

Chapter 14

Sheet Metal Design

Learning Objectives

After completing this chapter, you will be able to:

- *Set the parameters for creating the sheet metal parts*
- *Create the base of the sheet metal part*
- *Add various types of flanges to the sheet metal part*
- *Add a jog to the sheet metal part*
- *Bend or unbend a sheet metal part*
- *Add corner bends to the sheet metal parts*
- *Create dimples, louvers, drawn cutouts, and beads in the sheet metal component*
- *Convert solid parts to sheet metal parts*
- *Create the flat pattern of the sheet metal parts*
- *Renew features*

THE SHEET METAL MODULE

The components having a thickness greater than 0 and less than 12 mm are called sheet metal components. A sheet metal component is created by bending, cutting, or deforming an existing sheet of metal having uniform thickness, refer to Figure 14-1. As it is not possible to machine such a model, therefore, after creating the sheet metal component, you need to flatten it for manufacturing. Figure 14-2 shows the flattened view of the sheet metal component shown in Figure 14-1.

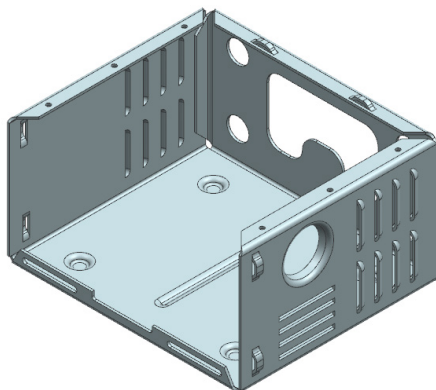


Figure 14-1 Sheet metal component

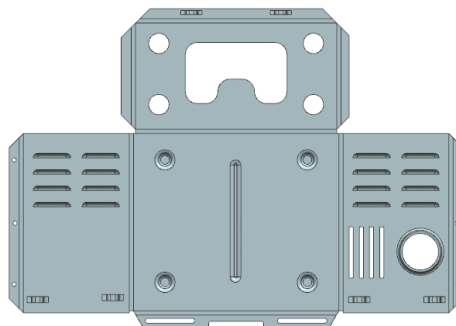


Figure 14-2 Flattened view of the sheet metal component



Note

All features of a sheet metal component cannot be flattened. For example, features such as louvers, dimples cannot be flattened because they are created using a punch and a die.

NX allows you to create the sheet metal components in a specific environment called the Sheet Metal environment. This environment provides all tools that are required for creating the sheet metal components.

To start a new document in the Sheet Metal environment, choose the **New** button from the **Standard** group of the **Home** tab in the **Ribbon**; the **New** dialog box will be displayed. Choose the **Sheet Metal** template from the **Templates** rollout in the **Model** tab, refer to Figure 14-3, and choose the **OK** button; a new document will open in the Sheet Metal environment, refer to Figure 14-4. In this environment, the different groups provide the tools that are required to create the sheet metal components.

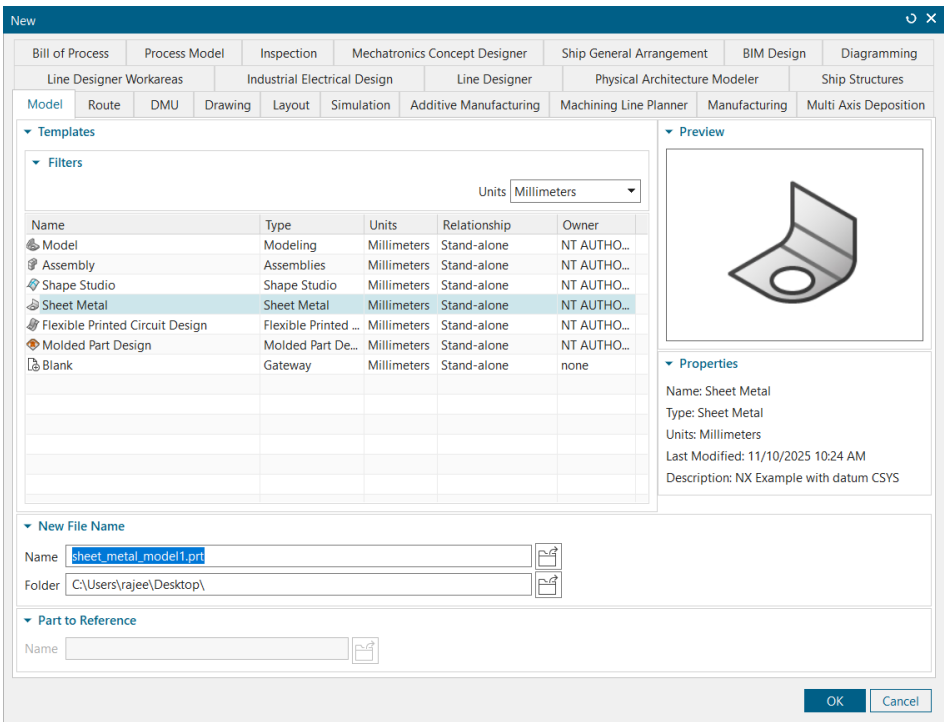


Figure 14-3 Starting a new sheet metal document

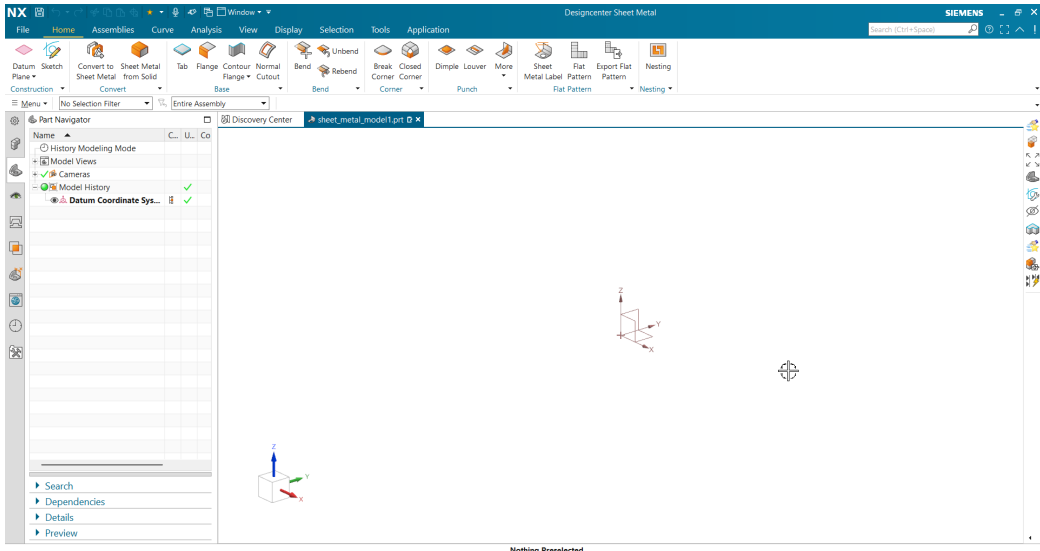


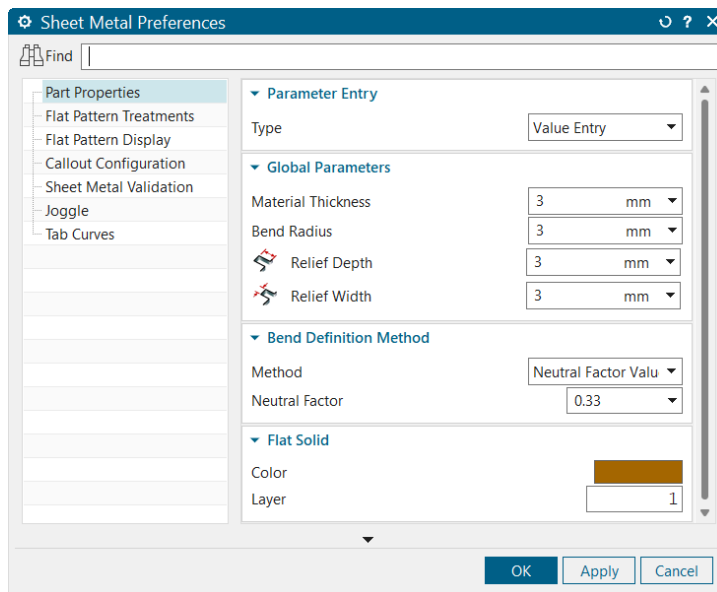
Figure 14-4 Default screen of the Sheet Metal environment

SETTING THE SHEET METAL PART PROPERTIES

Ribbon: File > Preferences > Sheet Metal
Menu: Preferences > Sheet Metal

Before proceeding with the process of creating the base feature of a sheet metal part, it is recommended that you set the parameters related to it. These parameters will determine the default values of the sheet thickness, bend radius, relief depth, and relief width that will be displayed while using various tools.

To set these parameters, choose **Menu > Preferences > Sheet Metal** from the **Top Border Bar**; the **Sheet Metal Preferences** dialog box will be displayed with the **Part Properties** tab chosen by default. This tab displays the parameters related to the sheet metal part, refer to Figure 14-5. The parameters available in the **Part Properties** tab are discussed next.



*Figure 14-5 The **Part Properties** tab of the **Sheet Metal Preferences** dialog box*

Parameter Entry Area

This area is used to specify the methods that will be used for defining the parameters of the sheet metal part. The options in the **Type** drop-down list of this area are discussed next.

Value Entry

On selecting this option, you can define the parameters related to sheet metal by using the options in the **Global Parameters** and **Bend Definition Method** areas. The options in these areas will be discussed later in this chapter.

Material Selection

Using this option, you can define the parameters of the sheet metal by selecting a predefined material available for the sheet metal part. On selecting this option, the **Select Material** button

will be activated. Choose this button to invoke the **Select Material** dialog box, refer to Figure 14-6. Select a material from the **Available Materials** rollout at the bottom of the dialog box or you can select the desired parameters using the **Material Characteristics** rollout.

Tool ID Selection

Using this option, you can define the sheet metal parameters by specifying the predefined tool ID. When you select the **Tool ID Selection** option, the **Bend Definition Method** area will be modified and the **Punch** and **Die** drop-down lists will become available in it, refer to Figure 14-7. Using the options available in these drop-down lists, you can select the punch and die required for the operation.

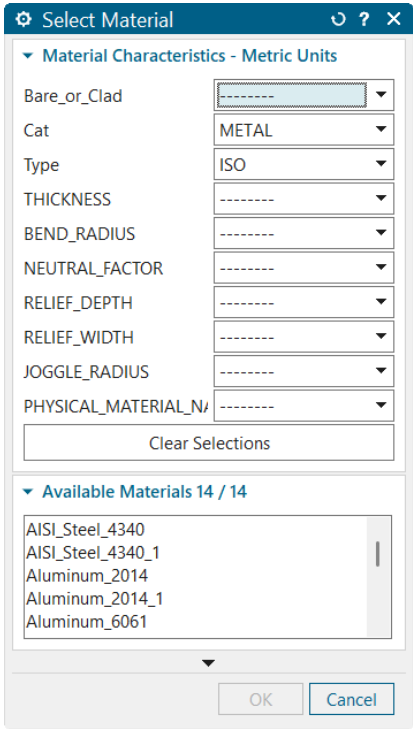


Figure 14-6 The Select Material dialog box



Figure 14-7 The modified Bend Definition Method area

Global Parameters Area

This area contains the options to specify the default values of the parameters for the sheet metal part. The options in this area are discussed next.

Material Thickness

This edit box is used to set the default thickness for the sheet metal part. The thickness specified in this edit box will be displayed as the default thickness whenever you invoke a tool to create the sheet metal part.

**Note**

While creating a sheet metal part, it is not necessary to accept the default sheet thickness. You can modify the default thickness as per your requirement.

Bend Radius

This edit box is used to set the default value for the bend radius. This value is used while adding flanges to the model or bending it. Figure 14-8 shows a sheet bend having a radius of 1 mm and Figure 14-9 shows a sheet bend having a radius of 5 mm.

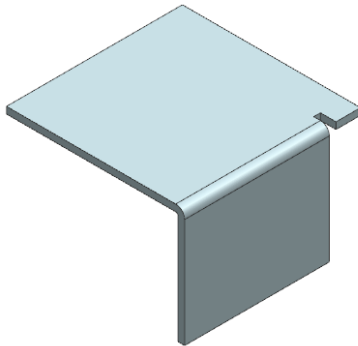


Figure 14-8 Sheet with 1 mm bend radius

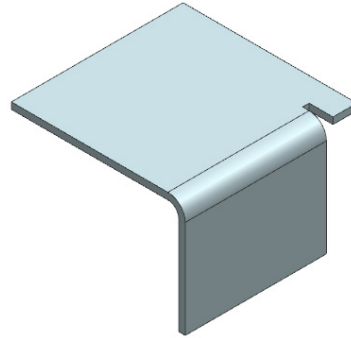


Figure 14-9 Sheet with 5 mm bend radius

Relief Depth

Whenever you bend a sheet metal component or create a flange such that the bend does not extend throughout the length of the edge, a groove is added at the end of the bend so that the walls of the sheet metal part do not intersect when folded or unfolded. This groove is known as relief. You can set a predefined depth for the relief using the **Relief Depth** edit box. Figure 14-10 shows a sheet metal part with relief depth of 2 mm and Figure 14-11 shows a sheet metal part with relief depth of 5 mm.

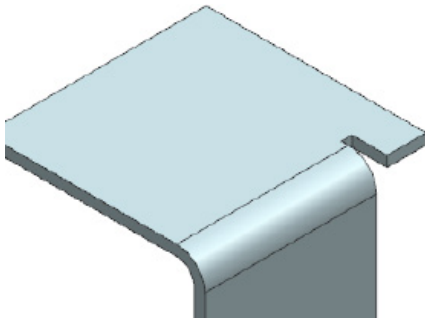


Figure 14-10 Relief depth = 2 mm

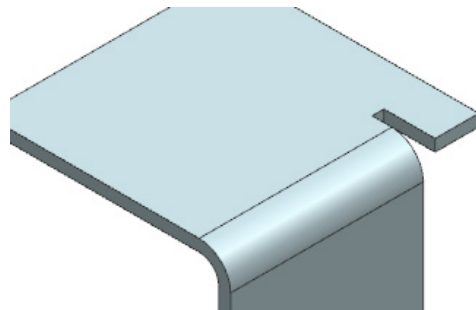


Figure 14-11 Relief depth = 5 mm

Relief Width

The **Relief Width** edit box is used to specify the value of width of the relief. The default value of the relief width is equal to the thickness of the sheet. Figure 14-12 shows a sheet metal component with a relief width of 1 mm and Figure 14-13 shows a sheet metal component with a relief width of 5 mm.

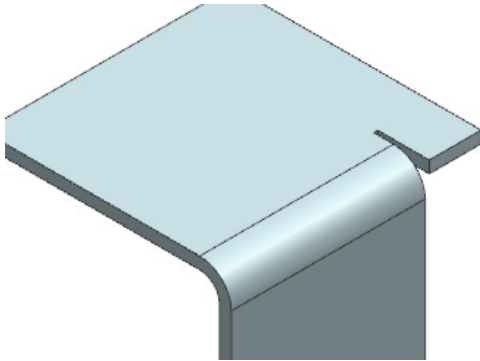


Figure 14-12 Relief width = 1 mm

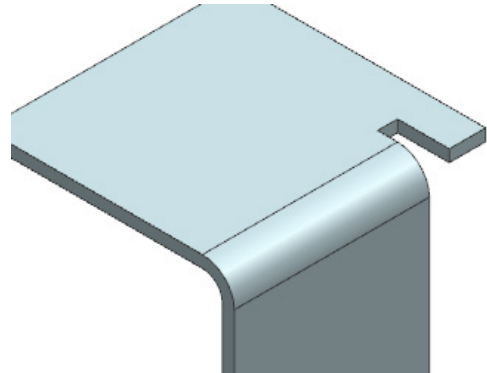


Figure 14-13 Relief width = 5 mm

Bend Definition Method Area

The options available in this area change depending upon the option selected from the **Type** drop-down list in the **Parameter Entry** area.

On selecting the **Value Entry** option from the **Type** drop-down list, the **Method** and **Neutral Factor** drop-down lists will be available in the **Bend Definition Method** area. The options in the **Method** drop-down list are discussed next.

Neutral Factor Value

On selecting this option, you can enter a value in the **Neutral Factor** edit box. The neutral factor depends on the properties of the material being bent. While bending, the outer surface of the bend is under tension and the inner surface is under compression. There is a layer in between them which is neither in compression nor in tension. The ratio of the distance of this surface from the inner bend to the thickness is the neutral factor. The default value is 0.33 and the value typically ranges from 0 to 1.

Bend Table

On selecting this option, the **Table** drop-down list will become available below the **Method** drop-down list. The options in this drop-down list are used to calculate the neutral factor. You can select the required option from this drop-down list.

Formula

This option is used to define a formula to calculate the bend allowance of the sheet metal part. On choosing this option, you can enter the bend allowance formula in the edit box available below the **Formula** drop-down list.

After performing the required settings, choose the **Apply** button to apply settings to the model and then choose the **OK** button to exit the dialog box.



Note

*You can change the values of parameters in the **Sheet Metal Preferences** dialog box any time during the designing process.*

CREATING THE BASE FEATURE

Ribbon: Home > Base > Tab
Menu: Insert > Tab



Tab

To create a sheet metal component, first you need to create a base feature. To create the base feature of the sheet metal component, choose the **Tab** tool from the **Base** group of the **Home** tab; the **Tab** dialog box will be displayed, as shown in Figure 14-14. Also, you will be prompted to select the sketch plane to sketch or the section geometry to be extruded.

If you select the sketch at this stage, the preview of the tab feature created using the default values will be displayed on the screen. Select the sketch plane, the sketching environment will be invoked.

Draw the sketch of the base feature of the sheet metal component and then exit the sketching environment; the preview of the tab feature will be displayed along with the Thickness handle, as shown in Figure 14-15(a). Note that no matter how long you drag the handle, the tab will be created based on the thickness specified in the **Sheet Metal Preferences** dialog box. You can choose the **Reverse Direction** button to reverse the direction of feature creation. After specifying all the settings, choose the **OK** button from the **Tab** dialog box; the base of the sheet metal is created.

