



# Chapter 15

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## Working with Sheet Metal Components

### Learning Objectives

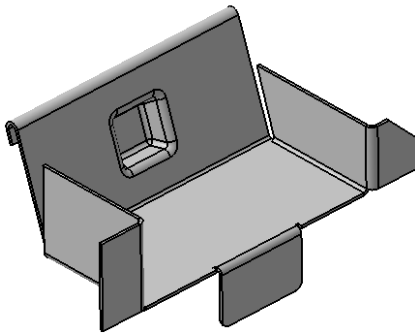
**After completing this chapter, you will be able to:**

- Set the parameters for creating sheet metal parts.
- Create reliefs in the sheet metal component.
- Create base wall and the wall on the edge feature.
- Create wall by extrusion.
- Create flange walls on sheet metal components.
- Create hems on sheet metal components.
- Create teardrop on sheet metal components.
- Create bends on sheet metal components.
- Create flat pattern of the sheet metal components.
- Create a Surface stamp.
- Create a Bead stamp.
- Create a Curve stamp.
- Create a Louver stamp.
- Create a Flange hole.
- Create a Bridge feature.
- Create a Circular stamp.
- Create a Stiffening rib.
- Create a Dowel.

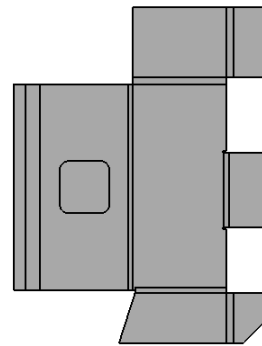
## THE SHEET METAL COMPONENTS

The component having a thickness greater than zero and less than 12 mm is called a sheet metal component. Sheet metal fabrication is a chipless process and is an easy way to create components by using the manufacturing processes such as bending, stamping, and so on.

A sheet metal component of uniform thickness is shown in Figure 15-1. It is not possible to machine such a thin component. After creating the sheet metal component, you need to flatten it in order to find the strip layout. Based on this layout detail, you can design the punch and die. Figure 15-2 shows the flattened view of the sheet metal component shown in Figure 15-1.



*Figure 15-1 Sheet metal component*



*Figure 15-2 Flattened view of the sheet metal component*

CATIA V5R18 allows you to create the sheet metal components in a separate environment called the **Generative Sheetmetal Design**.

### Starting a New File in Generative Sheetmetal Workbench

To start a new file in the **Generative Sheetmetal Design** workbench, choose **Start > Mechanical Design > Generative Sheetmetal Design** from the menu bar; the **New Part** dialog box will be displayed. Choose the **OK** button from the dialog box; a new file will be started in the **Generative Sheetmetal Design** workbench. Figure 15-3 shows the initial screen appearance of the **Generative Sheetmetal Design** workbench.



**Tip.** For the ease of locating and invoking tools, you can create customized toolbars and add the tools that you often use in those customized toolbars. To do so, choose **Tools > Customize** from the menu bar; the **Customize** dialog box will be displayed. Choose the **Toolbars** tab and then the **New** button; the **New Toolbar** dialog box will be displayed. Enter **Generative Sheetmetal Design** as the name of the toolbar in the **Toolbar Name** edit box and choose **OK**; the toolbar is added in the toolbar list. Next, you need to customize the tools. Choose the **Commands** tab from the **Customize** dialog box. From the **Categories** area, select the **All Commands** option; all commands will be displayed in the **Commands** area of the **Customize** dialog box. Select the tool you want to add and drag it to the **Generative Sheetmetal Design** toolbar.

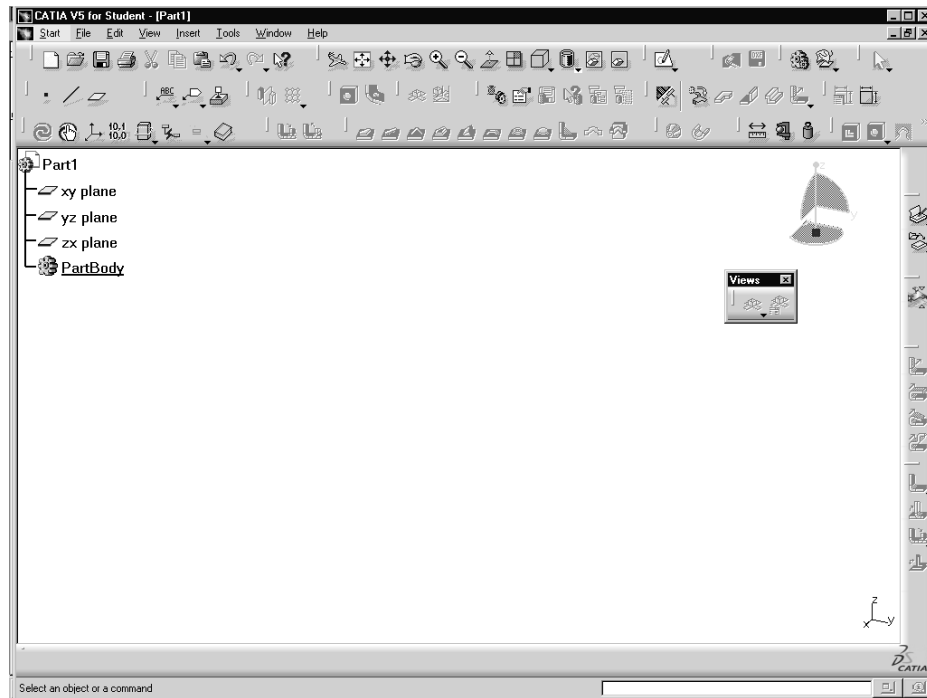


Figure 15-3 The initial screen appearance of the *Generative Sheetmetal Design* workbench

## SETTING SHEET METAL PARAMETERS

<b>Menu:</b>	Insert > Sheet Metal Parameters
<b>Toolbar:</b>	SmdNewDesign > Sheet Metal Parameters



After invoking the **Generative Sheetmetal Design** environment, it is recommended that you first set the sheet metal parameters. To do so, choose the **Sheet Metal Parameters** button from the **SmdNewDesign** toolbar; the **Sheet Metal Parameters** dialog box will be displayed, as shown in Figure 15-4. The parameters set in this dialog box will be used for all additional features. The options in the **Sheet Metal Parameters** dialog box are discussed next.

