal CommandManager** to roll the model to the bending stage. 

Figure 16-156 shows the model after rolling it back to the bending stage.

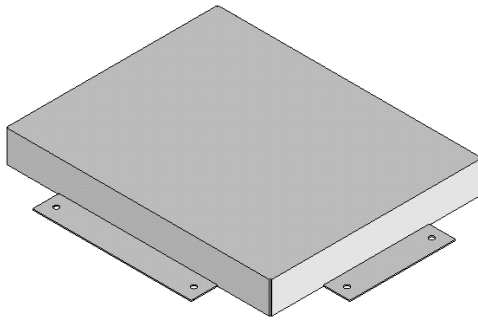


Figure 16-155 Model after extruding the flanges

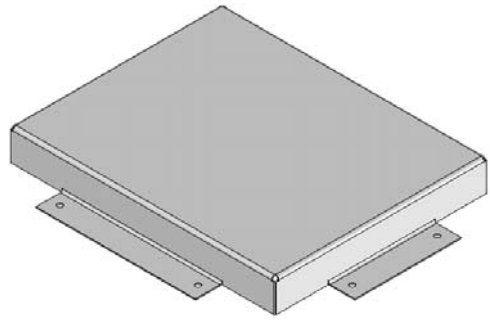



Figure 16-156 Model rolled to the bending stage

Creating Cuts across the Bends

The next feature that you need to create is the cut feature across the bends. For creating this feature, you first need to unfold the sheet metal component.

1. Choose the **Unfold** button from the **Sheet Metal CommandManager** and select the top face of the model as the fixed face. 
2. Choose the **Collect All Bends** button and then choose **OK** from the **Unfold PropertyManager**. The unfolded view of the model is shown in Figure 16-157.
3. Select the top face of the model and invoke the sketching plane. Next, draw the sketch of the cut feature, as shown in Figure 16-158.

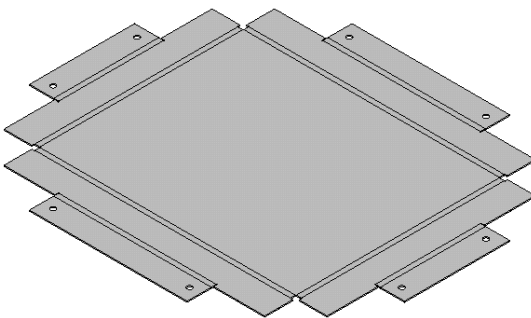


Figure 16-157 Unfolded view of the model

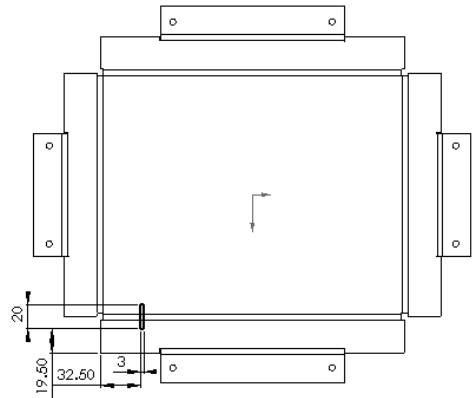


Figure 16-158 Sketch of the cut feature

4. Invoke the **Extruded Cut** tool and create the cut feature. The model after adding the cut feature is shown in Figure 16-159.
5. Pattern the cut feature. The model after patterning the cut feature is shown in Figure 16-160.

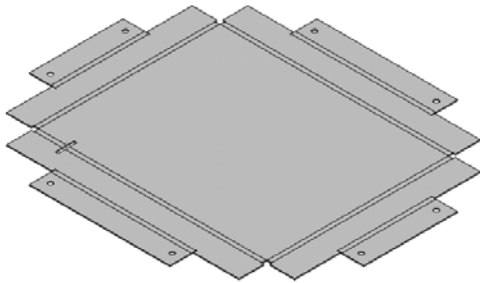


Figure 16-159 Model after adding the cut feature

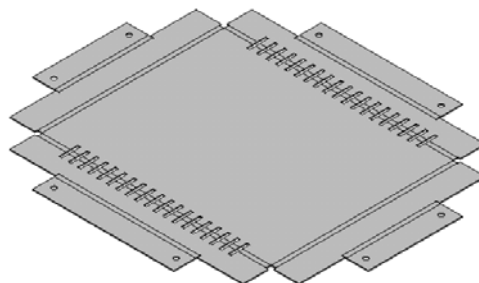



Figure 16-160 Model after patterning the cut feature

Refolding the Sheet Metal Part

After creating the sheet metal component, you need to refold the sheet.

1. Choose the **Fold** button from the **Sheet Metal CommandManager**; the **Fold PropertyManager** is displayed and the top face is selected by default. 
2. Choose the **Collect All Bends** button and then choose **OK**. The sheet metal component after refolding is shown in Figure 16-161.