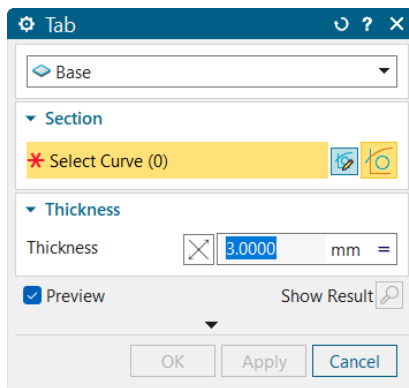


Figure 14-14 The **Tab** dialog box

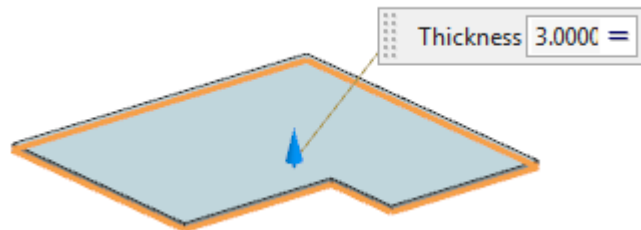


Figure 14-15(a) Preview of the tab feature

After creating the base feature, if you invoke the **Tab** dialog box again; the **Secondary** option will appear in the Type drop-down list. Using the **Secondary** option, you can create a planar feature attached to the base feature. Note that this option will be available only if you have created a

sheet metal feature earlier. Figure 14-15(b) shows a base feature and a sketch which will be used to create the secondary feature. Figure 14-15(c) shows the secondary feature created and attached to the base feature.

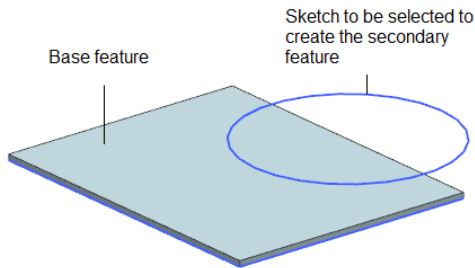


Figure 14-15(b) Base feature and the sketch to be selected

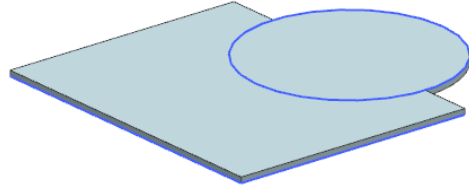


Figure 14-15(c) The resultant feature



Tip

To enter a new thickness value of the sheet, click on the **Launch the formula editor** button which is located next to the **Thickness** edit box; a shortcut menu will be displayed. Choose the **Use Local Value** option from the shortcut menu and enter a new thickness value in the **Thickness** edit box.



Note

In NX, you can also create the base feature by using the **Contour Flange** and **Lofted Flange** tools. You will learn more about these tools later in this chapter.

ADDING FLANGES TO A SHEET METAL PART

Ribbon: Home > Base > Flange
Menu: Insert > Bend > Flange



Flange

Flange is the bend section of the sheet metal. NX allows you to directly add a folded face to the existing sheet metal part by using the **Flange** tool. To do so, choose the **Flange** tool from the **Base** group of the **Home** tab, refer to Figure 14-16; the **Flange** dialog box will be displayed, as shown in Figure 14-17. The options in the **Flange** dialog box are discussed next.



Figure 14-16 The **Base** group

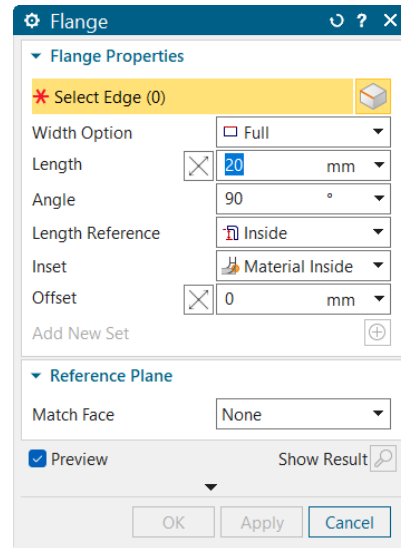


Figure 14-17 The **Flange** dialog box

Flange Properties Rollout

By default, the **Select Edge** area is highlighted in this rollout. As a result, you are prompted to select a linear edge. Select an edge from the base feature of the sheet metal part; preview of the flange will be displayed. The other options available in this rollout are discussed next.

Width Option

The options in this drop-down list are used to define width and position of the flange on the selected edge. These options are discussed next.

Full

This option is selected by default and is used to create the flange having width equal to the width of edge selected to create the flange.

At Center

This option is used to create the flange at an equal distance from the center of the selected edge. After selecting this option, the flange will be displayed at the center of the edge selected. To modify the width of the flange, specify the required value in the **Width** edit box.

At End

This option is used to create the flange at one of the ends of the selected edge. When you select this option, the **Specify Point** area will become available below the **Width Option** drop-down list. Next, click in the **Specify Point** area; you will be prompted to select the desired end of the edge. Select one of the endpoints of the edge; the preview of the flange will be displayed. Specify a value in the **Width** edit box to modify the width of the flange.

From End

This option is used to create the flange at a certain offset from one of the endpoints of the selected edge. When you select this option, the **Specify Point** area will be displayed below the **Width Option** drop-down list. Next, click in the **Specify Point** area; you will be prompted to select the desired end. After you select the desired end, the flange will be created with default values of its distance from the selected end and width. You can specify required values in the **Distance 1** and **Width** edit boxes to modify these values.

From Both Ends

This option is used to create the flange at a certain offset from both ends of the selected edge. After you select this option, the **Distance 1** and **Distance 2** edit boxes are displayed below the **Width Option** drop-down list. You can specify required values in these edit boxes to modify the distance from both the ends of the selected edge.

Length

This edit box is used to specify the length of the flange. This value is modified dynamically as you drag the Length handle displayed in the drawing window, refer to Figure 14-18.

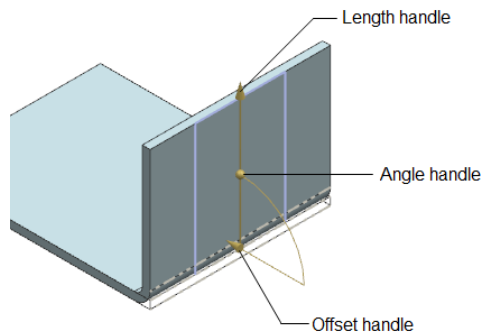


Figure 14-18 Handles displayed on the flange

Reverse Direction

This button is used to reverse the direction of the flange.

Angle

This edit box is used to specify the angle of the flange. The default value is 90-degree. You can enter the desired value in this edit box. You can also modify the angle dynamically by using the Angle handle, refer to Figure 14-18. Note that this edit box will be available only when you select the **None** option from the **Match Face** drop-down list in the **Reference Plane** rollout.

Length Reference

The options in this drop-down list are used to define the reference from which the length of the flange will be measured. These options are discussed next.

Inside

This option is selected by default and is used to specify the length of the flange from the virtual intersection of the tab faces that are adjacent to the inner bend face to the top of the flange.

Outside

This option is selected to specify the length of the flange from the virtual intersection of the tab faces that are adjacent to the outer bend face to the top of the flange.

Web

This option is used to specify the length of the flange from the starting point of the bend to the extreme edge of the flange created.

Tangent

This option is used to specify the length of the flange according to the DIN6935 standard.

Inset

The options in this drop-down list are used to specify material condition of the flange. These options are discussed next.

Material Inside

This option is selected by default and creates the flange such that the material is added inside the profile of the flange. The profile is automatically displayed when you place the flange.

Material Outside

Select this option to add the material outside the profile of the flange. You can also select this option after placing the flange.

Bend Outside

If this option is selected, the bend of the flange will be placed outside the profile of the flange.

Material Inside OML

If this option is selected, the bend of the flange sets into the base material so that outside face of the flange gets imaginary intersected with the opposite side of the selected edge.

Offset

This edit box is used to create a flange at an offset value from the selected edge. To do so, specify a value in the **Offset** edit box. You can reverse the offset direction by choosing the **Reverse Direction** button.

Geometry Properties

The options in this sub-rollout are used to replicate the geometry irrespective of the feature parameters. These options are discussed next.

Use Geometry Mirror and Pattern

On selecting this check box, the geometry is replicated irrespective of the feature parameters. However, if you clear this check box, the geometry is replicated considering the feature parameters.

Miter

On selecting this check box, miter is applied to remove interferences between adjacent edges in a given set. However, if you clear this check box, the adjacent flanges are created with interference between them.

Bend Parameters

The options in this sub-rollout are used to specify the bend parameters of the flange. These options are discussed next.

Bend Radius

This edit box is used to specify the bend radius. By default, the value that was set using the **Part Properties** tab of the **Sheet Metal Preferences** dialog box will be used as the bend radius. If you want to modify this value, click on the **Launch the formula editor** button located next to this edit box and then choose the **Use Local Value** option from the shortcut menu displayed. Next, enter the required value in the **Bend Radius** edit box.

Neutral Factor

This edit box is used to specify the neutral factor value. By default, the value that was set using the **Part Properties** tab of the **Sheet Metal Preferences** dialog box will be used as the neutral factor.

Relief

Whenever you bend a sheet metal component or create a flange such that bend does not extend throughout the length of the edge, a groove is added at the end of the bend such that the walls of the sheet metal part do not intersect when folded or unfolded. This groove is known as relief. Expand the **Relief** sub-rollout to define the type of relief. The options in this sub-rollout are discussed next.

Bend Relief

This drop-down list is used to specify the type of relief to be added. You can add a square shape relief or a round shape relief by selecting the **Square** or **Round** option, respectively. If you choose the **None** option, no relief will be created. You can use the default value of the relief depth and width or enter a new value in the corresponding edit boxes. Figure 14-19 shows a sheet metal part with a square relief and Figure 14-20 shows a sheet metal part with a round relief.

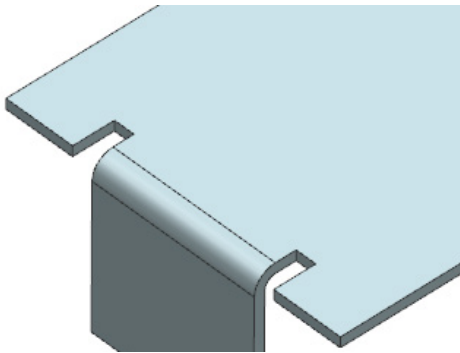


Figure 14-19 Part with square relief

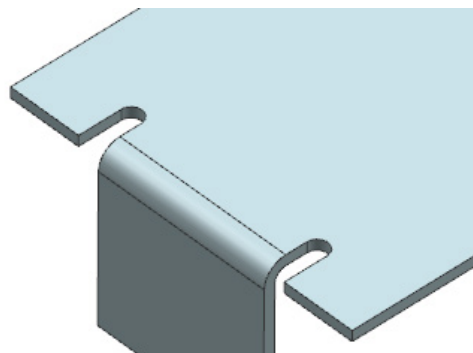
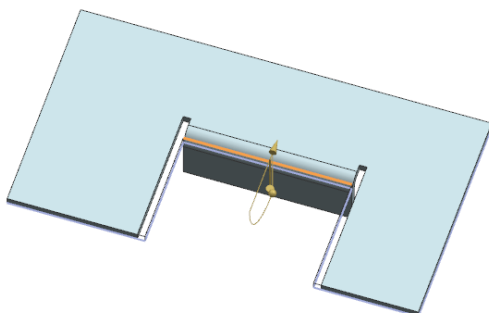


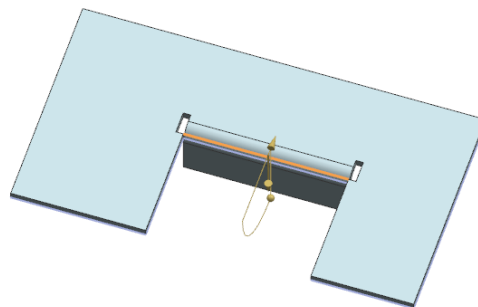
Figure 14-20 Part with round relief

Extend Relief

Select this check box to extend the relief to the edge selected to create the flange. If this check box is cleared, the relief will be applied only to the portion that is adjacent to the flange that you are creating. Figures 14-21 and 14-22 show flanges created with the **Extend Relief** check box selected and cleared, respectively.



*Figure 14-21 Flange created with the **Extend Relief** check box selected*



*Figure 14-22 Flange created with the **Extend Relief** check box cleared*

Corner Relief

This drop-down list is used to add a corner relief to the flange. A corner relief is added when the flange termination forms a corner with another flange. If you select the **None** option from this drop-down list no corner relief will be added between two flanges, refer to Figure 14-23. You can add three different types of corner reliefs by selecting their corresponding options from this drop-down list. Figure 14-24 shows the **Bend Only** corner relief. Note that in this figure the corner relief is added to the existing flange and not to the one that you are creating.

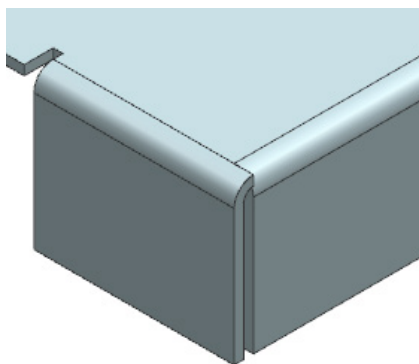
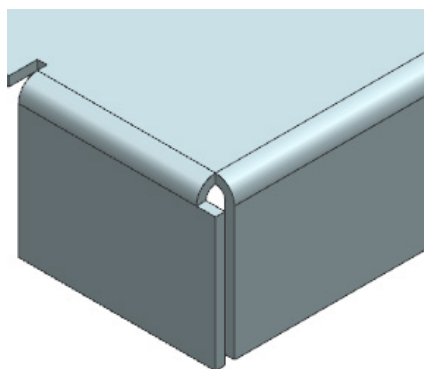


Figure 14-23 Flanges with no corner relief



*Figure 14-24 Flanges with the **Bend Only** corner relief*

The third type of corner relief, **Bend/Face Chain**, is used when the first flange has a chain of faces. In this case, the corner relief is applied to the face chain. Figure 14-25 shows a sheet metal part in which the first flange forms a chain with another flange. In this figure, the **Bend/Face** corner relief has been added. Figure 14-26 shows the same model after adding the **Bend/Face Chain** corner relief.

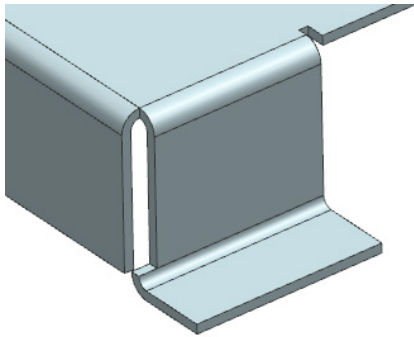


Figure 14-25 Flanges with the **Bend/Face** corner relief

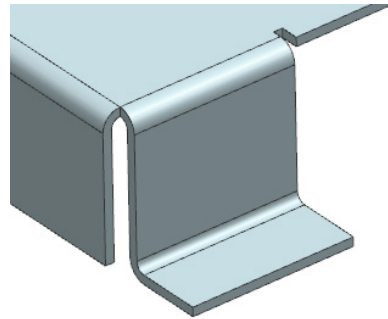


Figure 14-26 Flanges with the **Bend/Face Chain** corner relief

Reference Plane Rollout

The **Match Face** drop-down list available in this rollout is used to specify a face or plane on which the outer face of the flange will coincide. To do so, select the **Until Selected** option from this drop-down list; you will be prompted to select a plane or face. Select a face or plane from the drawing window, refer to Figure 14-27; the flange will be created coinciding with the selected face, refer to Figure 14-28.

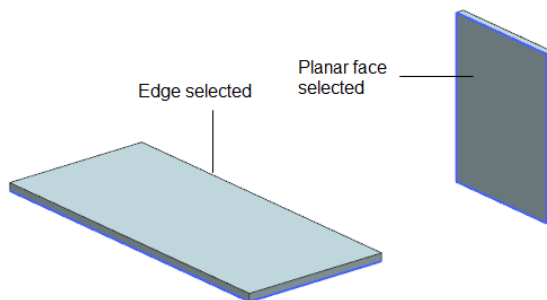


Figure 14-27 Face selected to match the outer face of the flange

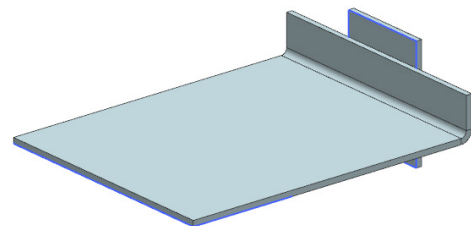
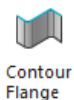


Figure 14-28 Flange created coinciding with the selected face

CREATING CONTOUR FLANGES

Ribbon: Home > Base > Flange Drop-down > Contour Flange
Menu: Insert > Bend > Contour Flange



Contour Flange

The flanges that are created by using an open sketched shape are called contour flanges. To create a contour flange, choose the **Contour Flange** tool from the **Flange** Drop-down of the **Base** group; the **Contour Flange** dialog box will be displayed, as shown in Figure 14-29. Next, select the edge at which the normal plane will be placed to create the profile of the contour flange; the preview of the plane and the **Create Sketch** dialog box will be displayed. Specify the location of the plane and choose the **OK** button from the **Create Sketch** dialog box; the sketching environment will be invoked. Now, in this environment, you can draw the profile of the contour flange using lines and arcs. Note that you cannot use splines in the sketch.

**Note**

*If you are creating a contour flange as the base feature, you have to select the **Base** option from the **Type** drop-down list. Next, you need to select a sketch plane to invoke the **Sketch** in **Task** environment.*

After drawing the open sketch for the contour flange, exit the sketch environment; the preview of the contour flange will be displayed. Next, use the options in the **Width** rollout of the **Contour Flange** dialog box to define the flange width. The options in the **Width** rollout are discussed next.

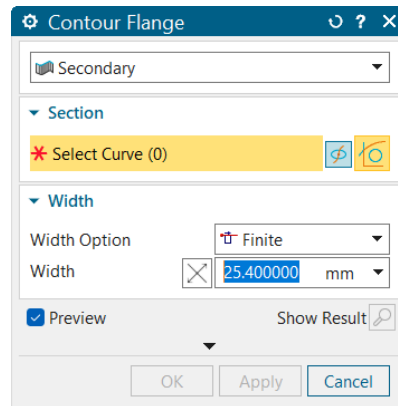


Figure 14-29 The *Contour Flange* dialog box

Width Rollout

The options in the **Width Option** drop-down list of this rollout are used to specify the method for calculating the width of the flange. These options are discussed next.

Finite



This option is used to specify the width of the contour flange using a finite value. This option is selected by default. As a result, the **Width** edit box is displayed in which you can enter the value. Figure 14-30 shows the preview of a contour flange being created. This figure also shows the open profile for the contour flange.

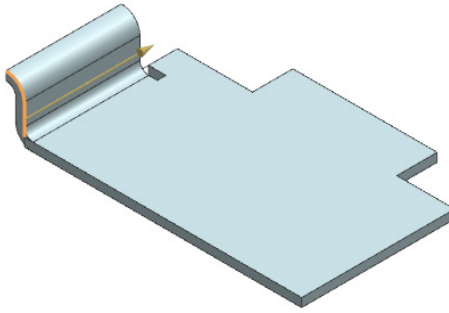


Figure 14-30 Preview of a contour flange being created

Symmetric



The **Symmetric** option is used to create a contour flange with the symmetric extent.