### Parameters Tab

The options in this tab are used to set the parameters related to the sheet thickness and bend radius. These options are discussed next.

#### Thickness

The **Thickness** spinner is used to specify the sheet thickness. The value that you set in this spinner will be displayed as the default value while creating the sheet metal part.

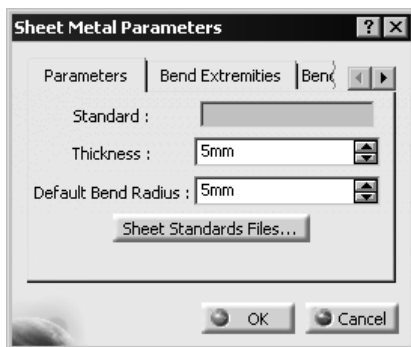
## Default Bend Radius

The **Default Bend Radius** spinner is used to set the radius of the bend. The default bend radius is the internal radius and is linked by default to the supporting walls of the sheetmetal part.

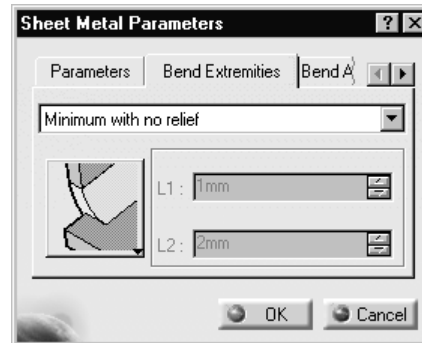
## Bend Extremities Tab

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.

Choose the **Bend Extremities** tab to define the type of relief, as shown in Figure 15-5. Select an option from the drop-down list under this tab to define the relief type. Alternatively, choose the down arrow on the button available underneath; a flyout will be displayed with different types of relief. Select the required relief from the flyout. The types of relief in CATIA V5 are discussed next.



**Figure 15-4** The *Sheet Metal Parameters* dialog box with the **Parameters** tab chosen



**Figure 15-5** The *Sheet Metal Parameters* dialog box with the **Bend Extremities** tab chosen

### Minimum with no relief



This option is selected by default, and it does not provide any relief to the common area between the supporting walls of the sheet metal part, refer to Figure 15-6.

### Square relief

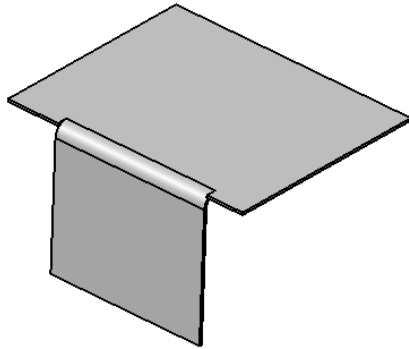


This option provides a square relief between the supporting walls of the sheet metal part, as shown in Figure 15-7. You can modify the L1 and L2 values in their respective spinners.

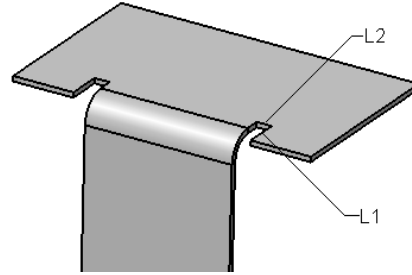
### Round relief



This option provides a round relief between the supporting walls of the sheet metal part, as shown in Figure 15-8. You can modify the L1 and L2 values in their respective spinners.



*Figure 15-6 Sheet metal with no relief*

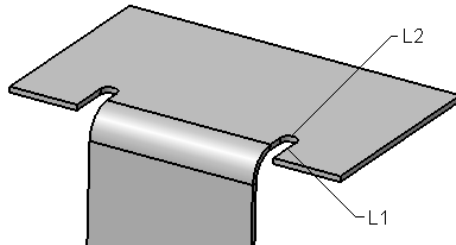


*Figure 15-7 Sheet metal with square relief*

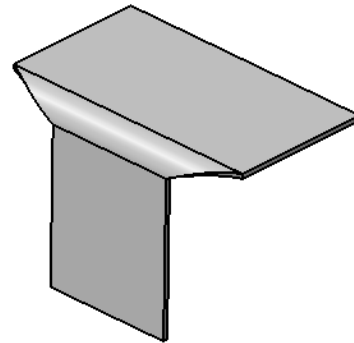
### Linear



The **Linear** option provides a linear relief between the supporting walls of the sheetmetal, as shown in Figure 15-9.



*Figure 15-8 Sheet metal with round relief*



*Figure 15-9 Sheet metal with linear relief*

### Tangent



The **Tangent** option provides a tangent relief between the supporting walls of the sheetmetal, as shown in Figure 15-10.

### Maximum

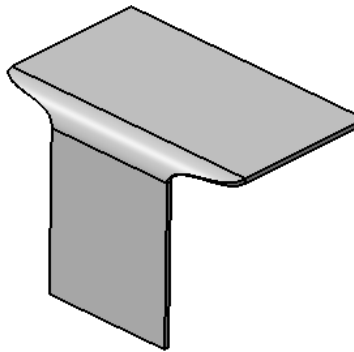


On selecting the **Maximum** option, the bend is calculated between the extreme edges of two support walls, as shown in Figure 15-11.

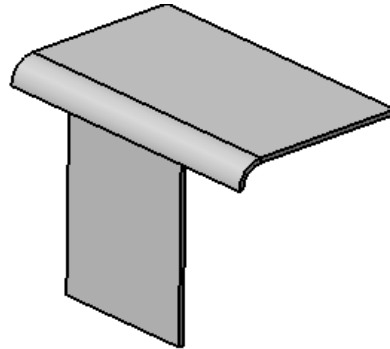
### Closed



The **Closed** option provides relief to the intersection between the bends of two supporting walls.