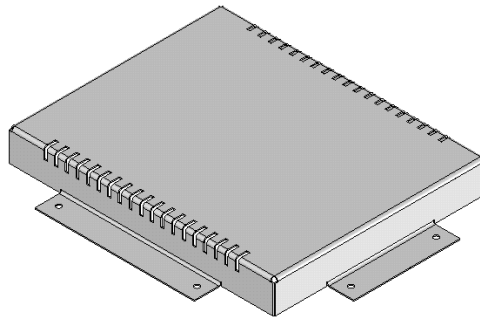


Figure 16-161 Sheet metal component after refolding



Note

You can also create the flat pattern of the sheet metal component as a new configuration. The procedure to add configurations is discussed in the next chapter.

Saving the Model

Next, you need to save the model before generating the drawing views.

1. Choose the **Save** button from the Menu Bar and save the drawing in the location and the

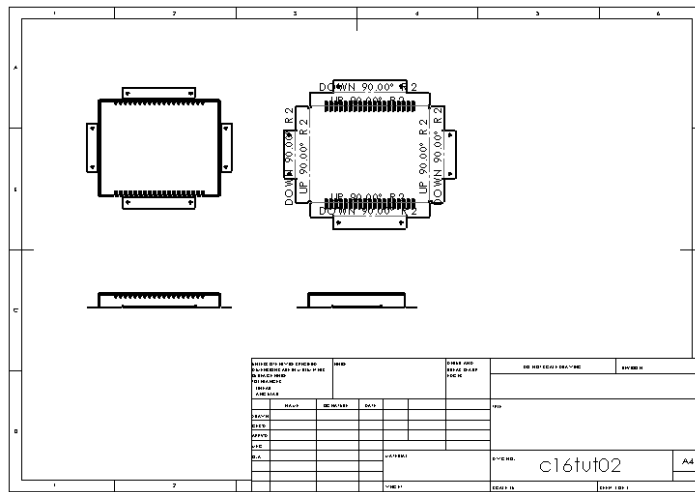


Figure 16-163 Drawing sheet after generating the flat pattern view

Figure 16-163 shows the drawing sheet after generating the drawing view of the flat pattern.

Next, you need to generate the isometric view.

6. Generate an isometric view and place the view close to the top right corner of the drawing sheet. Change the scale of the isometric view to 1:4.

The drawing sheet after generating all the drawing views is displayed in Figure 16-164.

7. Save the drawing file.

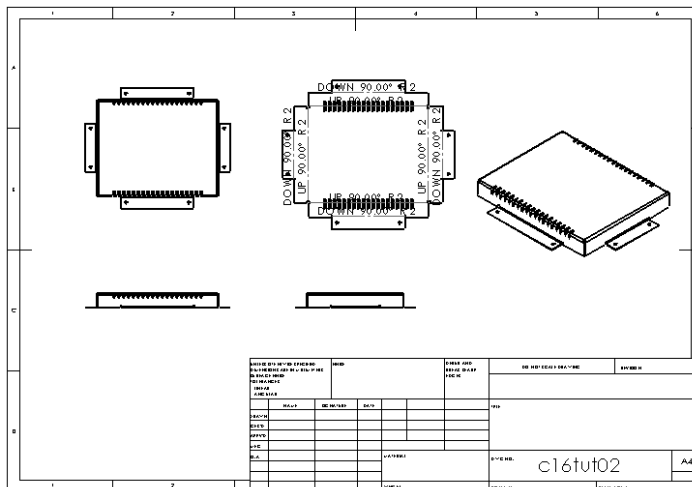


Figure 16-164 Final drawing sheet

SELF-EVALUATION TEST

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

1. In SolidWorks, the _____ button in the **Sheet Metal** toolbar is used to create a base flange.
2. The _____ option in the **Bend Allowance Type** drop-down list is selected to specify the bending allowance using the bend tables.
3. Select the _____ check box in the **View Orientation** rollout to generate the drawing view of the flat pattern of a sheet metal component.
4. The _____ button in the **Sheet Metal** toolbar is used to create hems.
5. The _____ rollout is used to define the bend allowance other than the default bend allowance that you have defined while creating the base flange.
6. To create the flat pattern of a sheet metal component, which is created by converting a part, you need to choose the **Flatten** button. (T/F)
7. If you create a part with sharp edges and convert it into a sheet metal component, then the bends added to the sheet metal component are recognized as round bends. (T/F)
8. To create a closed corner, choose the **Closed Corner** button from the **Features** toolbar. (T/F)
9. The **Rip Gap** spinner in the **Rip Parameters** rollout is used to define the rip distance between two consecutive flanges. (T/F)
10. The **Sketched Bend** tool is used to create a bend by using a sketch as the bending line. (T/F)

REVIEW QUESTIONS

Answer the following questions:

1. The _____ option in the **Bend Allowance Type** drop-down list is used to define the K-Factor.
2. The _____ tool is used to create a series of flanges along the edges of a sheet metal component.
3. The _____ check box is used to set the value of the feature termination according to the thickness of the sheet.