To End



This option is used to terminate the contour flange at the end of the selected edge.

Chain



This option is used to select a chain of edges on which the contour flange will be created. Figure 14-31 shows the base of the sheet metal component and the sketch to be used to create the contour flange. Figure 14-32 shows the contour flange created by selecting all four edges on the top face of the base.

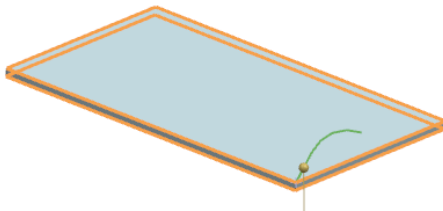


Figure 14-31 Sketch for the contour flange

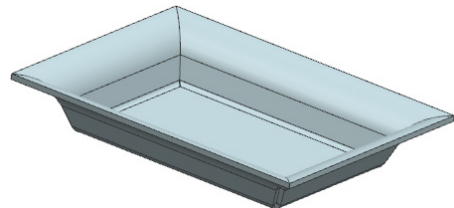


Figure 14-32 The contour flange created by selecting all four edges on the top face

CREATING LOFTED FLANGES

Ribbon: Home > Base > Flange Drop-down > Lofted Flange
Menu: Insert > Bend > Lofted Flange



You can create a sheet metal part between two selected profiles using the **Lofted Flange** tool. Note that the profiles being used should be open and they may or may not have the same number of elements. When you invoke this tool, the **Lofted Flange** dialog box will be displayed, as shown in Figure 14-33. Also, you will be prompted to select a plane to sketch or select a section. Select the sketching plane and then draw the first profile. Exit the sketch

environment; the **Sketch End Section** button will be activated in the **End Section** rollout. Create a parallel plane and draw the second profile. After drawing the sketch of the second profile, as soon as you exit from the sketch environment, the preview of the lofted flange will be displayed. Choose the **OK** button from the **Lofted Flange** dialog box; the lofted flange will be created. Figure 14-34 shows the sketches for creating the lofted flange. Figure 14-35 shows the preview of the resulting lofted flange.

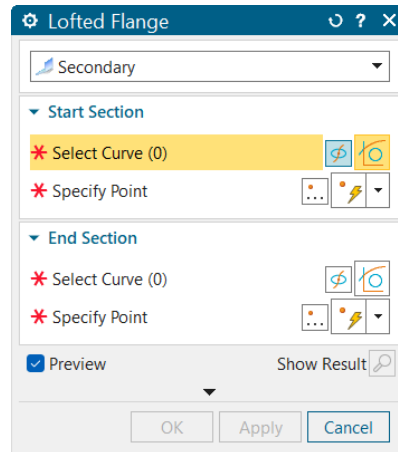


Figure 14-29 The *Lofted Flange* dialog box

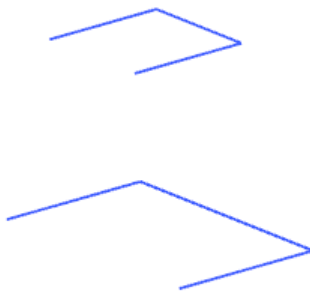


Figure 14-34 Sketches to be used to create the lofted flange

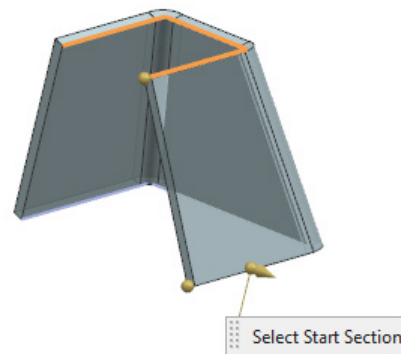


Figure 14-35 Preview of the resulting lofted flange

ADDING A JOG TO THE SHEET

Ribbon: Home > Bend > More Gallery > Bend Gallery > Jog (*Customize to Add*)
Menu: Insert > Bend > Jog



The **Jog** tool is used to add a jog to an existing sheet metal part using a sketched line segment. Figure 14-36 shows the base of a sheet metal part and the line to be used to add the jog. Figure 14-37 shows the sheet after adding the jog.

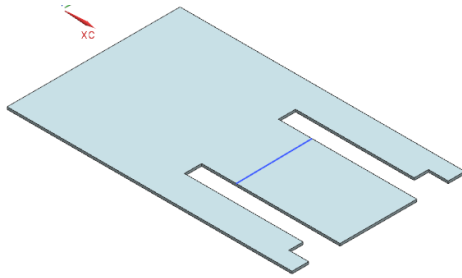


Figure 14-36 Base sheet and the line to add the jog

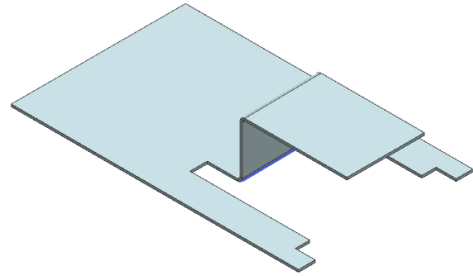


Figure 14-37 Sheet after adding the jog

To add a jog, invoke the **Jog** tool; the **Jog** dialog box will be displayed. Next, select the plane or the face of the base feature to draw the line segment for adding the jog. Draw a single line that will define the jog line on the base sheet and then exit the sketching environment; the default preview of the jog will be displayed. You can reverse the side and direction of the sheet on which the jog will be added by using the **Reverse Side** and **Reverse Direction** buttons. Next, you need to define the height of the jog by entering a value in the **Height** edit box of the **Jog Properties** rollout. Alternatively, you can drag the Height handle to specify the height of the jog. Next, set all the other options in the **Jog** dialog box. These options have been discussed earlier in the chapter. Choose the **OK** button from the **Jog** dialog box to create the jog.

BENDING THE SHEET METAL PART

Ribbon: Home > Bend > Bend
Menu: Insert > Bend > Bend



The **Bend** tool is used to bend an existing sheet metal part using a sketched line segment. To bend a sheet, invoke the **Bend** tool; the **Bend** dialog box will be displayed. Next, select the plane or the face of a feature to draw the line segment for adding a bend. Draw a single line that will define the bend line on the selected feature and then exit the sketching environment; the default preview of the bend will be displayed. You can use the **Reverse Side** button to reverse the side of the bend. The **Reverse Direction** button can be used to reverse the direction of the bend creation. Next, specify the angle of the bend by dragging the Angle handle or specifying a value in the **Angle** edit box of the **Bend Properties** rollout. You will notice that the **Outer Mold Line Profile** option is selected in the **Inset** drop-down list. Therefore, the sheet is bent outside the bend line. If you select the **Bend Center Line Profile** option, the sheet will be bent equally on both sides of the bend line. The **Inner Mold Line Profile** option is used to bend the sheet inside the bend line. The other options in the **Bend** dialog box are same as those discussed earlier. After setting all the parameters, choose the **OK** button from the **Bend** dialog box. Figure 14-38 shows a sheet metal part and the line to be used to bend it. Figure 14-39 shows the sheet after bending.

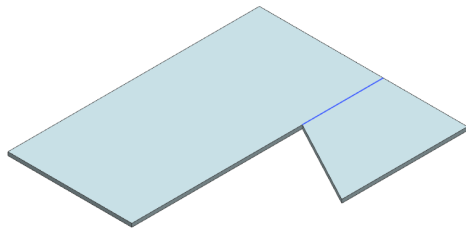


Figure 14-38 Base sheet and the line to bend the sheet

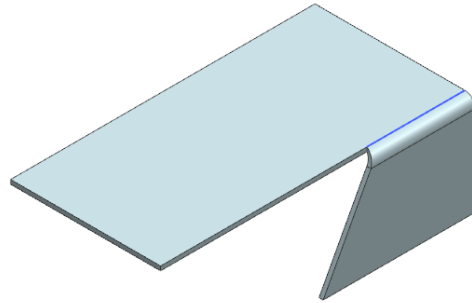


Figure 14-39 Sheet after bending

UNBENDING THE SHEET METAL PART

Ribbon: Home > Bend > Unbend
Menu: Insert > Form > Unbend



The **Unbend** tool is used to unbend the portion of the sheet which has been bent using the **Bend** or **Flange** tool. This tool is highly useful when you want to create a feature on the bent portion of the sheet. On choosing this tool, the **Unbend** dialog box will be displayed and you will be prompted to select a planar face or a linear edge to remain fixed while the part is unbent. Select the face to be fixed; you will be prompted to select bends. Select the rounds of the bends or flanges. Next, choose the **OK** button to complete the unbending of sheet.

REBENDING THE SHEET METAL PART

Ribbon: Home > Bend > Rebend
Menu: Insert > Form > Rebend



The **Rebend** tool is used to rebend the portion of the sheet that was unbent using the **Unbend** tool. When you invoke this tool, the **Rebend** dialog box will be displayed and you will be prompted to select the unbend faces. Move the cursor over the location where the bend was placed originally; the bend will be highlighted and then select it. Next, choose the **OK** button to exit the **Rebend** tool. You can also make the face stationary while rebending the sheet. To do so, choose the **Stationary Face or Edge** button from the **Stationary Face or Edge** rollout; you will be prompted to select the planar face or linear edge to remain fixed. Select the face to make it stationary.

Figure 14-40 shows an unbent sheet with the cut features created at the bends and Figure 14-41 shows the sheet after rebending.

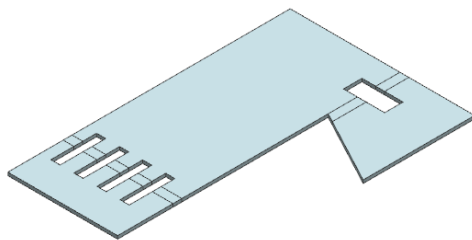


Figure 14-40 Cut features created on the unbent sheet metal part

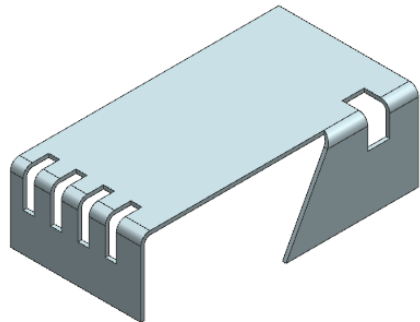


Figure 14-41 Model after creating the cuts and then rebending

FILLETING OR CHAMFERING CORNERS

Ribbon: Home > Corner > Break Corner
Menu: Insert > Corner > Break Corner

 The **Break Corner** tool is used to add fillets or chamfers to the selected corners of the sheet metal part. When you invoke this tool, the **Break Corner** dialog box will be displayed and you will be prompted to select the edges to break or select a face to break all thickness edges. Select the edges that you want to fillet or chamfer. By default, the **Blend** option is selected in the **Method** drop-down list of the **Break Properties** rollout of the **Break Corner** dialog box. As a result, a fillet is added to the model. If you want to add a chamfer, select the **Chamfer** option from the **Method** drop-down list. It creates a 45-degree chamfer. You can enter the fillet or chamfer value in the **Radius** or **Distance** edit box, respectively.

CLOSING THE CORNERS OF A SHEET METAL PART

Ribbon: Home > Corner > Closed Corner
Menu: Insert > Corner > Closed Corner

 The **Closed Corner** tool is used to close the corner created by two bends. When you invoke this tool, the **Closed Corner** dialog box will be displayed, as shown in Figure 14-42. Also, you will be prompted to click on the bends to be modified. Select the curved portion of the two bends to treat. The options of the **Closed Corner** dialog box are discussed next.

Type Drop-down List

Using the drop-down list, you can select whether to apply only relief to the corner or a relief with a closed

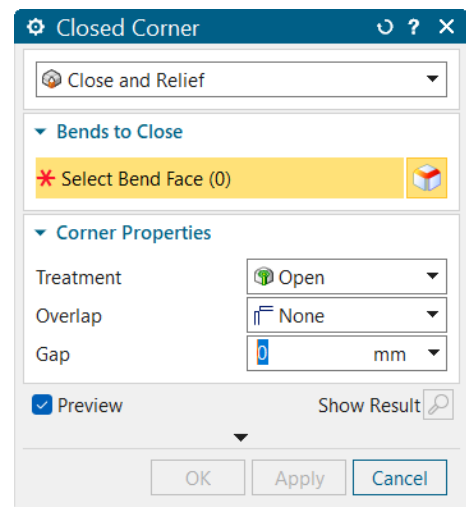


Figure 14-42 The **Closed Corner** dialog box

treatment. Select the **Close and Relief** option, if not selected by default, to apply the closed treatment along with the relief. To apply only relief, select the **Relief** option.

Bends to Close Rollout

By default, the **Select Adjacent Bends** area is highlighted in this rollout. As a result, you are prompted to select the bend face adjacent to a corner.

Corner Properties Rollout

The options in this rollout are discussed next.

Treatment

The options in this drop-down list are used to specify the type of corner treatment and are discussed next.

Open

On selecting this option, the walls are closed but the corner remains open. Figure 14-43 shows the corner after the open corner treatment.

Closed

On selecting this option, the corners are also closed along with the walls, as shown in Figure 14-44.

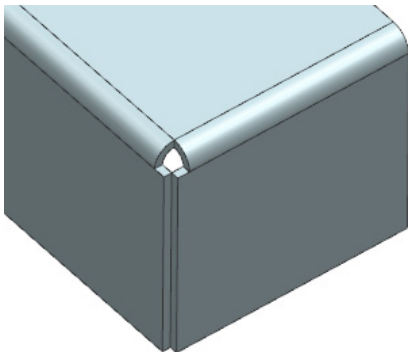


Figure 14-43 Model after performing the open corner treatment of two bends

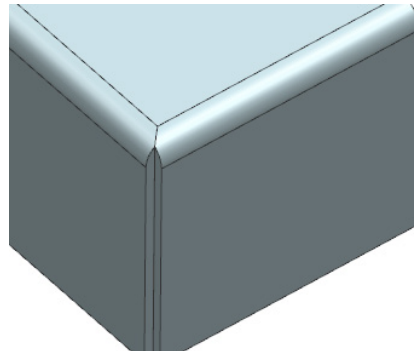


Figure 14-44 Model after performing the closed corner treatment

Circular Cutout

On selecting this option, a circular cutout is created at the corner, as shown in Figure 14-45. You can enter the diameter of the circle in the **Diameter** edit box of the **Relief Properties** rollout.

U-Cutout

On selecting this option, a U-shaped cutout is created at the corner, as shown in Figure 14-46.

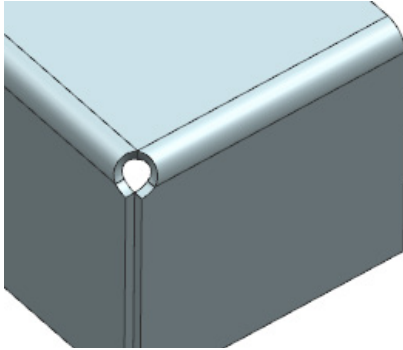


Figure 14-45 Model after performing the circular cutout corner treatment of the two bends

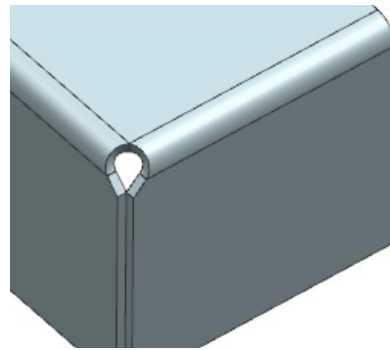


Figure 14-46 Model after performing the U-shaped cutout corner treatment of the two bends

V-Cutout

On selecting this option, a V-shaped cutout is created at the corner, as shown in Figure 14-47. You can specify the angles of the V-shape in the **Angle 1** and **Angle 2** edit boxes available in the **Relief Properties** rollout.

Rectangular Cutout

On selecting this option, a rectangular cutout is created at the corner, as shown in Figure 14-48. You can enter the length and width of the square in the respective edit boxes available in the **Relief Properties** rollout.

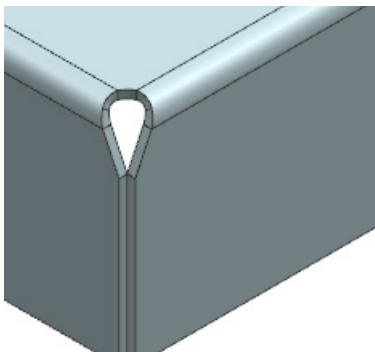


Figure 14-47 Model after performing the V-shaped cutout corner treatment

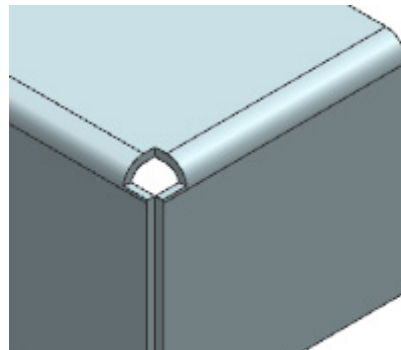


Figure 14-48 Model after performing the rectangular cutout corner treatment

Overlap

This drop-down list is used to specify the walls to be closed or overlapped. By default, the **Closed** option is selected. As a result, the walls are closed. You can overlap one wall on the other by selecting the **Overlapping** option from this drop-down list. Figure 14-49 shows the model with an overlap.

Gap

You can add a gap to the corner treatment by entering a value in this edit box. Figure 14-50 shows the model after performing the open corner treatment with a gap of 2 mm.

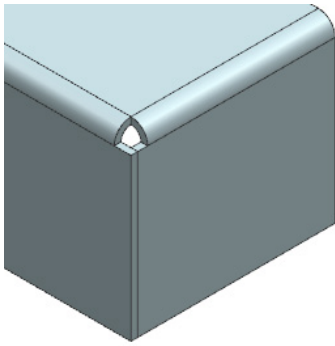


Figure 14-49 *Overlapping of walls in the corner treatment*

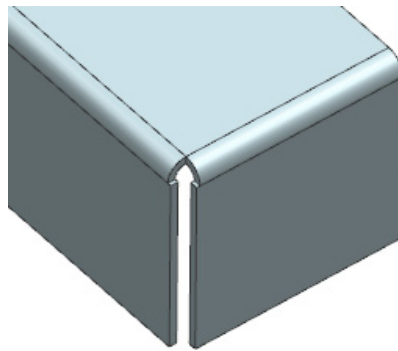


Figure 14-50 *Model after performing the open corner treatment with a gap of 2 mm*



Note

The gap value cannot be negative or greater than sheet thickness.

Miter Corner

On selecting this check box, a miter is created at the corner. Figures 14-51 and 14-52 show the model after performing the closed corner treatment with the **Miter Corner** check box selected and cleared.