**Figure 15-10** Sheet metal part with tangent relief

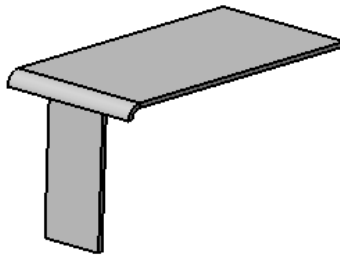


**Figure 15-11** Sheet metal part with maximum relief

### Flat joint



The **Flat joint** option provides flat relief to the intersection between the bends of two supporting walls, as shown in Figure 15-12.



**Figure 15-12** Sheet metal part with Flat joint relief



#### Note

You can also first draw the sketch of the base feature in the **Sketcher** workbench and then set the parameters in the **Sheet Metal Parameters** dialog box.

### Bend Allowance Tab

The option available in the **Bend Allowance** tab is discussed next.

### K Factor

K factor is the ratio between the distance from the neutral bend line and the upper surface of the sheet metal part to the total thickness of the part.

**Note**

You can change the values of parameters in the **Sheet Metal Parameters** dialog box any time during the design.

The tools in this workbench will be available only after setting the parameters in the **Sheet Metal Parameters** dialog box.

## INTRODUCTION TO SHEETMETAL WALLS

A wall refers to any section in the sheetmetal design. There are two types of walls in CATIA V5 and these are discussed next.

### Creating the Base Wall

**Menu:** Insert > Walls > Wall  
**Toolbar:** SmdNewDesign > Wall



The first feature created while designing a sheet metal part is called base wall. The other features are added to this base wall.

Remember that the parameters of the base wall and additional features will be based on the parameters set earlier in the **Sheet Metal Parameters** dialog box. The **Wall** tool is used to create the base feature of the sheetmetal component. To create a wall, choose the **Wall** button from the **SmdNewDesign** toolbar; the **Wall Definition** dialog box will be displayed, as shown in Figure 15-13. You can draw the sketch of the base feature before invoking the **Wall Definition** dialog box. Alternatively, you can draw the sketch of the base feature after invoking the **Wall Definition** dialog box. To do so, choose the **Sketch** button on the right of the **Profile** selection area in the **Wall Definition** dialog box; you will be prompted to select the sketching plane. Select the sketching plane and draw the sketch of the base feature in the **Sketcher** workbench, as shown in Figure 15-14. Next, exit the **Sketcher** workbench to redisplay the **Wall Definition** dialog box.

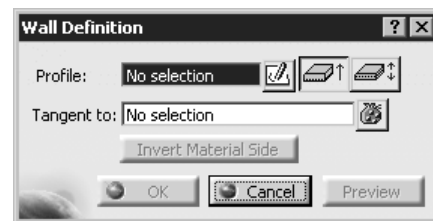


Figure 15-13 The **Wall Definition** dialog box

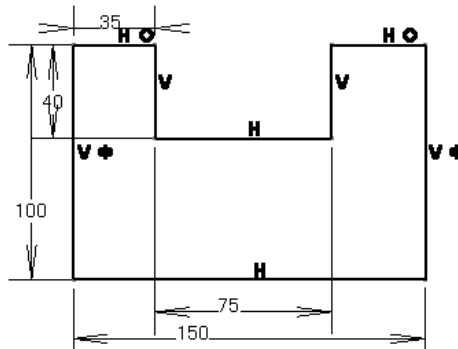


Figure 15-14 The sketch of the base feature

If the sketch is already drawn, select the sketch from the geometry area; the preview of the wall will be displayed. You can flip the the direction of the material by choosing the **Invert Material Side** button from the dialog box. The **Sketch at extreme position** button on the right of the **Profile** selection area in the **Wall Definition** dialog box is chosen by default. This button allows you to set the sketch at the extreme position of the wall thickness. To set the sketch at the middle position of the wall thickness, choose the **Sketch at middle position** button on the right of the **Sketch at extreme position** button. Next, choose the **OK** button to create the base wall. Figure 15-15 shows the resulting base wall.

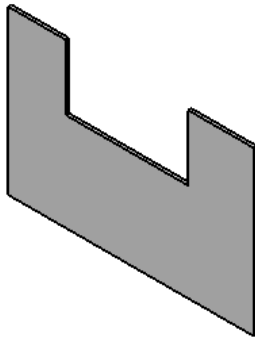


Figure 15-15 The resulting base feature

## Creating the Wall On Edge

**Menu:** Insert > Walls > Wall On Edge  
**Toolbar:** SmdNewDesign > Wall On Edge



The **Wall On Edge** tool is used to create a wall on the edges of an existing wall. To do so, choose the **Wall On Edge** button from the **SmdNewDesign** toolbar; the **Wall On Edge Definition** dialog box will be displayed, as shown in Figure 15-16. In this dialog box, the **Type** drop-down list has two options, **Automatic** and **Sketch Based**. These options are discussed next.

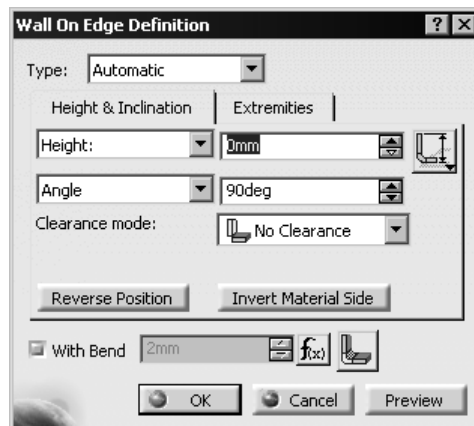


Figure 15-16 The **Wall On Edge Definition** dialog box



## Automatic

The **Automatic** option lets you select an existing edge to create a wall. The options that will be displayed on selecting the **Automatic** option from the **Type** drop-down list are discussed next.

### Height & Inclination Tab

By default, the **Height** option is selected in the first drop-down list under the **Height & Inclination** tab. Set the value of the height in the spinner. To define the height, click on the down arrow available on the **Length type** button; a flyout will be displayed with four options. These options are discussed next.



This option allows you to specify the vertical distance from the lower face of the base wall to the extreme edge of the wall created.



This option allows you to specify the vertical distance from the upper face of the base wall to the extreme edge of the wall created.