4. While creating a miter flange, the distance value of the rip is modified using the _____ spinner.
5. To create lofted bends, choose the _____ button from the **Sheet Metal** toolbar.
6. Which button in the **Sheet Metal** toolbar is used to fold an unfolded sheet?
 - (a) **Flattened**
 - (b) **Fold**
 - (c) **Bends**
 - (d) **Unfold**
7. Which tool is used to roll back the sheet metal component to the stage when it did not have any bends?
 - (a) **Flattened**
 - (b) **No Bends**
 - (c) **Unfold**
 - (d) **Remove Bends**
8. To unfold a sheet, you need to invoke the _____ tool?
 - (a) **Unfold**
 - (b) **Flattened**
 - (c) **No Bends**
 - (d) **None of these**
9. Which tool is used to draw the bending lines?
 - (a) **Centerline**
 - (b) **Arc**
 - (c) **Spline**
 - (d) **Line**
10. Lofted bends are created between_____.

EXERCISE

Exercise 1

In this exercise, you will create the sheet metal component shown in Figure 16-165. The flat pattern of this model is shown in Figure 16-166. To create this model, first create the base flange. Then, you need to create a miter flange to complete the sheet metal component. The default bend radius is 2 mm, K-Factor is 0.5, and Rectangular Relief ratio is 0.5. Thickness of the sheet is 1 mm. Rip gap for miter flange is 2 mm. Views and dimensions for this model are shown in Figure 16-167.

(Expected time: 20 min)

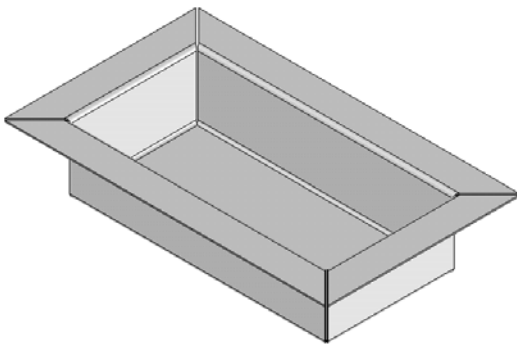


Figure 16-165 Sheet metal component

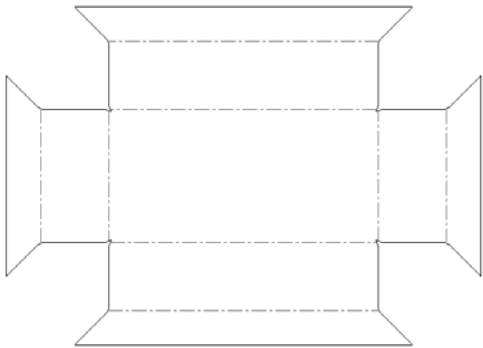


Figure 16-166 Flat pattern of the sheet metal component

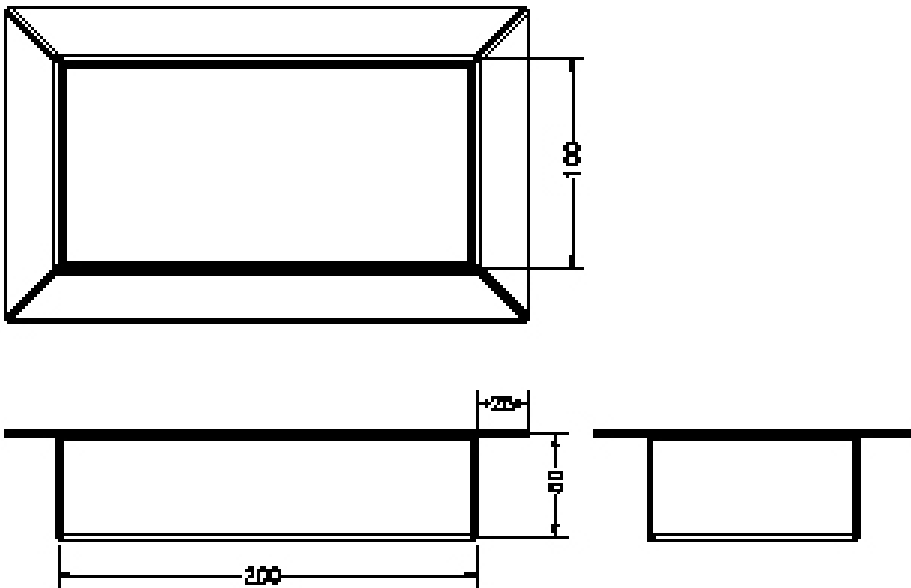


Figure 16-167 Views and dimensions of the sheet metal component for Exercise 1

Answers to Self-Evaluation Test

1. Base-Flange/Tab, 2. Bend Table, 3. Flat Pattern, 4. Hem, 5. Custom Bend Allowance,
6. T, 7. F, 8. F, 9. T, 10. T