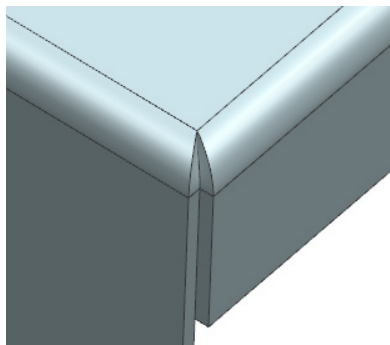


Figure 14-51 *The closed corner treatment with the **Miter Corner** check box selected*

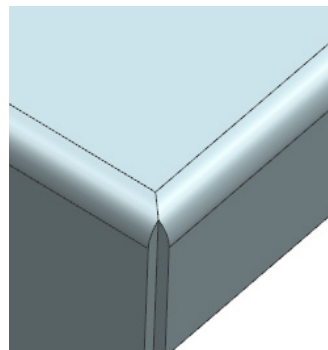


Figure 14-52 *The closed corner treatment with the **Miter Corner** check box cleared*

Relief Properties Rollout

The options in this rollout are used to specify the size and position of the corner treatment. These options are discussed next.

Origin

The options in this drop-down list are used to specify the origin point of the corner treatment. Select the **Bend Center** or **Corner Point** option to specify the origin of the corner treatment.

Offset

You can enter a value in this edit box to create the corner at an offset from the origin of the corner treatment.

The other options in this vary with the type of corner treatment selected. These options were discussed earlier. After specifying all the options in the **Closed Corner** dialog box, choose the **OK** button to create the corner treatment.

CREATING DIMPLES IN A SHEET METAL PART

Ribbon: Home > Punch > Dimple
Menu: Insert > Punch > Dimple

 NX allows you to sketch a user-defined shape and then use it to create a dimple in the sheet metal component. To create the dimple, invoke the **Dimple** tool; the **Dimple** dialog box will be displayed, as shown in Figure 14-53. The options in the **Dimple** dialog box are discussed next.

Section Rollout

The **Select Curve** area in this rollout is activated by default when you invoke the **Dimple** tool. As a result, you are prompted to select a planar face or section geometry. Select a face on which you want to sketch the profile of the dimple or an existing sketch.

Dimple Properties Rollout

The options in this rollout are discussed next.

Depth

This edit box is used to specify the depth of the dimple.

This value is modified dynamically as you drag the depth handle displayed in the drawing window.

Reverse Direction

This button is used to reverse the direction of the dimple.

Side Angle

This edit box is used to specify the taper angle for the dimple. Figure 14-54 shows a dimple with no taper angle and Figure 14-55 shows a dimple with a taper angle of 25 degrees.

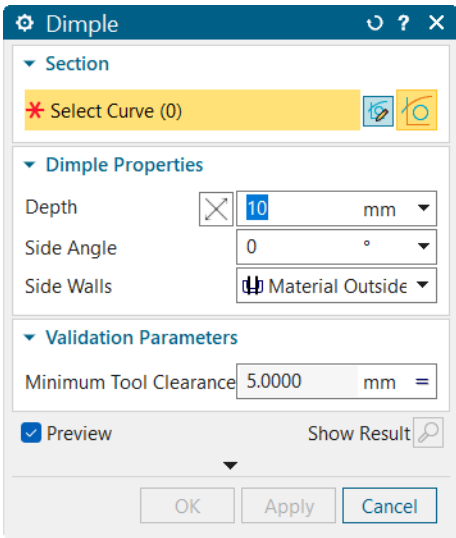


Figure 14-53 The *Dimple* dialog box

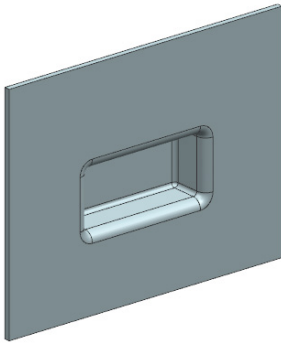


Figure 14-54 Dimple with no taper

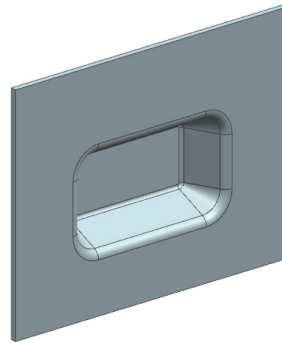


Figure 14-55 Dimple with a 25 degree taper

Side Walls

The options in this drop-down list are used to specify the material condition for side walls of the dimple.

Material Inside



This option is selected to create the dimple such that the material is added inside the profile of the dimple.

Material Outside



Select this option to add the material outside the profile of the dimple.

Settings Rollout

The options in this rollout are used to apply rounds to edges and corners of a dimple. These options are discussed next.

Blend Dimple Edges

This check box is selected to apply rounds to dimple edges. On selecting this check box, the **Punch Radius** and **Die Radius** edit boxes are displayed and they are discussed next.

Punch Radius

This edit box is used to specify the punch radius of a dimple feature, refer to Figure 14-56.

Die Radius

This edit box is used to specify the die radius of a dimple feature, refer to Figure 14-56.

Blend Section Corners

This check box is selected to apply rounds to the corners of a dimple section, refer to Figure 14-56. On selecting this check box, the **Corner Radius** edit box is displayed which is used to specify corner radius.

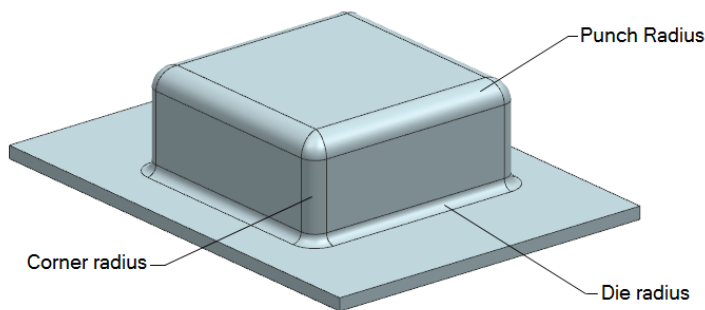


Figure 14-56 Dimple feature with rounds

NX also allows you to create open profiles for the dimple. Note that the open profile should be created such that when extended, the open entities intersect the edges of the model. Figure 14-57 shows the preview of a dimple created using an open profile. The open profile is also shown in the same figure. Note that in this profile, when the two inclined lines are extended, they intersect with the top edge of the sheet metal part.

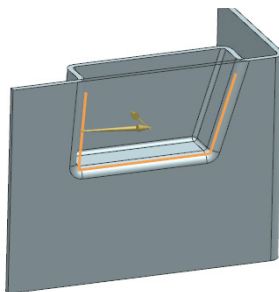


Figure 14-57 Dimple created using an open profile

CREATING LOUVERS IN A SHEET METAL PART

Ribbon: Home > Punch > Louver

Menu: Insert > Punch > Louver



Louvers are created in a sheet metal part to provide openings in it. Figure 14-58 shows a sheet metal part with a rectangular pattern of louvers on its top face.

In NX, louvers are created by sketching a single line segment defining the length of the louver. To create the louver, invoke the **Louver** tool; the **Louver** dialog box will be displayed, as shown in Figure 14-59. In this dialog box, select the shape of the louver from the **Louver Shape** drop-down list available in the **Louver Properties** rollout. There are two types of louvers available: Lanced and Formed. Figure 14-60 shows a pattern of the formed louver and Figure 14-61 shows a pattern of the lanced louvers.

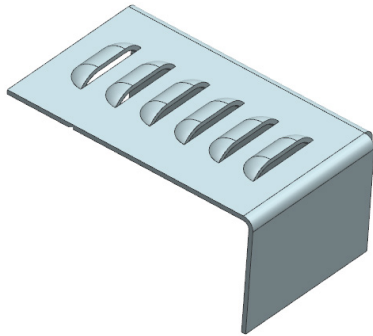


Figure 14-58 Sheet metal part with a pattern of louvers on the top face

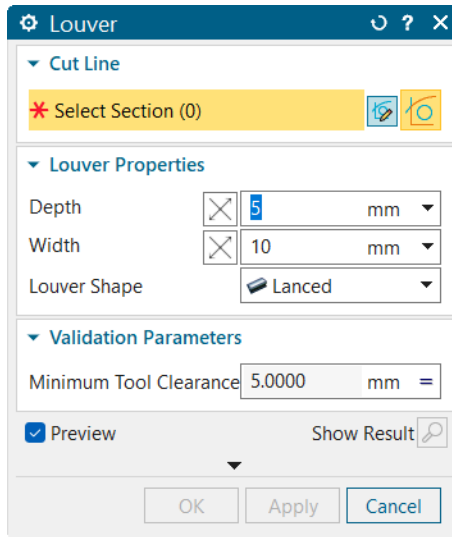


Figure 14-59 The **Louver** dialog box

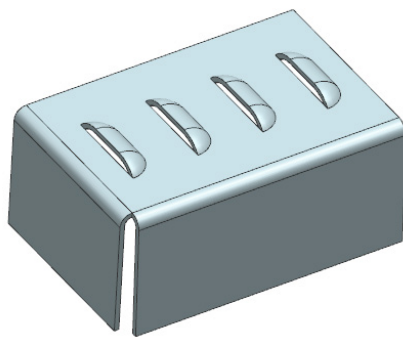


Figure 14-60 Formed louvers

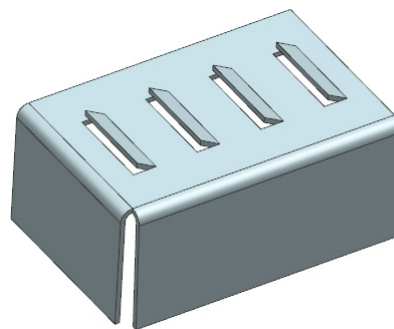


Figure 14-61 Lanced louvers

After selecting the type of louvers, select the face on which you want to sketch the profile of the louver; the sketching environment is invoked and you are prompted to draw the sketch. Draw the profile of the louver and then exit the sketching environment. Note that the profile has to be a single line segment. Next, you need to specify the width of the louver. You can enter the width in the **Width** edit box or specify it dynamically in the drawing window. Note that the width value is specified along the face on which the sketch is drawn and it should be less than half of the line length. You can use the **Reverse Direction** button located adjacent to the **Width** edit box to specify the side of the line on which the width of the louver will be added.

Next, you need to specify the depth of the louver. The depth is specified normal to the face on which the sketch is drawn. You can enter the depth in the **Depth** edit box or specify it dynamically in the drawing window. You can use the **Reverse Direction** button located adjacent to the **Depth** edit box to specify the side of the sheet on which the louver will be added. Note that the depth should be less than or equal to the difference of the width of the louver and the sheet thickness. Also, the depth should be more than the sheet thickness. For example, if the

width of the louver is 10 mm and the sheet thickness is 2 mm, the height of the louver should be more than 2 and less than or equal to 8. Figure 14-62 shows the preview of a formed louver. The width of the louver is 10 mm and its depth is 5 mm.

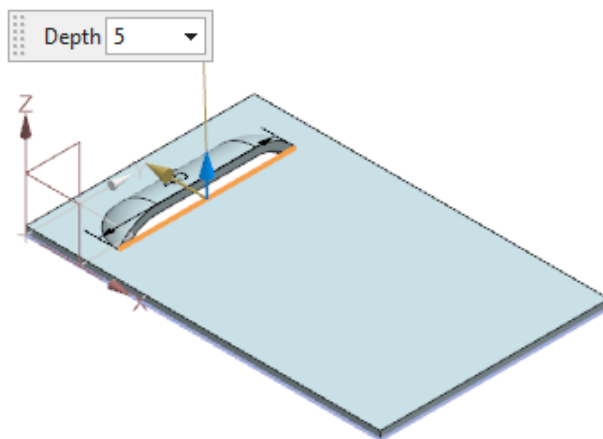


Figure 14-62 Preview of the formed louver

CREATING DRAWN CUTOUTS IN A SHEET METAL PART

Ribbon: Home > Punch > More Gallery > Punch Gallery > Drawn Cutout
Menu: Insert > Punch > Drawn Cutout



The drawn cutouts are exactly the same as the dimples with the only difference that in drawn cutouts the end face is open. Figure 14-63 shows a sheet metal part with two drawn cutouts of different shapes. In this figure, the rectangular cutout is created in the upward direction and the oblong cutout is created in the downward direction.

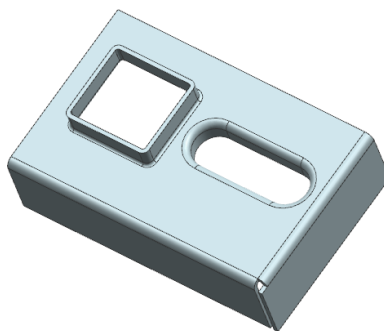


Figure 14-61 Drawn cutouts of different shapes

To create a drawn cutout, invoke the **Drawn Cutout** tool and select the face on which you want to sketch the profile of the drawn cutout; the sketching environment is invoked and you are prompted to draw the sketch. Draw the profile of the cutout and then exit the sketching environment. On doing so, the preview of the cutout will be displayed along with the Depth handle and an edit box showing the default depth of the cutout. Specify the depth of the cutout by entering the

desired value in the edit box or by using the depth handle. You can change the direction in which the cutout is created by using the **Reverse Direction** button. You can also specify a taper to the cutout by entering the taper angle in the **Side Angle** edit box of the **Drawn Cutout** dialog box.

**Note**

*The working of the **Drawn Cutout** tool is same as that of the **Dimple** tool.*

CREATING BEADS IN A SHEET METAL PART

Ribbon: Home > Punch > More Gallery > Punch Gallery > Bead
Menu: Insert > Punch > Bead



The **Bead** tool is used to create an embossed or an engraved bead on a sheet metal part using a single entity or a set of tangentially connected entities, refer to Figure 14-64. Figure 14-65 shows an embossed bead and Figure 14-66 shows an engraved bead.

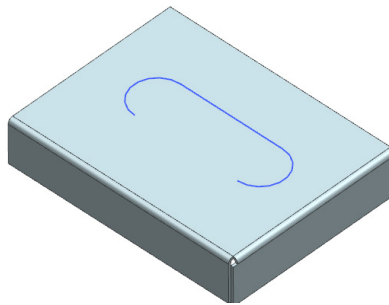


Figure 14-64 A sketch to create the bead feature

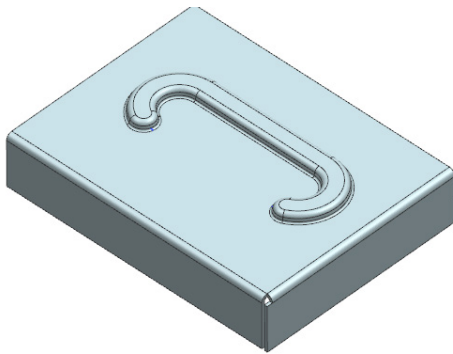


Figure 14-65 Bead created in the upward direction, resulting in the embossed feature

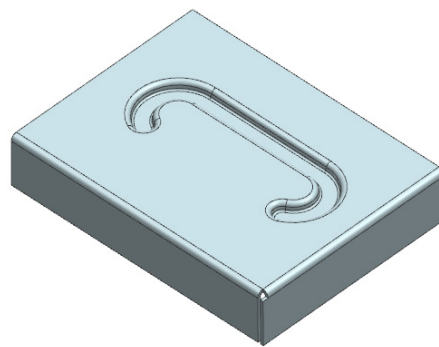
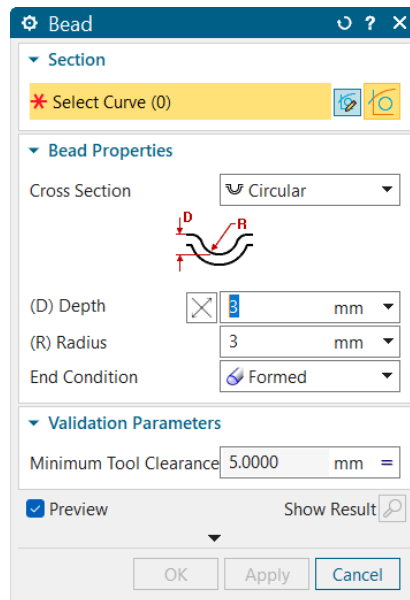


Figure 14-66 Bead created in the downward direction, resulting in the engraved feature

When you invoke this tool, the **Bead** dialog box is displayed, as shown in Figure 14-67.



*Figure 14-67 The **Bead** dialog box*

Bead Dialog Box

The options available in this dialog box are discussed next.

Section Rollout

The **Select Curve** area in this rollout is activated by default. As a result, you are prompted to select a planar face. Select a face on which you want to sketch the profile of the bead; the sketching environment is invoked and you are prompted to draw the sketch. Draw the profile of the bead and then exit the sketching environment. Note that the profile has to be a single line segment or a set of tangentially connected open or closed entities.

Bead Properties Rollout

The options available in this rollout are used to specify the parameters of the bead. You can specify the cross-section of the bead using the options in the **Cross Section** drop-down list. You can specify the depth, radius, width, and the angle of the bead using the edit boxes available in this rollout. Figures 14-68 through 14-70 show beads created using different cross-sections.

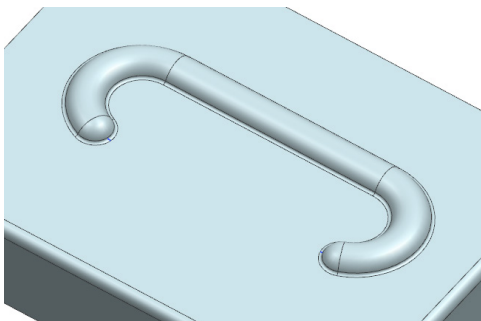


Figure 14-68 Circular cross-section bead

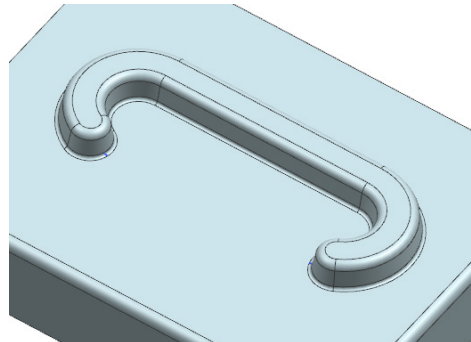


Figure 14-69 U-shaped bead

The **End Condition** drop-down list is used to specify the end conditions of the bead. You can specify the end condition as formed, lanced, punched, or tapered. The punch gap can be specified in the **Punch Width** edit box that will be available on selecting the **Punched** option from this drop-down list. Figures 14-71 through 14-73 show beads using different end conditions.