This option is used to specify the height of the flange from the start of the bend to the extreme edge of the wall created.



This option allows you to specify the slant distance from the extreme edge of the vertical wall to the apparent intersection of the vertical and horizontal wall.

If you need the wall on the edge to be created upto a pre-defined plane, select the **Up to Plane/Surface** option from the first drop-down list and then select the plane from the geometry area or the specification tree. You can also create a new plane. To do so, right-click in the **Up to Plane/Surface** selection area; the contextual menu will be displayed. Choose the **Create Plane** option from the contextual menu and create a plane as discussed earlier. While creating the wall on edge using the **Up to Plane/Surface** option, if you select the plane from the geometry area, the **Offset** spinner will be available in the dialog box. You can use this spinner to terminate the wall at a certain offset from the selected plane.

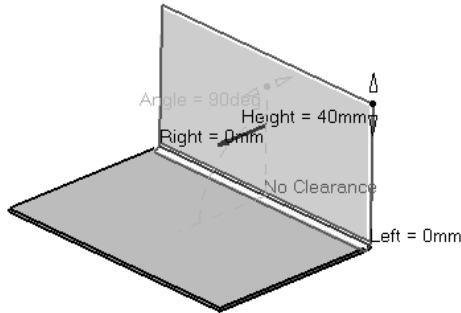
To create a wall at an angle with respect to the support wall, select the **Angle** option, if it is not already selected, in the second drop-down list and set the inclination in the **Angle** spinner.

If you need to create an inclined wall with respect to the existing face or plane, select the **Orientation plane** option from the second drop-down list. Select the plane as discussed earlier. If you need to change the inclination of the wall with respect to the selected plane, set the angle in the **Rotation angle** spinner.

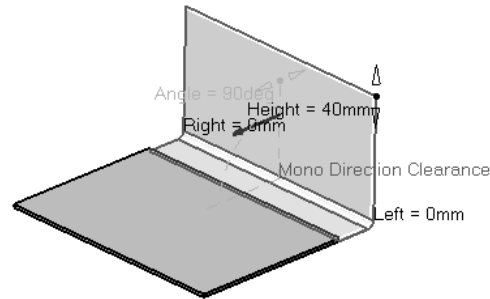
### Clearance mode

The wall can be created at an offset from the selected edge by using the options in the **Clearance mode** drop-down list. The options in this drop-down list are **No Clearance**, **Monodirectional**, and **Bidirectional**. On selecting the **No Clearance** option, the wall will be created without the clearance between the base wall and the

side wall, as shown in Figure 15-17. When you select the **Monodirectional** option, the wall will be created with the horizontal clearance, as shown in Figure 15-18. You can set the value for the horizontal clearance in the **Clearance value** spinner.

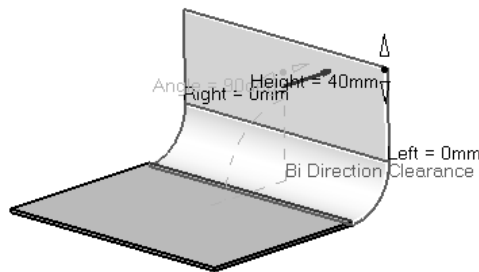


**Figure 15-17** Side wall with no clearance



**Figure 15-18** Side wall with the horizontal clearance

On selecting the **Bidirectional** option, the **Feature Definition Warning** message box will be displayed. If you choose the **Yes** button, the **Clearance value** spinner will not be available and the clearance value will be the same as set for the bend radius in the **Sheet Metal Parameters** dialog box. When you choose the **No** button from the message box, the **Clearance value** spinner will be available and you can change its value. The **Bidirectional** option provides clearance in the horizontal and vertical directions, as shown in Figure 15-19.



**Figure 15-19** Side wall with the bidirectional clearance

To change the direction of the wall on the edge, choose the **Reverse Position** button from the **Wall On Edge Definition** dialog box. To change the material side of the wall, choose the **Invert Material Side** button from the dialog box. Note that the **Invert Material Side** button will not be available, if the **Bidirectional** option is selected in the **Clearance mode** drop-down list.

### Extremities Tab

The options in the **Extremities** tab are used to define the length of the wall with respect

to a reference plane/wall. The options and the procedure to set the limits are discussed next.

**Left limit**

Select the predefined plane or the wall that you need to set as the left limit from the geometry area; the name of the selected plane will be displayed in the **Left limit** selection area. You can also create a plane as discussed earlier.

**Left offset**

The **Left offset** spinner is used to set the offset distance from the plane that is selected as the left limit in the **Left limit** selection area. If there is no selection in the **Left limit** selection area, the endpoint on the left side of the edge is selected as the left limit, by default.

**Right limit**

Select the predefined plane or the wall that you need to set as the right limit from the geometry area; the name of the selected plane will be displayed in the **Right limit** selection area. You can also create a plane as discussed earlier.

**Right offset**

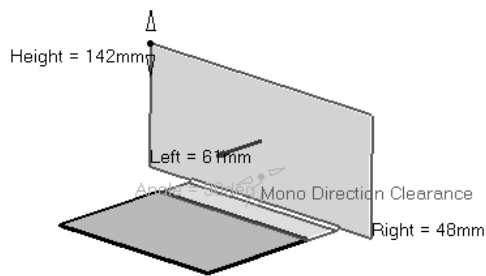
The **Right offset** spinner is used to set the offset distance from the plane that is selected as the right limit in the **Right limit** selection area. If there is no selection in the **Right limit** selection area, the endpoint on the right side of the edge is selected as the right limit, by default.

**With Bend**

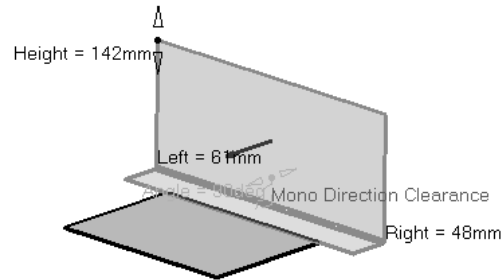
The **With Bend** check box is used to create a wall with or without the bend. Select the **With Bend** check box, if it is not already selected, and the wall will be created with bend. To change the type of relief, choose the **Bends parameters** button on the right of the **With Bend** check box; the **Bend Definition** dialog box will be displayed. In this dialog box, you can change the type of relief as discussed earlier. The **Left Extremity** and **Right Extremity** tabs in the **Bend Definition** dialog box can be used to specify the type of relief for the left and right ends of the resulting wall.