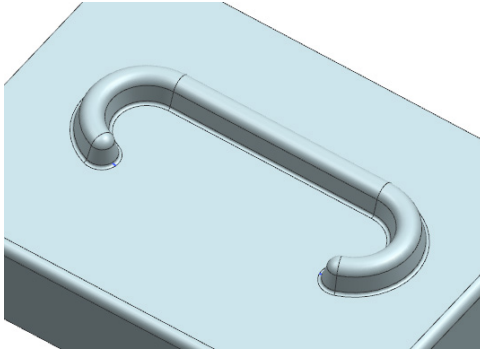


Figure 14-70 V-shaped bead

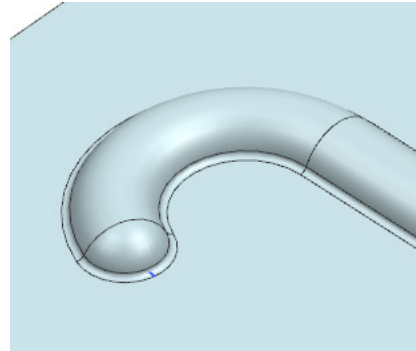


Figure 14-71 Bead created using formed end condition

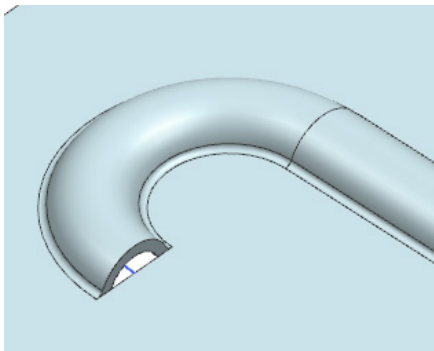


Figure 14-72 Bead created using lanced end condition

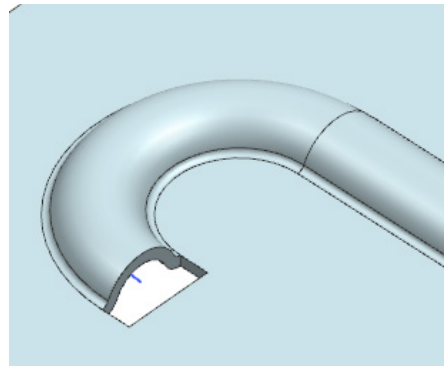


Figure 14-73 Bead created using punched end condition with a gap of 2 mm



Note

The remaining options in the **Bead** dialog box have already been discussed.

After specifying the bead options, choose the **OK** button from the **Bead** dialog box to create the bead.

ADDING GUSSETS TO A SHEET METAL PART

Ribbon: Home > Punch > More Gallery > Punch Gallery > Gusset
Menu: Insert > Punch > Gusset



Gussets are rib like stiffeners that are added to the sheet metal part to increase its strength. In NX, you can create an automatic gusset of round or square shape or a gusset with a user-defined profile. Figures 14-74 through 14-77 show various types of gussets that can be created in NX.

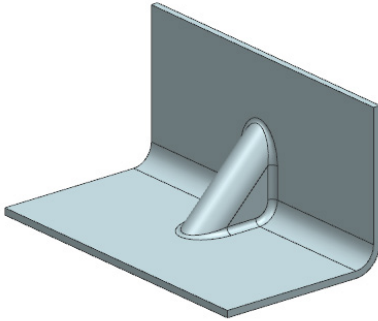


Figure 14-74 Front view of a round gusset

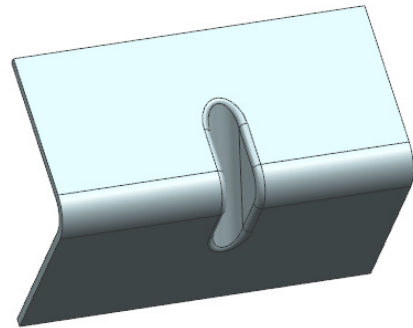


Figure 14-75 Back view of the round gusset

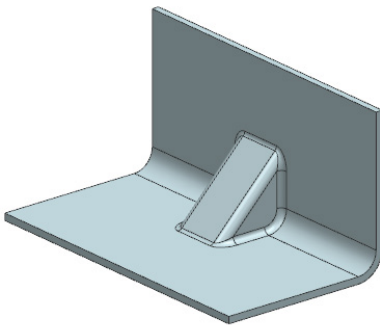


Figure 14-76 A square gusset

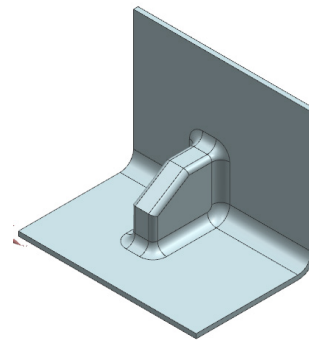


Figure 14-77 A user-defined gusset

To add a gusset to a sheet metal part, invoke the **Gusset** tool from the **Punch** gallery of the **Home** tab; the **Gusset** dialog box will be displayed, refer to Figure 14-78. In NX, you can create two types of gussets; Automatic and User-defined. These two types are discussed next.

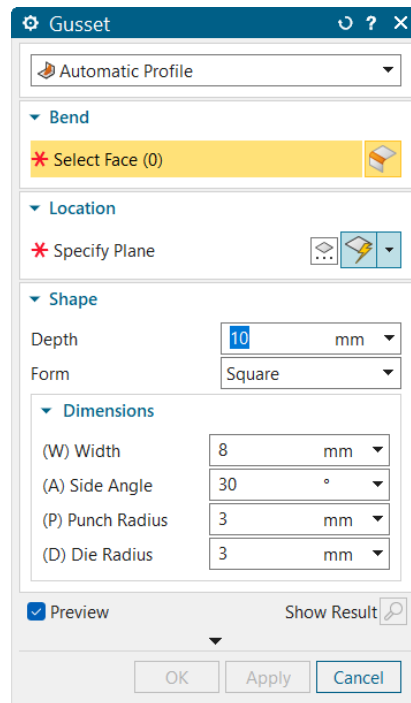


Figure 14-76 The Gusset dialog box

Creating a Gusset Using the Automatic Profile Option

By default, the **Automatic Profile** option is selected in the Type drop-down list and the **Select Face** area is activated in the **Bend** rollout. Select the bend face and then specify plane from the **Location** rollout to position the gusset; the preview of the gusset along with the handle and dynamic edit box will be displayed. The other options to create an automatic gusset are discussed next.

Shape Rollout

You can use the options in this rollout to control the parameters of a gusset. The **Depth** edit box is used to specify the depth of the gusset. The options in the **Form** drop-down list are used to define the shape of the gusset. You can create a gusset with a round shape by selecting the **Round** option, refer to Figure 14-79. The **Square** option is used to create a gusset with a square shape. You can also apply fillets to the sharp edges of a square gusset by entering the radius values in the **Punch Radius** and **Die Radius** edit boxes in the **Dimensions** sub-rollout.

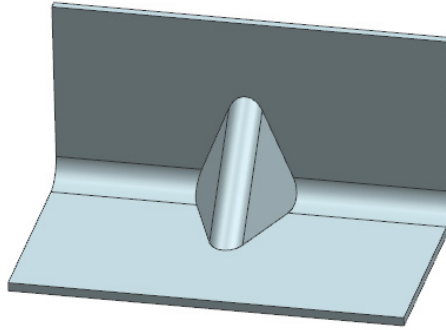


Figure 14-79 A gusset with a taper angle of 30 degrees

You can specify the width of the gusset in the **Width** edit box. The **Side Angle** edit box is used to enter the taper angle for the gusset. Figure 14-79 shows a gusset with a taper angle of 30 degrees.

Creating a Gusset Using the User Defined Profile Option

By selecting the **User Defined Profile** option from the Type drop-down list, you can create gusset with a user-defined profile. To create this type gusset, select the **User Defined Profile** option from the Type drop-down list and select an edge on the round on which the gusset is to be placed, refer to Figure 14-80; a plane normal to the selected edge will be displayed. Also, the **Create Sketch** dialog box will be displayed. Move the plane to the location at which you want to place the gusset and choose the **OK** button from the **Create Sketch** dialog box; the sketch environment will be invoked. Create the profile of the gusset and exit the sketch environment; the preview of the gusset will be displayed.

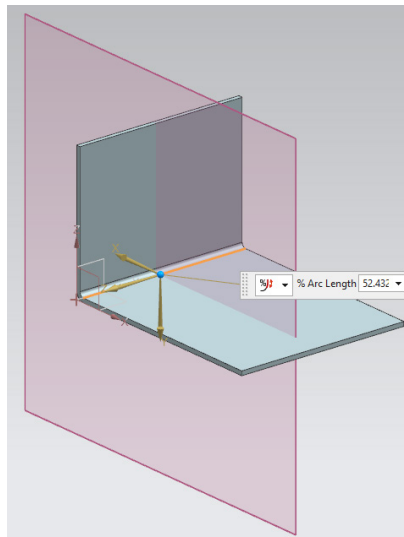


Figure 14-80 Edge selected on the round

You can use the options in the **Width Side** drop-down list to specify the side of the profile on which the gusset will be created. You can select the **Side 1**, **Side 2**, or **Symmetric** option from this drop-down list.

You can use the **Alternate Solution** button to cycle through the alternate possible solutions of the gusset.

ADDING HEMS

Ribbon: Home > Bend > More Gallery > Bend Gallery > Hem Flange (*Customize to Add*)

Menu: Insert > Bend > Hem Flange



Hems are defined as the rounded faces created on the sharp edges of a sheet metal component to reduce the area of the sharpness on the edges. This makes the sheet metal component easy to handle and assemble. To create a hem, invoke the **Hem Flange** tool; the **Hem** dialog box will be displayed, as shown in Figure 14-81. Also, you will be prompted to select an edge to hem. Select the edge and specify the hem parameters in the **Hem** dialog box.

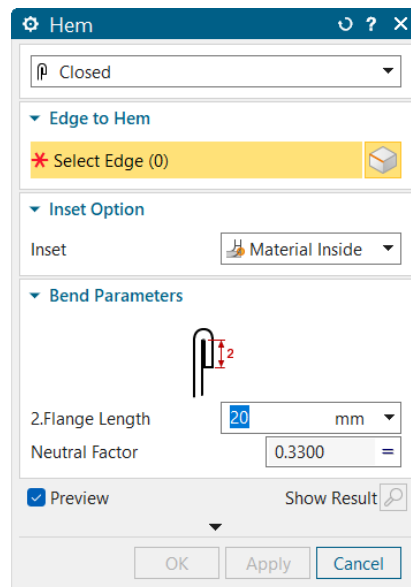


Figure 14-81 The **Hem** dialog box

The options in the **Hem** dialog box vary depending on the type of hem selected in the Type drop-down. Some of the options in this dialog box are discussed next.

Type Drop-down

The Type drop-down list is used to specify the hem type. There are seven types of hems available in this drop-down list which are discussed next.

Closed

This option is used to create a closed hem, as shown in Figure 14-82. The flange length can be specified in the **2.Flange Length** edit box available in the **Bend Parameters** rollout.

Open

This option is used to create an open hem, as shown in Figure 14-83. The bend radius and the flange length can be specified in the **1.Bend Radius** and **2.Flange Length** edit boxes of the **Bend Parameters** rollout, respectively.

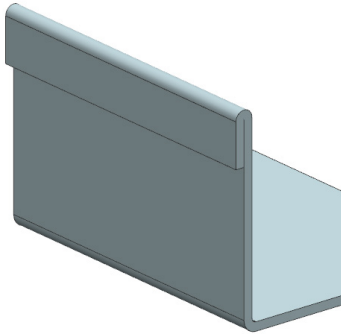


Figure 14-82 A closed hem

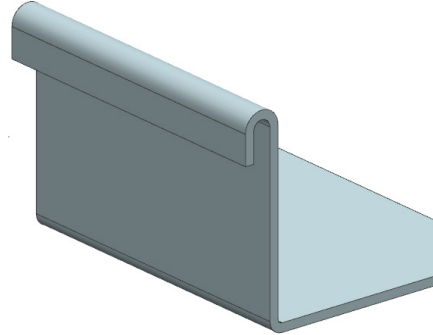


Figure 14-83 An open hem

S-Type

This option is used to create a hem with the S shape, as shown in Figure 14-84. The bend radius and the flange lengths can be specified in their respective edit boxes in the **Bend Parameters** rollout.

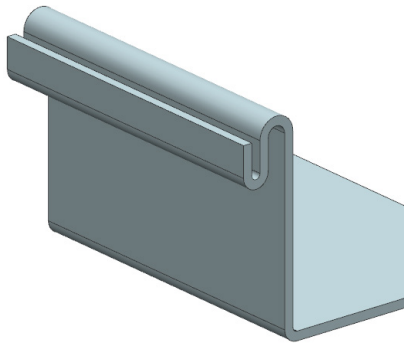


Figure 14-84 A S-shaped hem

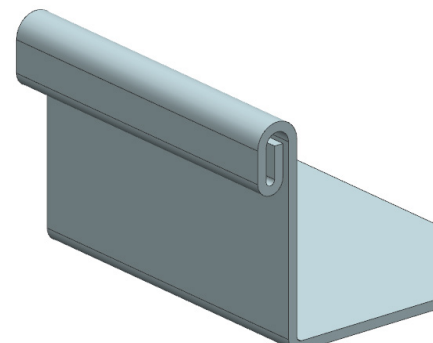


Figure 14-85 A curled hem

Curl

This option is used to create a curled hem, as shown in Figure 14-85. The bend radius and the flange lengths can be specified in their respective edit boxes in the **Bend Parameters** rollout.

Open Loop

This option is used to create a hem that has an open loop, as shown in Figure 14-86. The bend radius and the included angle of the open loop can be specified in the **1.Bend Radius** and **5.Sweep Angle** edit boxes available in the **Bend Parameters** rollout, respectively.

Closed Loop

This option is used to create a hem that has a closed loop, as shown in Figure 14-87. The bend radius and the flange length can be specified in their respective edit boxes available in the **Bend Parameters** rollout.

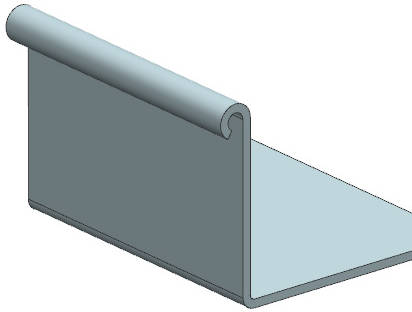


Figure 14-86 An open loop hem

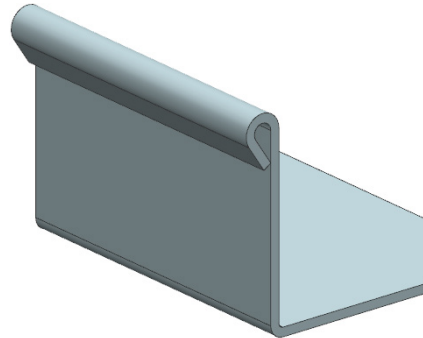


Figure 14-87 A closed loop hem

Centered Loop

This option is used to create a hem with a centered loop, as shown in Figure 14-88. The bend radius and the included angle can be specified in their respective edit boxes available in the **Bend Parameters** rollout.

Miter Rollout

The **Miter Hem** check box in this rollout is used to add a miter to the hem, as shown in Figure 14-89. To do so, select this check box and specify a value in the **Miter Angle** edit box.

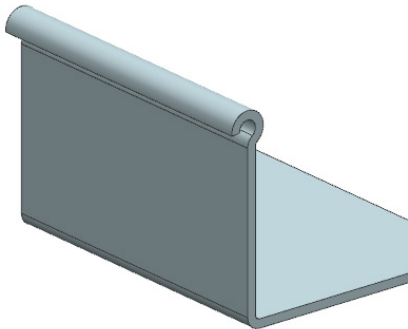


Figure 14-88 Hem with a centered loop

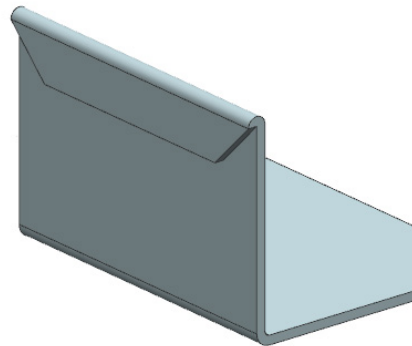


Figure 14-89 Hem with a miter



Note

The remaining options of the **Hem** dialog box have already been discussed in this chapter.

CREATING A SHEET METAL PART USING SOLID BODY

Ribbon: Home > Convert > Sheet Metal from Solid
Menu: Insert > Bend > Sheet Metal from Solid



NX allows you to create a sheet metal part by using the shape of a solid part. To create a sheet metal by using a solid, invoke the **Sheet Metal from Solid** tool; the **Sheet Metal from Solid** dialog box will be displayed. Also, you will be prompted to select planar web faces. Select faces of the solid part. Note that you need to select a minimum of two faces from the solid. Also, if you select more than two faces, you need to select the bend edges manually. Next, choose the **OK** button from the **Sheet Metal from Solid** dialog box; the sheet metal part will be created on selected faces. Figure 14-90 shows the faces selected on a solid part and Figure 14-91 shows a sheet metal part created.

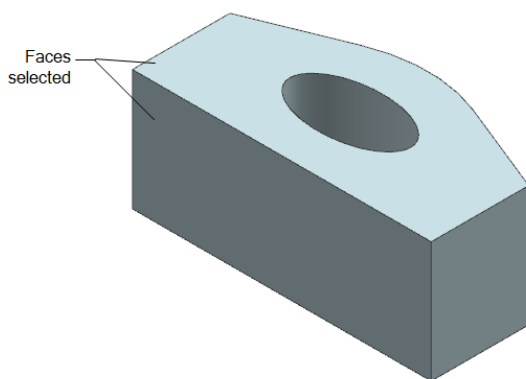


Figure 14-90 Faces selected on the solid part

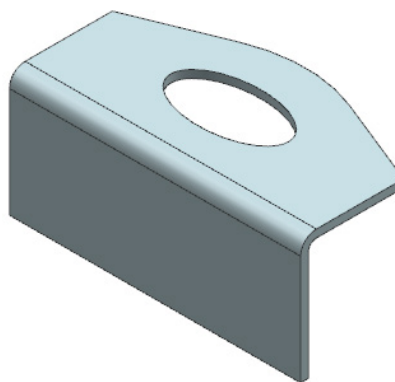


Figure 14-91 Sheet Metal part created

CONVERTING A SOLID PART INTO A SHEET METAL PART

Ribbon: Home > Convert > Convert to Sheet Metal
Menu: Insert > Convert > Convert to Sheet Metal



You can convert a solid part into a sheet metal part using the **Convert to Sheet Metal** tool. To do so, create a solid part in the Modeling environment and then invoke the Sheet Metal environment by choosing **Application > Design > Sheet Metal** from the **Ribbon**. Next, choose the **Convert to Sheet Metal** tool from the **Convert** group. Figure 14-92 shows a solid part created in the Modeling environment and Figure 14-93 shows the converted sheet metal part from the solid part. The procedure to convert a solid part into sheet metal part is discussed next.

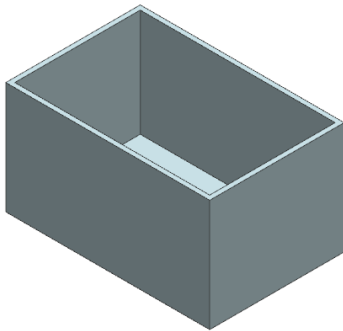


Figure 14-92 The part created in the Modeling environment

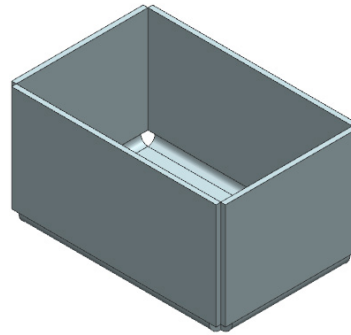


Figure 14-93 Part converted to Sheet Metal component

When you invoke the **Convert to Sheet Metal** dialog box, the **Select Base Face** area is activated by default in the **Global Conversion** rollout. As a result, you will be prompted to select the base face. Figure 14-94 shows the four vertical edges selected to be ripped. After selecting the edges to be ripped, choose the **OK** button from the dialog box; the solid model will be converted into a sheet metal part, as shown in Figure 14-94.

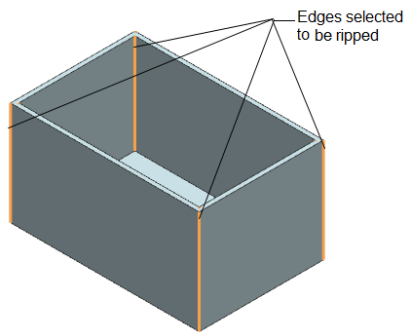


Figure 14-94 Edges selected to be ripped

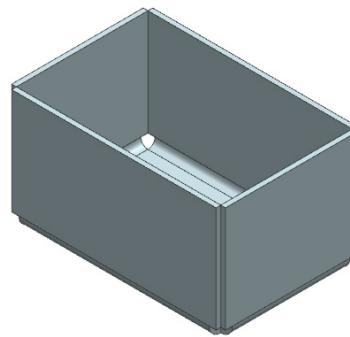


Figure 14-95 Sheet Metal part after ripping the corners

RIPPING THE CORNERS OF A SOLID PART

Ribbon: Home > Convert > More Gallery > Convert Gallery > Rip (Customize to Add)

Menu: Insert > Convert > Rip



You can rip the corners of a solid part using the **Rip** tool. Note that when you use this tool, no corner relief is added to the model. To rip the corners, invoke this tool and select the edges whose corners you want to rip.



Note

You can also use the other tools such as **Normal Cutout**, **Mirror**, and so on in the sheet metal environment. These tools are discussed in detail in the part modeling environment.

CREATING THE FLAT PATTERN OF A SHEET METAL PART

As mentioned earlier, you need to flatten a sheet metal part after creating it in order to generate its flattened drawing views and manufacture it. NX provides a number of options to flatten a sheet metal part. Some of the options are discussed next.

Creating the Flat Pattern

Ribbon: Home > Flat Pattern > Flat Pattern
Menu: Insert > Flat Pattern > Flat Pattern



In NX, you need to create the flat pattern of a sheet metal component by using the **Flat Pattern** tool. On invoking this tool, the **Flat Pattern** dialog box will be displayed and you will be prompted to click on the face to be oriented upward in the flat pattern. Select the base face of the sheet metal part.

Figure 14-96 shows a sheet metal part and Figure 14-97 shows the flat pattern. Note that in the flat pattern, the lower horizontal edge close to the left endpoint was selected to define the X axis and the origin. To view the flat pattern, choose **Menu > View > Layout > Replace View** from the **Top Border Bar**; the **Replace View with** dialog box will be displayed. Select **FLAT-PATTERN** and then choose the **OK** button from this dialog box; the flat pattern will be displayed. To switch back to the model view, again invoke the **Replace View with** dialog box and select **Isometric** from the dialog box. Next, choose the **OK** button; the isometric view of the model will be displayed.



Note

1. In NX, you can control the orientation of the flat pattern view for the upward face having non-linear edges. To do so, specify a CSYS while defining the orientation.
2. You can represent the holes smaller than or equal to a specified diameter value as centermarks in the flat pattern view. This is useful when the hole creation needs a different manufacturing process.
3. You can create both associative and non-associative flat patterns with respect to the parent body.

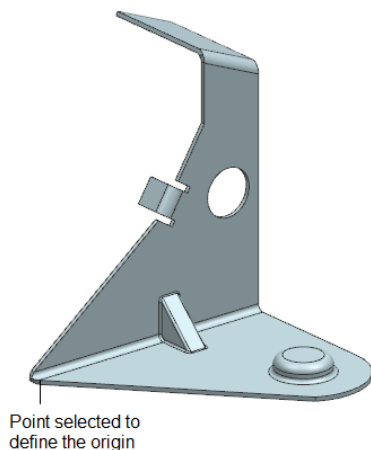


Figure 14-96 The sheet metal part

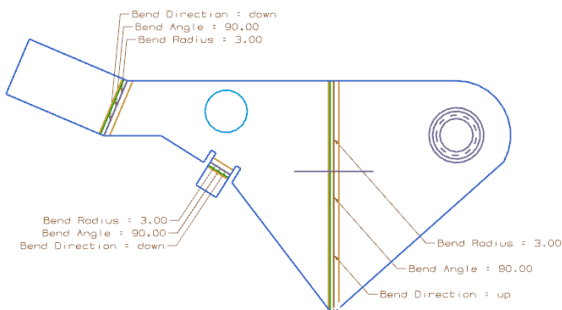


Figure 14-97 The flat pattern

Creating the Flat Solid

Ribbon: Home > Flat Pattern > Flat Solid (*Customize to Add*)

Menu: Insert > Flat Pattern > Flat Solid

T In NX, you can create a flat solid of a sheet metal part. Further, you can use this flat solid to create drawing views. To create a flat solid of a sheet metal part, choose the **Flat Solid** tool from the **Flat Pattern** group of the **Home** tab; the **Flat Solid** dialog box will be displayed and you will be prompted to select a face that will remain stationary in the flat solid. Select a face from the drawing window and choose the **OK** button; the flat solid will be created, refer to Figure 14-98. If you select the **Select Edge** option from the **Orientation Method** drop-down list of the **Orientation** rollout, then after selecting the stationary face, you will be prompted to select an edge for X-axis orientation. Select an edge from the model, refer to Figure 14-99 and then choose the **OK** button; the flat solid will be created at the specified orientation, refer to Figure 14-100.

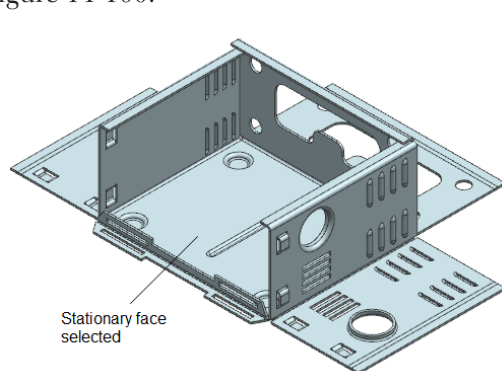


Figure 14-98 The stationary face selected to create the flat solid

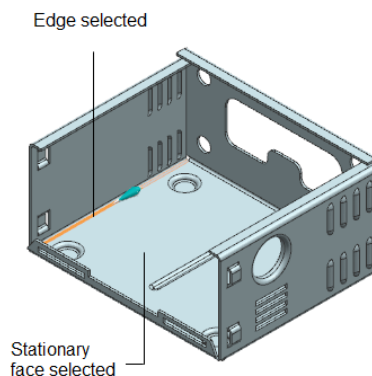


Figure 14-99 The stationary face and edge selected to create the flat solid

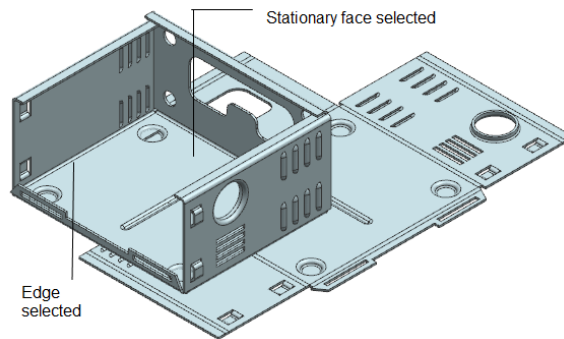


Figure 14-100 The flat solid created at the specified orientation

Exporting a Flat Pattern

Ribbon: Home > Flat Pattern > Export Flat Pattern
Menu: Insert > Flat Pattern > Export Flat Pattern



You can export a flat pattern to AutoCAD (*.dxf) or *.geo file. To export the flat pattern file, choose the **Export Flat Pattern** tool from the sheet metal environment; the **Export Flat Pattern** dialog box will be displayed. Select the file type from the Type drop-down list and then specify the name and location of the flat pattern file by choosing the **Browse** button from the **Output File** rollout.

RENEWING FEATURES

Ribbon: Home > Edit > More Gallery > Edit Gallery > Renew Feature (*Customize to Add*)
Menu: Edit > Feature > Renew Feature



You can recompute the features created in earlier versions of NX to the current feature version by using the **Renew Feature** tool. This tool recreates a feature using code from the current NX version. Features are enhanced in each NX release to improve their performance and quality. To take advantage of these improvements, you can use the **Renew Feature** tool to recompute features up to the current feature version. To do so, choose the **Renew Feature** tool from the **Edit** gallery of the **More** gallery in the **Edit** group of the **Home** tab; the **Renew Feature** dialog box will be displayed, refer to Figure 14-101.



Features Rollout

You can also select the **Feature Type** option from the **Sort** drop-down list. In this case, you need to expand the node of the feature to be renewed and then select it. Next, choose the **Apply** button; the feature will be renewed to the current version. Next, choose the **Cancel** button to exit the dialog box.

TUTORIALS

The tutorials given next are available in video format. Scan the QR code or visit the following link to get access to the video tutorials.

<https://www.cadcim.com/siemens-nx-2026-tutorial-videos>



Tutorial 1

In this tutorial, you will create the sheet metal part of the Holder Clip shown in Figure 14-102. The flat pattern of the component is shown in Figure 14-103 and its dimensions are given in Figure 14-104. The material thickness, bend radius, relief depth, and relief width is 1 mm each. The specifications of the louver are given next.

Type: Formed-end louver
Width: 4mm

Length: 18mm
Height: 2.5 mm

After creating the sheet metal component, create its flat pattern. Save the component with the name *c14tut1.prt* at the location given next:

|NX|c14

(Expected time: 45 min)

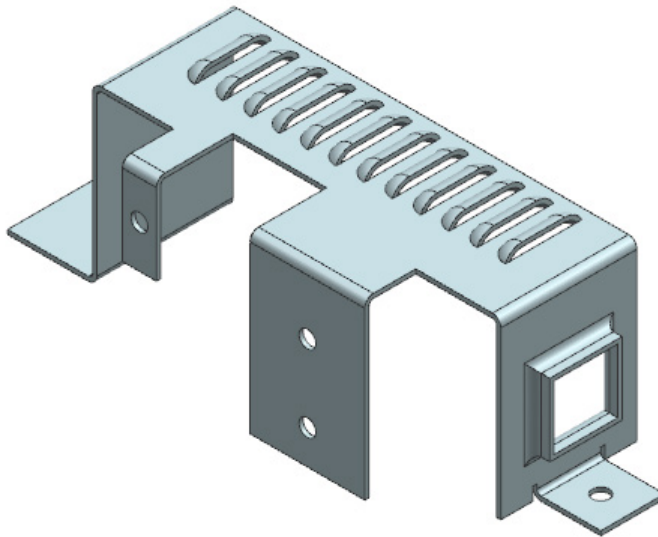


Figure 14-102 Sheet metal part for Tutorial 1

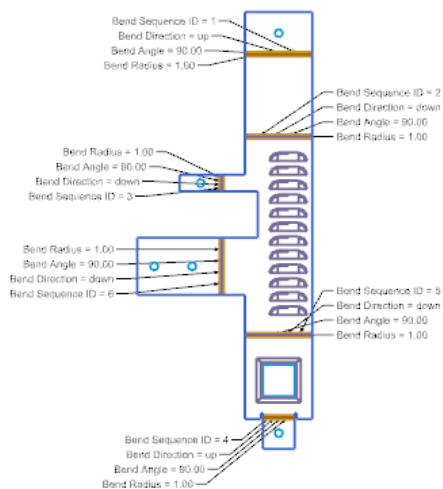


Figure 14-103 Flat pattern of the sheet metal part for Tutorial 1

BEND RADIUS= 1MM
SHEET THICKNESS= 1MM

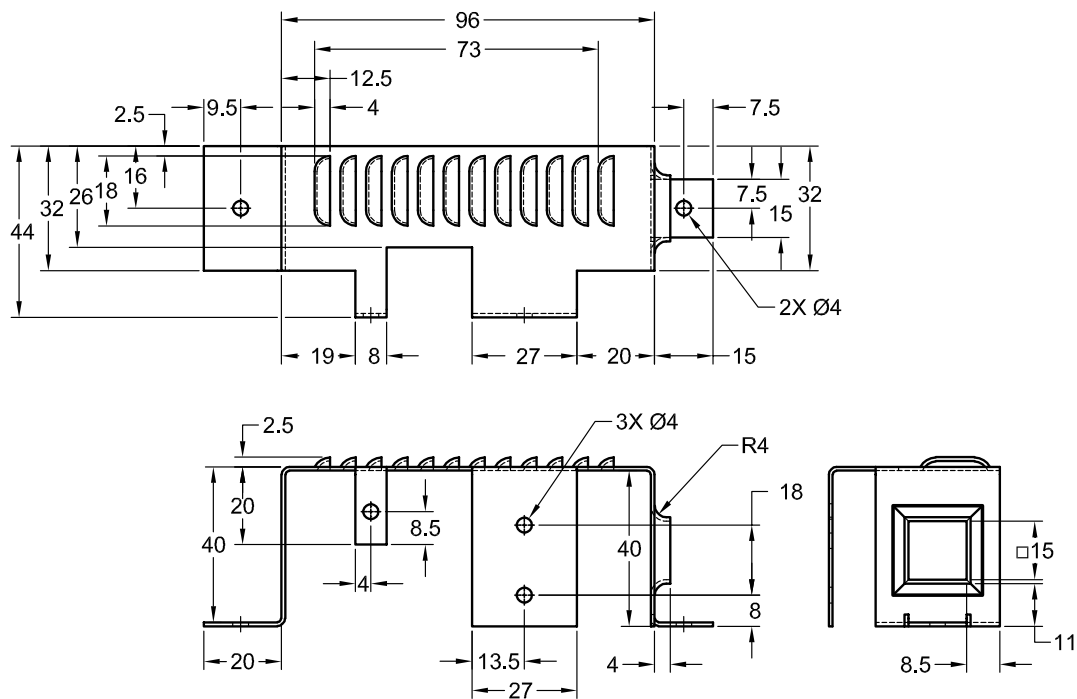


Figure 14-104 Dimensions of the sheet metal part

The following steps are required to complete this tutorial:

- a. Start a new sheet metal file and set the sheet metal parameters.
- b. Create the base feature by using the **Tab** tool.
- c. Create the flanges.
- d. Create the louvers.
- e. Create the holes on the flanges.
- f. Create the drawn cutout on the right face of the model.
- g. Generate the flat pattern of the model.
- h. Save the model.

Starting a New Sheet Metal File

1. Start NX by double-clicking on its shortcut icon on the desktop.
2. To start a new file, choose the **New** tool from the **Standard** group or choose **Menu > File > New** from the **Top Border Bar**; the **New** dialog box is displayed. 
3. Select the **Sheet Metal** template from the **Templates** rollout of the dialog box.
4. Enter **c14tut1** as the name of the document in the **Name** text box of the dialog box.
5. Choose the button on the right side of the **Folder** text box; the **Choose Directory** dialog box is displayed. 
6. In this dialog box, browse to *NX/c14* and then choose the **OK** button twice; the new file is started in the Sheet Metal environment.
7. Choose **Menu > Preferences > Sheet Metal** from the **Top Border Bar**; the **Sheet Metal Preferences** dialog box is displayed.
8. Choose the **Part Properties** tab and set the value of the material thickness, bend radius, relief depth, and relief width to 1mm each.
9. Specify neutral factor as **0.33** and choose the **OK** button to apply it to the model.

Creating the Tab Feature

1. Choose the **Tab** tool from the **Base** group of the **Home** tab; the **Tab** dialog box is invoked and you are prompted to select a planar face to sketch or select the section geometry.
2. Select the **X-Y** plane as the sketching plane and invoke the sketching environment.
3. Draw the sketch for the base feature of the part, as shown in Figure 14-105.
4. Exit the sketching environment and specify the thickness of the sheet metal part in the upward direction. Exit the **Tab** tool to create the base of the sheet metal part, as shown in Figure 14-106.

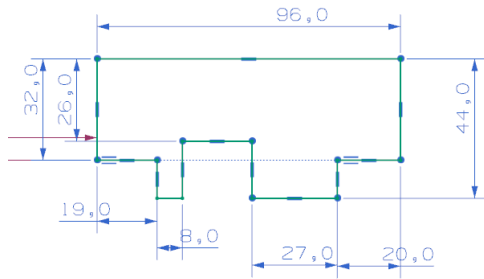


Figure 14-105 Sketch for the top face of the part

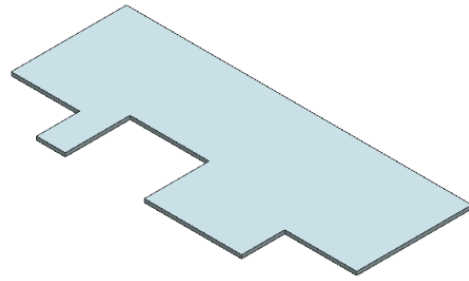


Figure 14-106 The tab feature created

Creating the Flanges

Next, you need to create various flanges on the top face of the sheet metal part. These flanges will be created using the **Flange** tool.

1. Choose the **Flange** tool from the **Base** group of the **Home** tab; the **Flange** dialog box is invoked and you are prompted to select a linear edge.
2. Select the edge on tab to create the first flange, refer to Figure 14-107; the preview of the full-width flange along with an edit box is displayed.