If you have selected the **Monodirectional** option from the **Clearance mode** drop-down list in the **Wall On Edge Definition** dialog box and the horizontal clearance set is more than the bend radius, the **Extrapolation mode** button will be available next to the **Bends parameters** button. Click on the down arrow of the **Extrapolation mode** button; a flyout will be displayed with two buttons. If you choose the first button, the width of the clearance will be equal to the width of the base feature, as shown in Figure 15-20. On choosing the second button, the width of the clearance will be equal to the width of the wall on the edge feature, as shown in Figure 15-21.

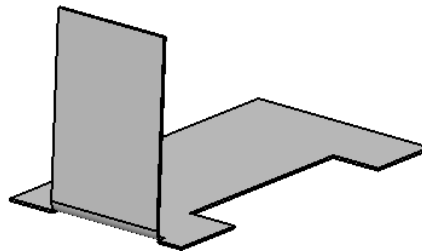
Choose the **OK** button; the resultant side wall will be created, as shown in Figure 15-22.



**Figure 15-20** Preview of the wall to be created when the first button is chosen from the **Extrapolation mode** flyout




**Figure 15-21** Preview of the wall to be created when the second button is chosen from the **Extrapolation mode** flyout



**Figure 15-22** Wall created by selecting the **Automatic** option



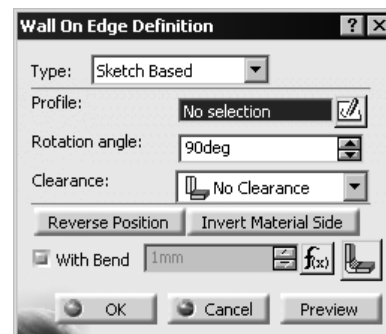
**Tip.** On choosing the **Swap limits** symbol  displayed on the left of the spinners, the values and limits on the left and right sides of the wall will get interchanged.

## Sketch Based

Select the **Sketch Based** option in the **Type** drop-down list; the **Wall On Edge Definition** dialog box will be modified, as shown in Figure 15-23. The options in this dialog box are discussed next.

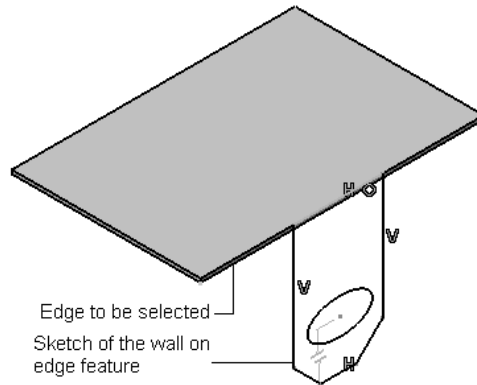
### Profile

Select the edge of the base wall feature on which you need to create a wall, as shown in Figure 15-24. Next, choose the

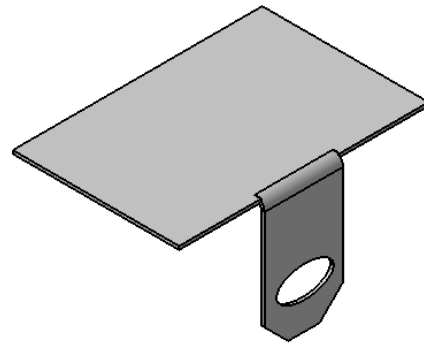


**Figure 15-23** The **Wall On Edge Definition** dialog box with the **Sketch Based** option selected

**Sketch** button on the right of the **Profile** selection area; you will be prompted to select the sketching plane. Select the face adjacent to the selected edge. This face will act as the sketching plane of the wall feature. After selecting the sketching plane, system takes you to the **Sketcher** workbench. Create a closed sketch in the **Sketcher** workbench, as shown in Figure 15-24. Next, exit the **Sketcher** workbench. The preview of the feature will be displayed in the geometry area. Choose the **OK** button; the wall on the edge will be created, as shown in Figure 15-25.



**Figure 15-24** Edge and Sketch for the wall on edge feature



**Figure 15-25** Wall on edge created using the **Sketch Based** option

### Rotation angle

You can also create a wall inclined at an angle to the selected edge. To do so, set the angular value in the **Rotation angle** spinner. The default value in the **Rotation angle** spinner is **90deg**.

The other options in this dialog box have been discussed earlier.



### Note

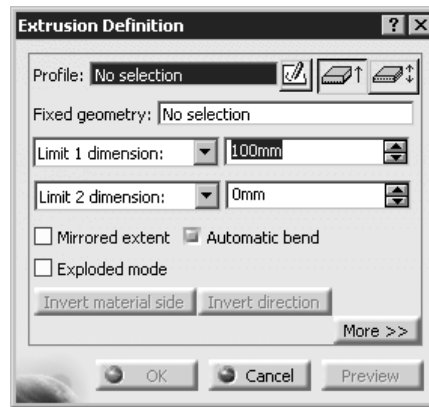
*If you have created the wall without bend, you can create the bend later using the **Bend** tool. This tool will be discussed later in this chapter.*

## CREATING EXTRUSIONS

**Menu:** Insert > Walls > Extrusion  
**Toolbar:** SmdNewDesign > Extrusion



The **Extrusion** tool is used to extrude an open sketch drawn on an existing wall. To extrude the sketch, choose the **Extrusion** button from the **SmdNewDesign** toolbar; the **Extrusion Definition** dialog box will be displayed, as shown in Figure 15-26. Also, you will be prompted to select a profile. Select the existing profile (sketch) from the geometry area to extrude it. If there is no existing profile in the geometry area, then you need to create a sketch using the **Sketch** button available on the right of the **Profile** selection area, as discussed earlier. The options in the **Extrusion Definition** dialog box are discussed next.