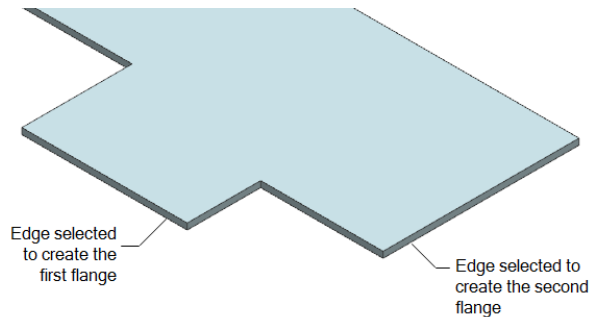


Figure 14-107 Edges to be selected to create flanges

3. Specify **40** mm as the distance of the flange in the **Length** edit box. Figure 14-108 shows the preview of the flange.
4. Similarly, create the second flange of 40 mm length by selecting the edge, refer to Figure 14-107. Figure 14-109 shows the preview of the flange created on the second edge selected. Next, choose the **Apply** button to create the flange.

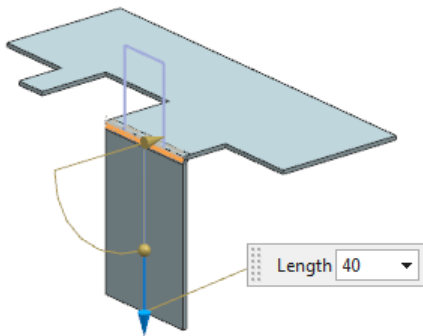


Figure 14-108 Preview of the first flange

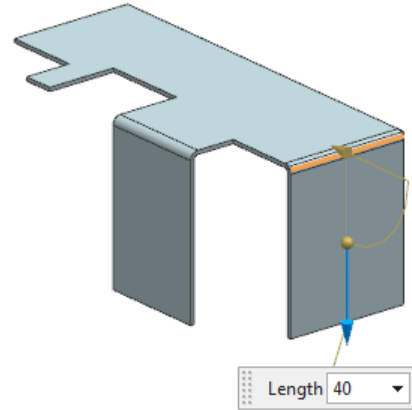


Figure 14-109 Preview of the second flange

Now, you need to create the centered flange.

5. Select the bottom edge of the second flange, refer to Figure 14-110.
6. Select the **At Center** option from the **Width Option** drop-down list in the **Flange** dialog box.
7. Enter **15** in both the **Width** and **Length** edit boxes of the **Flange** dialog box, and choose the **Apply** button; the centered flange is created, as shown in Figure 14-111.

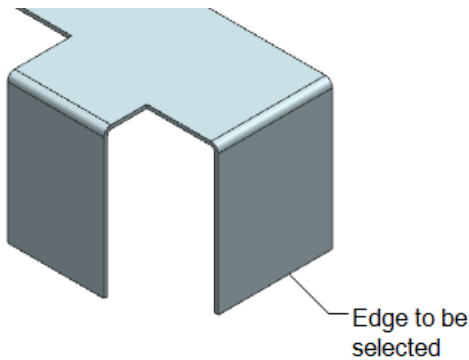


Figure 14-110 Edge to be selected

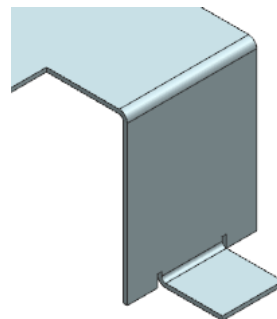


Figure 14-111 The centered flange

8. Select the edge, as shown in Figure 14-112.

9. Select the **Full** option from the **Width Option** drop-down list in the **Flange** dialog box.
10. Enter **20** mm in the **Length** edit box and choose the **Apply** button from the **Flange** dialog box; the fourth flange is created, as shown in Figure 14-113.

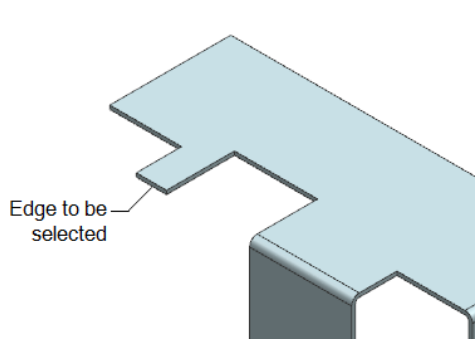


Figure 14-112 Edge to be selected

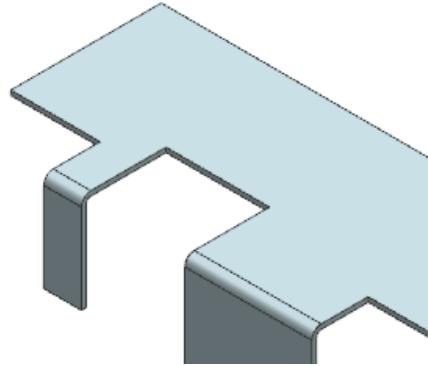


Figure 14-113 The fourth flange

11. Next, create the fifth flange of 40 mm length by selecting the edge, as shown in Figure 14-114. The preview of the flange is shown in Figure 14-115.

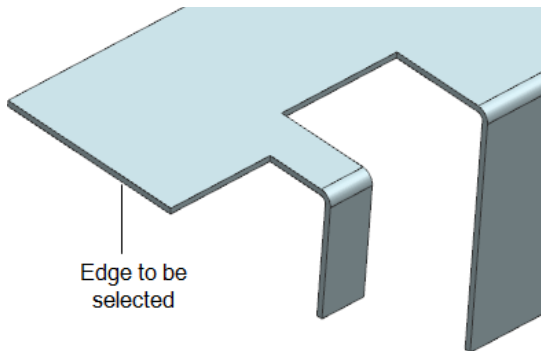


Figure 14-114 Edge to be selected

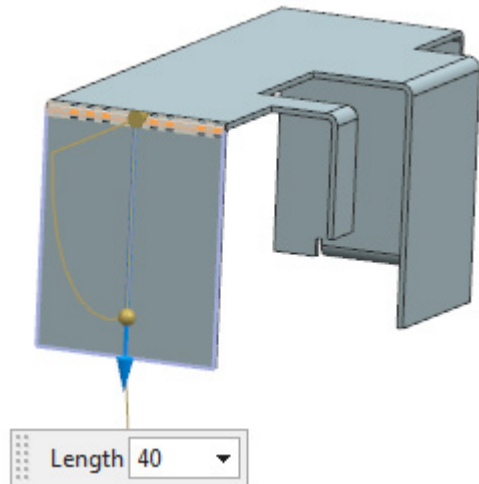


Figure 14-115 Preview of the fifth flange

12. Finally, create the sixth flange of 20 mm length on the bottom edge of the previous flange, refer to Figure 14-116 and then exit the **Flange** dialog box.

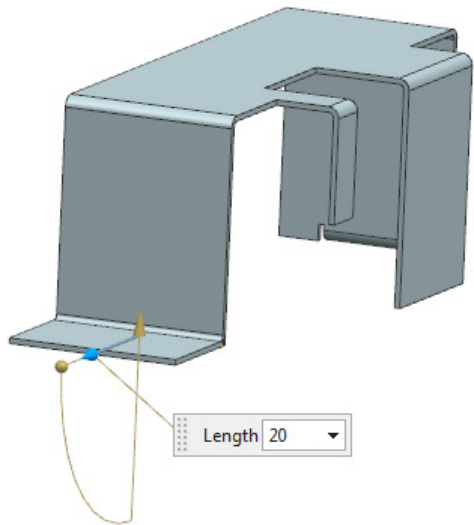


Figure 14-116 Preview of the sixth flange

Creating Louvers on the Top Face

Next, you need to create louvers on the top face. In this process, first you need to create one of the louvers using the **Louver** tool and then create its rectangular pattern.

1. Choose the **Louver** tool from the **Punch** group of the **Home** tab; the **Louver** dialog box is displayed; you are prompted to select a planar face or select a section geometry.
2. Select the top planar face of the sheet metal part; the Sketch in Task environment is invoked.
3. Draw the profile of the louver and add the required dimensions, as shown in Figure 14-117. Exit the sketching environment; the preview of the louver is displayed.
4. Specify the settings in the **Louver Properties** rollout, as shown in Figure 14-118, in the **Louver** dialog box for the louver to be created. Choose the **OK** button; the louver is created, as shown in Figure 14-119.

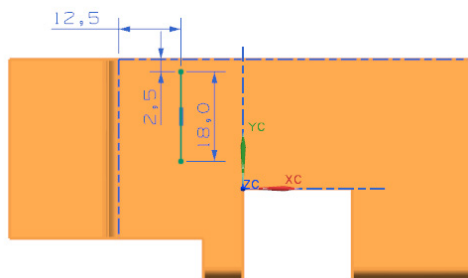


Figure 14-117 Dimensions to position the louver

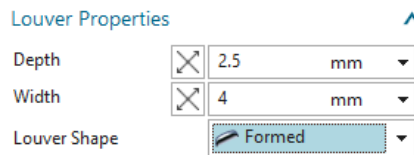


Figure 14-118 The settings required for the louver to be created

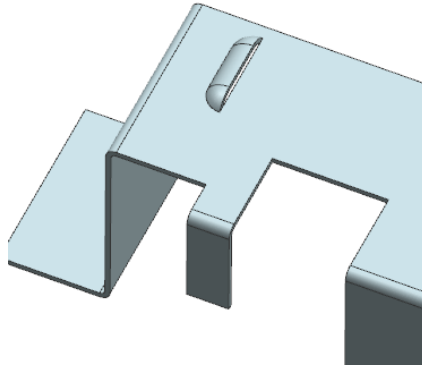


Figure 14-119 Louver created on the top face

Now, you need to create the pattern of the louvers.

5. Select the louver and choose **Menu > Insert > Associate Copy > Pattern Feature** from the **Top Border Bar**; the **Pattern Feature** dialog box is displayed.
6. In this dialog box, select the **Linear** option from the **Layout** drop-down list and click in the **Specify Vector** area of the **Direction 1** sub-rollout; a triad is displayed in the drawing window.
7. Select the axis of the triad, as shown in Figure 14-120 and then select the **Count and Span** option from the **Spacing** drop-down list.

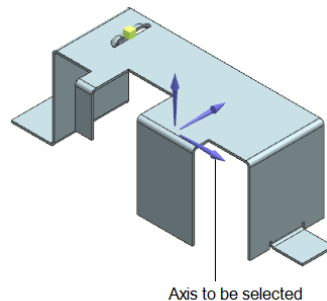


Figure 14-120 Axis to be selected from the triad

8. Enter **12** in the **Count** edit box and **73** in the **Span Distance** edit box to create a rectangular pattern of the louver with 12 instances along the selected direction. Ensure that the **Variational** option is selected in the **Method** drop-down of the **Pattern Method** rollout.
9. Choose the **OK** button from the **Pattern Feature** dialog box to accept the pattern created.

Creating the Holes

1. Choose **Menu > Insert > Design Feature > Hole** from the **Top Border Bar**; the **Hole** dialog box is displayed and you are prompted to select the planar face to sketch or specify points.

2. Select the **Simple** option from the Type drop-down list, if it is not selected by default.
3. Select the front face of the first flange; the point is placed on the selected face. Place another point and specify the position of both the points by adding dimensions. For positional dimensions, refer to Figure 14-104.
4. Enter **4** in the **Hole Diameter** edit box of the **Form** rollout.
5. Select the **Through Body** option from the **Depth Limit** drop-down list in the **Limit** rollout. Next, select the **Subtract** option from the **Boolean** drop-down list.
6. Accept the other default settings and choose the **OK** button from the dialog box; two holes are created on the first flange, as shown in Figure 14-121.
7. Similarly, create the remaining holes on other faces of the sheet metal, refer to Figure 14-122. For dimensions of the holes, refer to Figure 14-104. The part after creating the holes is shown in Figure 14-122.

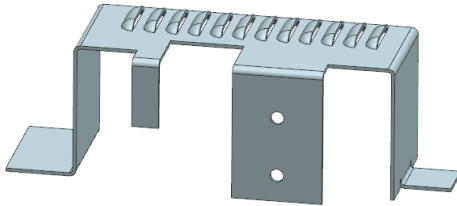


Figure 14-121 Two holes created on the first flange

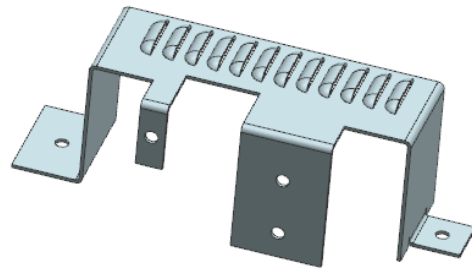


Figure 14-122 Sheet metal part after creating the remaining holes

Creating the Drawn Cutout on the Right Face of the Model

Next, you need to create the drawn cutout on the right face. The sketch for this cutout is a square that will be drawn in the Sketch in Task environment.

1. Choose the **Drawn Cutout** tool from the **Punch** gallery of the **More** gallery of the **Punch** group in **Home** tab; the **Drawn Cutout** dialog box is displayed; you are prompted to select a planar face or select a section geometry.
2. Select the right face of the model and draw the profile of the drawn cutout, as shown in Figure 14-123.
3. Exit the Sketch in Task environment; the preview of the drawn cutout is displayed along with the **Depth** edit box.
4. Enter **4** in the **Depth** edit box and select the **Material Outside** option from the **Side Walls** drop-down list.

5. Expand the **Drawn Cutout** dialog box and enter **4** in the **Die Radius** edit box of the **Settings** rollout.
6. Clear the **Blend Section Corners** check box and choose the **OK** button from the dialog box; the drawn cutout is created.

The final sheet metal part is created, as shown in Figure 14-124.

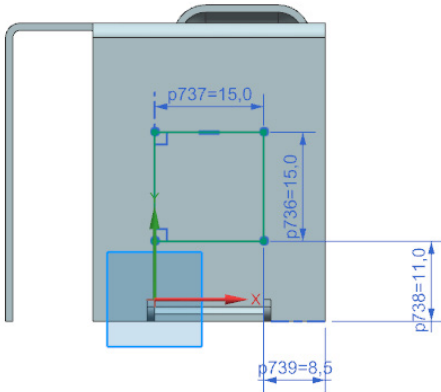


Figure 14-123 Profile of the drawn cutout

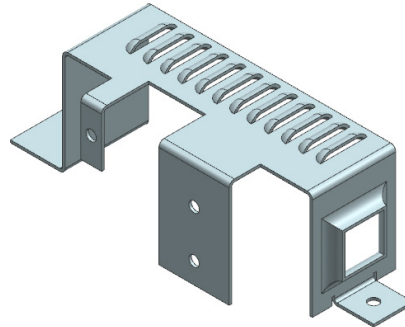


Figure 14-124 The final sheet metal part

Generating the Flat Pattern

Next, you need to generate the flat pattern of the sheet metal part created.

1. Select the **Flat Pattern** tool from the **Flat Pattern** group of the **Home** tab; the **Flat Pattern** dialog box is displayed. Also, you are prompted to click on the face to be oriented upward.
2. Select the top face of the sheet metal part.
3. Next, choose the **OK** button; the flat pattern is created. Note that the flat pattern is not displayed in the current view layout. You need to change the view layout to display the flat pattern.
4. Choose **Menu > View > Layout > Replace View** from the **Top Border Bar**; the **Replace View with** dialog box is displayed.
5. Select **FLAT-PATTERN#1** from the list box available in the **Replace View with** dialog box and then choose the **OK** button; the flat pattern is displayed, as shown in Figure 14-125.

Saving the Model

1. Save the model with the name *c14tut1.prt* at the location given next and then close the file.

|NX|c14

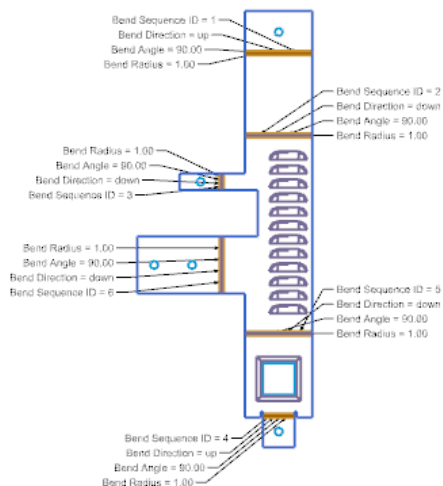


Figure 14-125 Flat pattern of the sheet metal part

Tutorial 2

In this tutorial, you will create the sheet metal component shown in Figure 14-126. The flat solid of the component is shown in Figure 14-127. The dimensions of the component are shown in Figure 14-128. The material thickness, bend radius, relief depth, and relief width is 1 mm each. Save the component with the name *c14tut2.prt* at the location given next:

|NX|c14

(Expected time: 30 min)

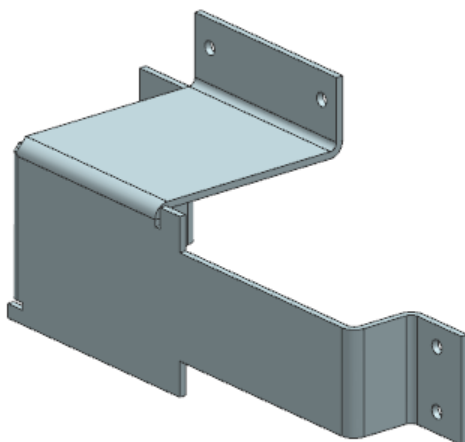


Figure 14-126 Sheet metal part for Tutorial 2

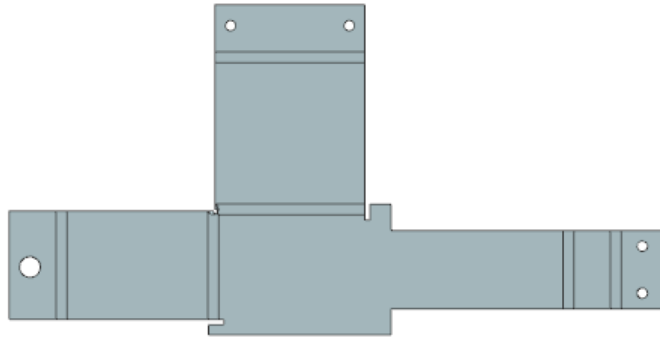


Figure 14-127 Flat solid of the part

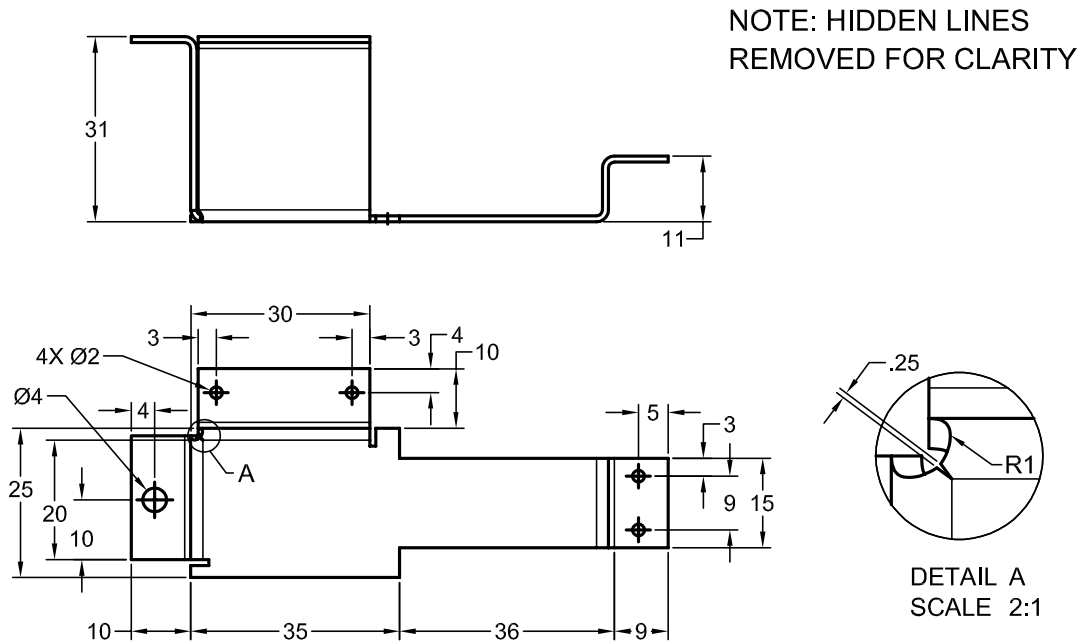


Figure 14-128 Dimensions of the sheet metal part

The following steps are required to complete this tutorial:

- Start a new sheet metal file and set the sheet metal parameters.
- Create the base feature of the model by using the **Tab** tool.
- Create the flanges on the front face base feature.
- Close the corners of the flanges.
- Create the holes.
- Generate the flat solid of the sheet metal part.
- Save the model.

Starting a New Sheet Metal File

1. Choose the **New** tool from the **Standard** group or choose **Menu > File > New** from the **Top Border Bar**; the **New** dialog box is displayed.
2. Select the **Sheet Metal** template from the **Templates** rollout.
3. Enter **c14tut2** as the name of the document in the **Name** text box of the dialog box.
4. Choose the button on the right side of the **Folder** text box; the **Choose Directory** dialog box is displayed. 
5. In this dialog box, browse to **NX/c14** and then choose the **OK** button twice; the new file is started in the Sheet Metal environment.
6. Choose **Menu > Preferences > Sheet Metal** from the **Top Border Bar**; the **Sheet Metal Preferences** dialog box is displayed.
7. Choose the **Part Properties** tab and set the value of the material thickness, bend radius, relief depth, and relief width to 1mm each and choose the **OK** button to apply it to the model.

Creating the Base Feature

1. Choose the **Tab** tool from the **Base** group of the **Home** tab; the **Tab** dialog box is invoked and you are prompted to select a planar face to sketch or select the section geometry.
2. Select the **X-Z** plane as the sketching plane and invoke the sketch environment.
3. Draw the sketch for the base feature of the part, as shown in Figure 14-129.
4. Exit the sketching environment and specify the thickness of the sheet metal part in the backward direction. Exit the **Tab** tool to create the base of the sheet metal part, as shown in Figure 14-130.

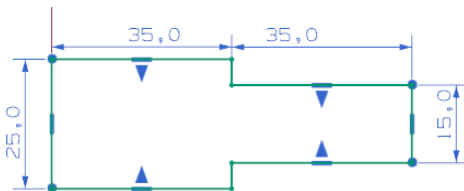


Figure 14-129 Sketch for the base feature

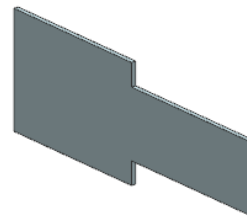


Figure 14-130 Base of the part

Creating the Flanges

Next, you need to create flanges on the front face of the sheet metal part. These flanges will be created using the **Flange** tool.

1. Choose the **Flange** tool from the **Base** group of the **Home** tab; the **Flange** dialog box is displayed and you are prompted to select a linear edge.
2. Select the horizontal edge to create the flange, refer for Figure 14-131; the preview of the flange is displayed. Now, select the **At End** option from the **Width Option** drop-down list. Then, select the left endpoint of the top edge to locate the flange.
3. Specify the width of the flange as **30** mm in the **Width** edit box; the preview of the flange gets modified. Make sure that the flange is created in backward direction. Use the **Reverse Direction** button from the **Flange Properties** rollout, if the flange is displayed in the forward direction.
4. Enter **30** in the **Length** edit box of the **Flange Properties** rollout. Figure 14-131 shows the preview of the flange after modifying the width. Choose the **OK** button to create the flange.

Now, you need to create a flange on the left face of the feature.

5. Select the left vertical edge and specify the length of the flange as **30 mm**.
6. Select the **At End** option from the **Width Option** drop-down list and select the top endpoint of the edge to position the flange. Next, click to define the side of flange and modify its width to **20** mm in the preview.
7. Select the **Bend/Face** option from the **Corner Relief** drop-down list in the **Relief** sub-rollout and choose the **OK** button to exit the dialog box; the second flange is created, as shown in Figure 14-132.

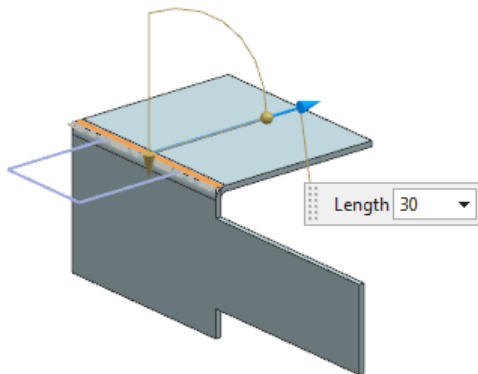


Figure 14-131 Preview of the top flange

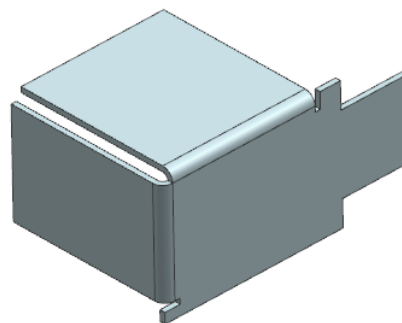


Figure 14-132 Preview of the left flange

8. Next, create the remaining flanges, as shown in Figure 14-133. For dimensions, refer to Figure 14-128.

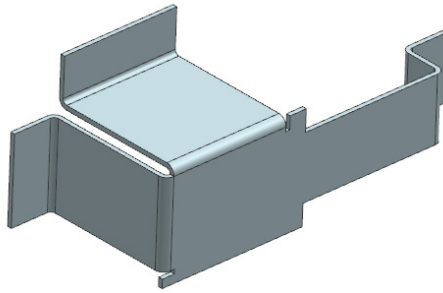


Figure 14-133 Model after creating all flanges

Closing the Corner between the First Two Flanges

The corner between the first two flanges needs to be closed. This is done using the **Closed Corner** tool.

1. Choose the **Closed Corner** tool from the **Corner** group of the **Home** tab; the **Closed Corner** dialog box is displayed and you are prompted to select the bend face adjacent to corner.
2. Select the bends of the first two flanges created to define the corner to be closed.
3. Select the **Circular Cutout** option from the **Treatment** drop-down list in the **Corner Properties** rollout. Specify the gap value as **0.25 mm** and the diameter as **2 mm**.
4. Choose the **OK** button to close the corner. The part after closing the corner is shown in Figure 14-134.

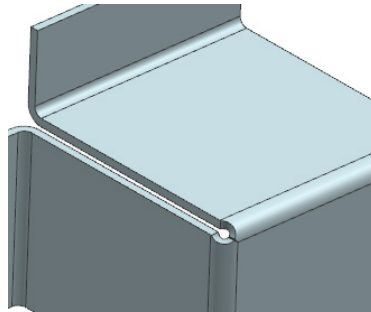


Figure 14-134 Partial view of the part after closing the corner

Creating Holes

1. Choose the **Hole** tool from **Menu > Insert > Design Feature** in the **Top Border Bar**; the **Hole** dialog box is displayed. Change the dimensions of the holes and create the holes on the left most flange. For dimensions of the holes, refer to Figure 14-128. Similarly, create the other holes. The final sheet metal part after creating the holes is shown in Figure 14-135(a). Figure 14-135(b) shows the final sheet metal part in trimetric view.

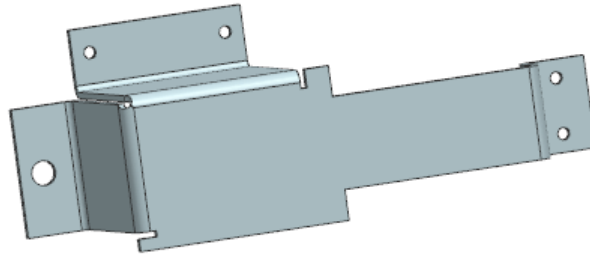


Figure 14-135(a) Final sheet metal part after creating the holes

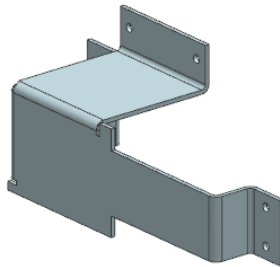


Figure 14-135(b) Final sheet metal part displayed in trimetric view

Generating the Flat Solid

Next, you need to generate the flat pattern of the sheet metal part.

1. Select the **Flat Solid** tool from the **Flat Pattern** group of the **Home** tab; the **Flat Solid** dialog box is displayed. Also, you are prompted to click on a face that you want to remain stationary in the flat solid.
2. Select the front face of the sheet metal part and then choose the **OK** button; the flat solid of the model is created, as shown in Figure 14-136. Next, hide the sheet metal part.

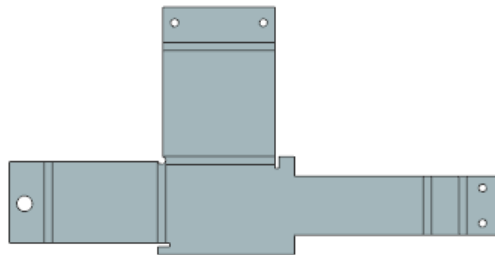


Figure 14-136 Flat solid of the sheet metal part

Saving the Model

1. Save the model with the name *c14tut2.prt* at the location given next and then close the file.

|NX|c14

Self-Evaluation Test

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

1. You can unbend a sheet metal part by using the _____ tool.
2. The unbent sheet metal parts can be bent again using the _____ tool.
3. In NX, the sheet metal parts are bent using a sketched _____.
4. NX allows you to close the corner between two flanges using the _____ tool.
5. To create the flange at the center of the selected edge, you need to choose the _____ option from the **Width Option** drop-down list in the **Flange** dialog box.
6. To convert a solid model into a sheet metal component, you first need to _____ it.
7. The sheet metal files are saved as the *.prt files. (T/F)
8. When you invoke a new sheet metal file, the sketching environment is invoked by default. (T/F)
9. NX allows you to sketch a user-defined shape and then use it to create a dimple in the sheet metal component. (T/F)
10. You can draw a closed or an open sketch for creating the dimple feature. (T/F)

Review Questions

Answer the following questions:

1. Which of the following is the shape of a louver?
(a) Lanced (b) Formed
(c) Both (a) and (b) (d) None
2. Which of the following types is not a type of treatment for a corner?
(a) Open (b) Close
(c) Circular cutout (d) Rectangular cutout

3. Which of the following tools is used to create a base of the sheet metal component?
 - (a) **Flange**
 - (b) **Contour Flange**
 - (c) **Tab**
 - (d) **Louver**
4. Which of the following tools can be used to fillet all corners of the base feature?
 - (a) **Round**
 - (b) **Break Corner**
 - (c) **Chamfer**
 - (d) **Closed Corner**
5. After creating a sheet metal part, you can modify its thickness using the **Part Properties** tab in the **Sheet Metal Preferences** dialog box. (T/F)
6. You need to change the view layout to view the flat pattern. (T/F)
7. You can set the material for the sheet metal part using the **Sheet Metal Preferences** dialog box. (T/F)
8. The height and the width of a louver are inter-related. (T/F)
9. The **Drawn Cutout** tool can be invoked from the **Punch** gallery. (T/F)
10. Beads can be created using the open or closed sketches. (T/F)

EXERCISES

Exercise 1

Create the sheet metal part shown in Figure 14-137. The flat solid of the part is shown in Figure 14-138. Its dimensions are shown in Figure 14-139. The value of material thickness, bend radius, relief depth, and relief width is 0.5 mm each. **(Expected time: 30 min)**

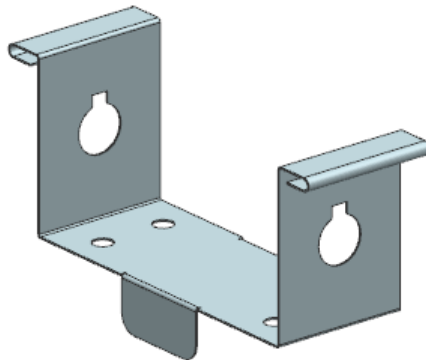


Figure 14-137 Sheet metal part for Exercise 1

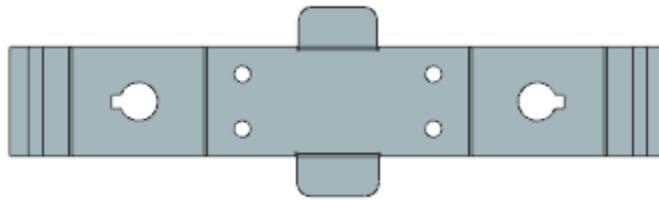


Figure 14-138 Flat solid of the part

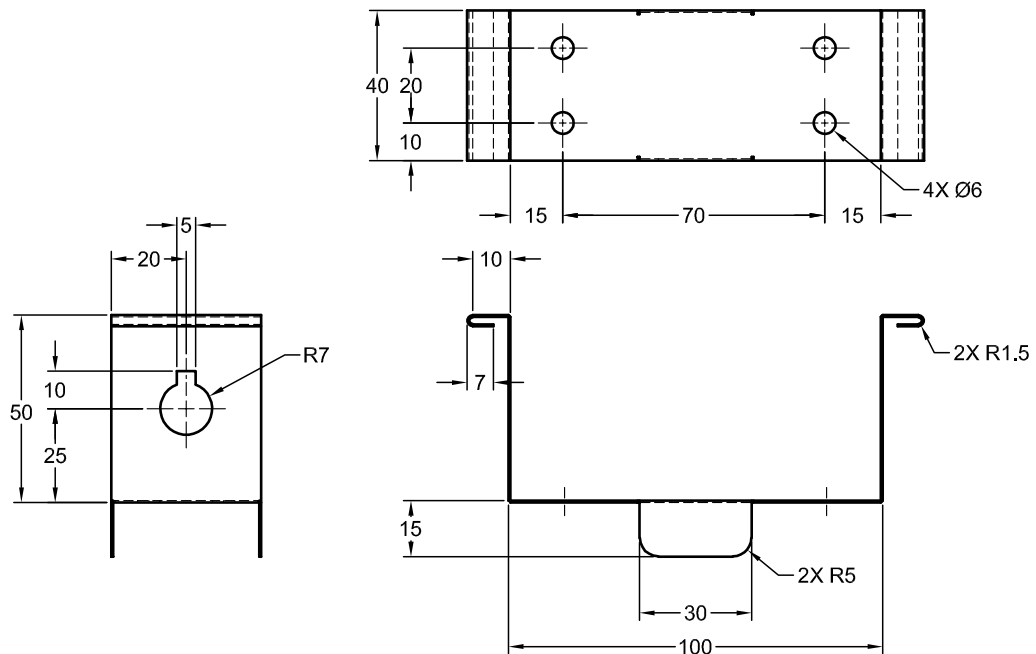


Figure 14-139 Dimensions of the sheet metal part

Exercise 2

Create the Holder Clip part shown in Figure 14-140. The flat solid of the part is shown in Figure 14-141. Its dimensions are shown in Figures 14-142. The value of material thickness, bend radius, relief depth, and relief width is 1 mm. Assume the missing dimensions.

(Expected time: 30 min)

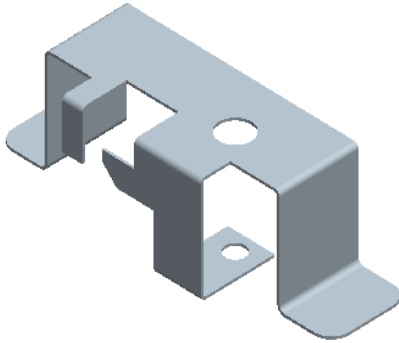


Figure 14-140 The Holder Clip part



Figure 14-141 The flat solid of the Holder Clip part

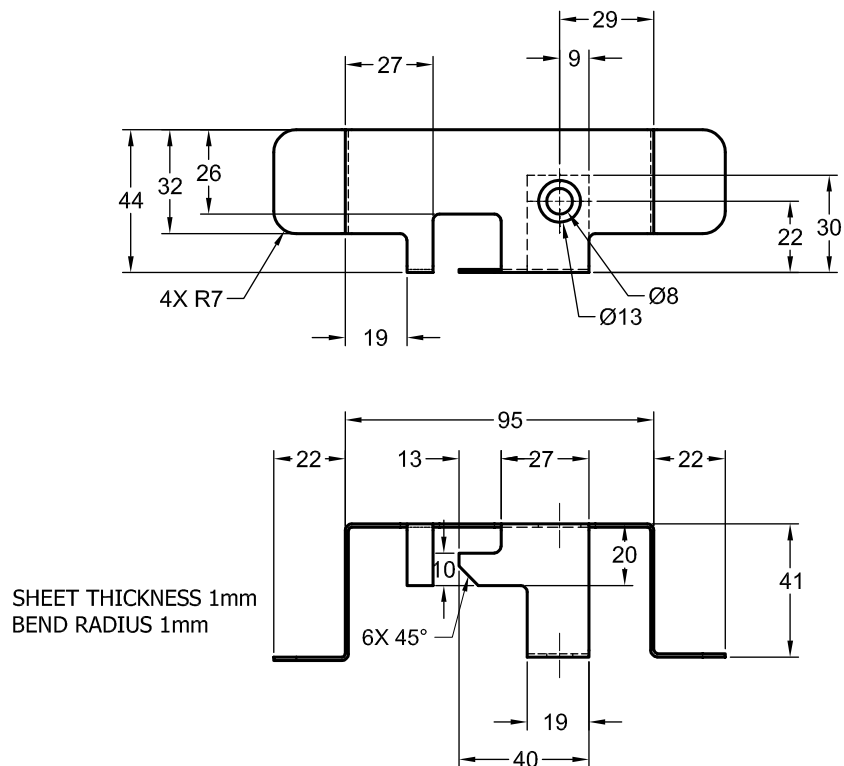


Figure 14-142 Dimensions of the Holder Clip

Answers to Self-Evaluation Test

1. Unbend, 2. Rebend, 3. Line, 4. Closed Corner, 5. At Center, 6. Shell, 7. T, 8. F, 9. T, 10. T