*Figure 15-26 The Extrusion Definition dialog box*

By default, the **Limit 1 dimension** option is selected in the first drop-down list and the **Limit 2 dimension** option is selected in the second drop-down list of the **Extrusion Definition** dialog box. Therefore, the wall will extrude normal to the base wall.

You can specify the length of the wall to be extruded in the limit 1 and limit 2 directions by using the **Length 1** and **Length 2** spinners that are available on the right of the first and second drop-down lists, respectively.

To extrude the sketch in the limit 1 direction up to a selected plane or surface, select the **Limit 1 up to plane** or **Limit 1 up to surface** option from the first drop-down list. You can also extrude the sketch in the limit 2 direction up to a selected plane or surface. To do so, select the **Limit 2 up to plane** or **Limit 2 up to surface** option from the second drop-down list. Next, select the plane or surface from the geometry area.

The options in the second drop-down list are used to extrude the wall on the other side of the base wall.

The **Automatic bend** check box in the **Extrusion Definition** dialog box is selected by default. This check box allows you to create bends on the sharp vertices of the sketch. To mirror the extruded feature, select the **Mirrored extent** check box. But note that on selecting this check box, the first and second drop-down lists will not be available. The **Sketch at extreme position** button located on the right of the **Profile** selection area is chosen by default. So, the material is added above the sketch. If you choose the **Sketch at middle position** button, the sketch will be considered as a neutral plane and the material will be added to both sides of the sketch. You can change the direction of the material by choosing the **Invert Material Side** button from the dialog box. To preview the feature, you can choose the **Preview** button from this dialog box. For specifying the K factor on the bend, you need to expand the dialog box by choosing the **More** button.

Figure 15-27 shows the sketch of the extrusion wall and Figure 15-28 shows the resulting extruded wall.

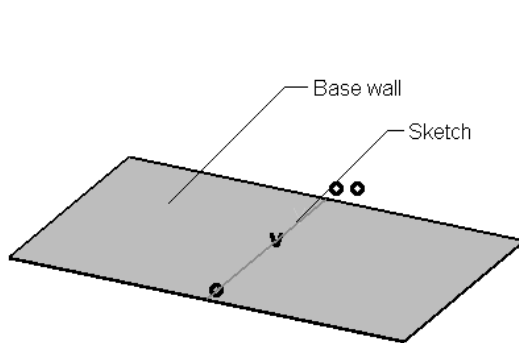


Figure 15-27 Sketch of the extrusion wall

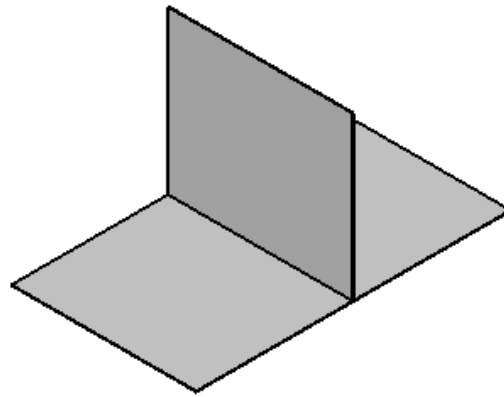


Figure 15-28 Resulting extruded wall

**Note**

You need to create the bend after extruding the profile. This will be discussed later in this chapter.

## CREATING SWEEP WALLS

**Menu:** Insert > Walls > Swept Walls > Flange  
**Toolbar:** SmdNewDesign > Flange



Swept walls are created by sweeping a profile along the selected edge. The different types of swept walls are **Flange**, **Hem**, **Tear Drop**, and **User Flange** swept walls. These walls are discussed next.

### Creating Flanges on the Sheet Metal Component

Flange is the bend section of the sheet metal. To create a flange feature, choose the **Flange** button from the **Swept Walls** toolbar; the **Flange Definition** dialog box will be displayed, as shown in Figure 15-29. The options in this dialog box are discussed next.

#### Flange type

There are two options in the **Flange type** drop-down list and these are discussed next.

#### Basic

The **Basic** option in the **Flange Type** drop-down list is selected by default and in this case, the width of the flange created will be equal to the width of the selected edge. The options available on selecting the **Basic** option from the **Flange type** drop-down list are discussed next.

#### Length

Set the length of the wall in the **Length** spinner. To change the length type of the flange, choose the down arrow on the right side of the **Length type** button; a flyout will be displayed with four options. The options in the flyout have been discussed earlier.

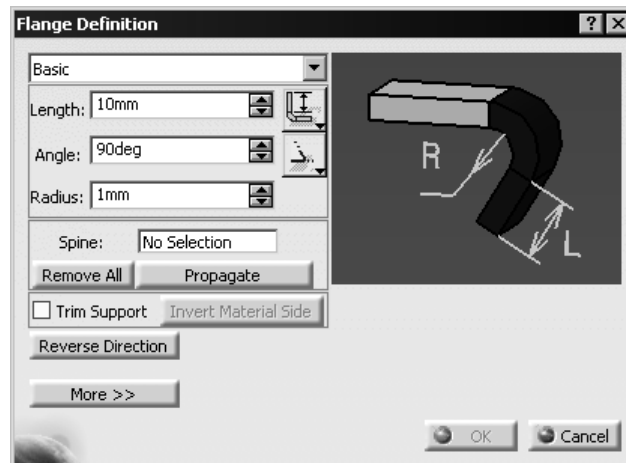


Figure 15-29 The *Flange Definition* dialog box

### Angle

You can also create a flange wall inclined at an angle to the selected edge. To do so, set the angular value in the **Angle** spinner. To change the angle type of the flange, choose the down arrow on the right of the **Angle type** button; a flyout will be displayed with two buttons. If you choose the **Inner Angle type** button from the flyout, the angle that you define will be considered as the included angle from the face. If you choose the **Outer Angle type** button, the angle that you define will be considered as the excluded angle.

### Radius

Set the bend radius of the flange in the **Radius** spinner.

### Spine

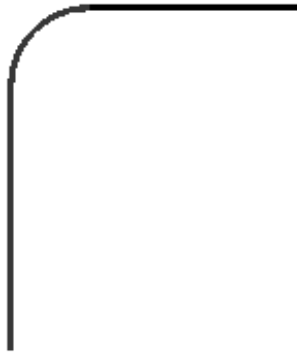
Select an edge from the existing wall; the selected entity will be displayed in the **Spine** selection area. To remove the selected entity from the **Spine** selection area, choose the **Remove All** button available below this selection area.

### Trim Support

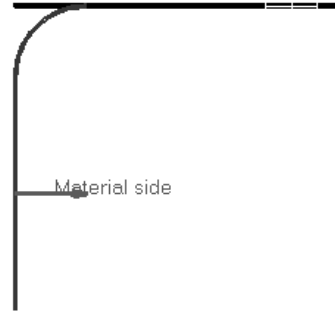
By default, this check box is not selected. So, the flange will be created outward from the edge selected, as shown in Figure 15-30. When you select the **Trim Support** check box, the flange will be created inward from the selected edge at a distance equal to the radius of the flange, as shown in Figure 15-31. Also, the portion of the wall beyond the radius will be trimmed.

Choose the **Propagate** button to select the edges that are connected tangentially. Choose the **Invert Material Side** button to change the material side of the flange. The **Invert Material Side** button will be available when the **Trim Support** check box is selected.





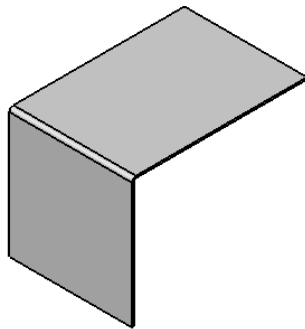
*Figure 15-30* Flange created with the **Trim Support** check box cleared



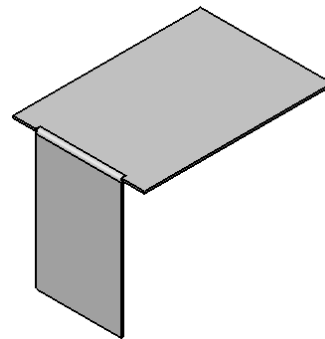
*Figure 15-31* Flange created with the **Trim Support** check box selected

### Relimited

To create the flange up to a specified limit, select the **Relimited** option from the **Flange type** drop-down list. On selecting this option, you need to define the limits in the **Limit 1** and the **Limit 2** selection area. You can select the predefined limits from the geometry area or create the new one by right-clicking in the **Limit 1** and **Limit 2** selection area as discussed earlier. The rest of the options are the same as discussed in the **Basic** option. Figure 15-32 shows the flange feature created by selecting the **Basic** option. Figure 15-33 shows the flange feature created by selecting the **Relimited** option.



*Figure 15-32* The flange feature created by selecting the **Basic** option



*Figure 15-33* The flange feature created by selecting the **Relimited** option

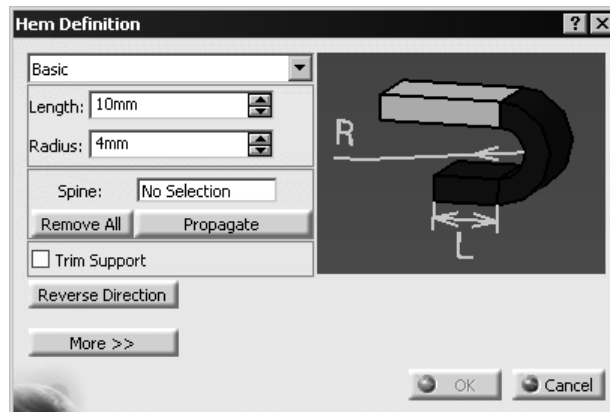
## Creating Hems on the Sheet Metal Component

**Menu:** Insert > Walls > Swept Walls > Hem

**Toolbar:** Swept Walls > Hem



Hem is the rounded face created on the sharp edges of the sheet metal component in order to reduce the sharpness in a sheet metal component. This makes the sheet metal component easy to handle and assemble. To create hems, choose the **Hem** button from the **Swept Walls** toolbar; the **Hem Definition** dialog box will be displayed, as shown in Figure 15-34. The options in the **Hem Definition** dialog box are discussed next.

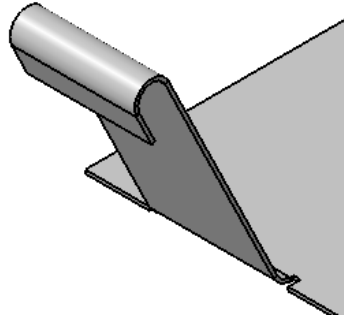


*Figure 15-34 The Hem Definition dialog box*

By default, the **Basic** option is selected in the **Flange type** drop-down list, so the width of the hem created will be equal to the width of the selected edge. On selecting the **Relimited** option from the **Flange type** drop-down list, you need to select the limits from the geometry area. Next, you need to set the length and radius of the hem in their respective spinners.

Select the edge, also known as the guide element, on which you need to create the hem. The selected guide element will be displayed in the **Spine** selection area. To remove the selected guide element, choose the **Remove All** button.

To select the edges that are connected tangentially, choose the **Propagate** button. To change the material side of the flange, choose the **Reverse Direction** button. Choose the **OK** button to create the final feature. Figure 15-35 shows the Hem on a side wall.



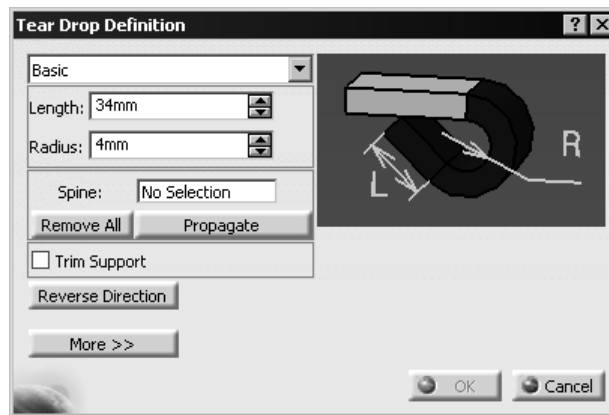
*Figure 15-35 The Hem created on the side wall*

## Creating a Tear Drop on the Sheet Metal Component

**Menu:** Insert > Walls > Swept Walls > Tear Drop  
**Toolbar:** Swept Walls > Tear Drop



A tear drop is similar to a hem but in this case, the flat wall is inclined at an angle. Choose the **Tear Drop** button from the **Swept Walls** toolbar; the **Tear Drop Definition** dialog box will be displayed, as shown in Figure 15-36. The options in the **Tear Drop Definition** dialog box are discussed next.



*Figure 15-36 The Tear Drop Definition dialog box*

By default, the **Basic** option is selected in the **Flange type** drop-down list, so the width of the tear drop feature created will be equal to the width of the selected edge. On selecting the **Relimited** option from the **Flange type** drop-down list, you need to select the limits from the geometry area. Next, you need to set the length of the flat wall and radius of the tear drop in their respective spinners.

Select the edge on which you need to create the tear drop. The selected edge will be displayed in the **Spine** selection area. To remove the selected edge, choose the **Remove All** button.

To select the edges that are connected tangentially, choose the **Propagate** button. To change the material side of the flange, choose the **Reverse Direction** button. Choose the **OK** button to create the final feature. Figure 15-37 shows the tear drop feature created on a side wall.

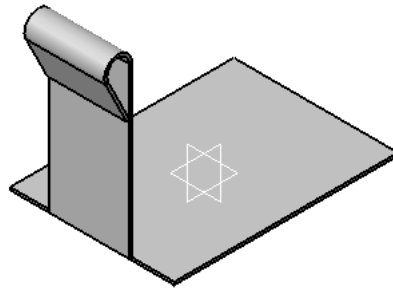


Figure 15-37 Side wall with a tear drop feature created

## Creating a User Flange on the Sheet Metal Component

**Menu:** Insert > Walls > Swept Walls > User Flange  
**Toolbar:** Swept Walls > User Flange



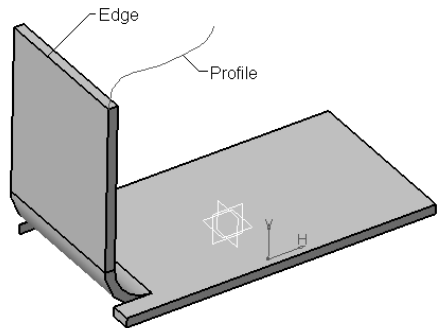
User flanges are created by sweeping an open sketch along an edge. Choose the **User Flange** button from the **Swept Walls** toolbar; the **User-Defined Flange Definition** dialog box will be displayed, as shown in Figure 15-38. The options in this dialog box are discussed next.

By default, the **Basic** option is selected in the **Flange type** drop-down list, so the width of the flange created will be equal to the width of the selected edge. On selecting the **Relimited** option from the **Flange type** drop-down list, you need to select the limits from the geometry area. Next, select the profile from the geometry area; the selected profile will be displayed in the **Profile** selection area. Note that the profile must be tangent to the supporting wall. Next, select an edge along which the profile has to be swept. To remove the selected edge from the **Spine** selection area, choose the **Remove All** button. To select the edges that are connected tangentially, choose the **Propagate** button. Choose the **OK** button to create the feature.

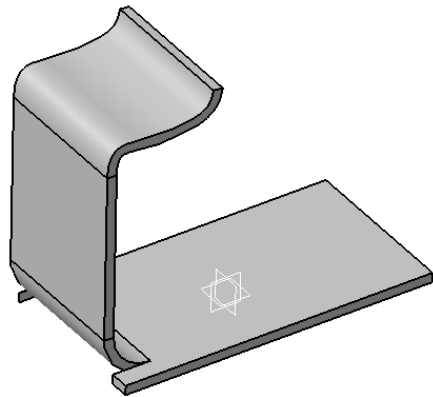


Figure 15-38 The *User-Defined Flange Definition* dialog box

Figure 15-39 shows the profile and the edge selected for creating the user flange and Figure 15-40 shows the resulting user flange feature.



**Figure 15-39** The edge and the profile selected for creating the user flange



**Figure 15-40** The resulting user flange feature created

# CREATING A BEND

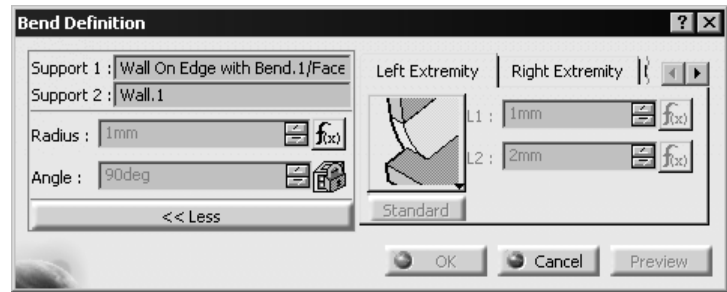
**Menu:** Insert > Bending > Bend  
**Toolbar:** Bends > Bend



This tool is used to create a bent face between the intersection of two walls. To create a bent face, choose the **Bend** button from the **Bends** toolbar; the **Bend Definition** dialog box will be displayed. The procedure to create a bend, and the options in the **Bend Definition** dialog box are discussed next.

Select two walls in succession from the geometry area; the selected walls will be displayed in the **Support 1** and **Support 2** selection areas. The bend radius and the angle value that were set in the **Sheet Metal Parameters** dialog box will be displayed as the default values in the **Radius** and **Angle** spinners of the **Bend Definition** dialog box, respectively.

To change the type of relief provided to the bend, choose the **More** button; the **Bend Definition** dialog box will get expanded, as shown in Figure 15-41. The **Left Extremity** tab in the dialog box is chosen by default. The options in this tab are used to provide relief to



**Figure 15-41** The expanded form of the **Bend Definition** dialog box

the left side of the wall. Select the down arrow on the **Select the extremity type** button; a flyout will be displayed with various relief options. Select the type of relief from the options in the flyout. Next, choose the **Right Extremity** tab and select the type of relief to be provided on the right side of the wall.

Choose the **Preview** button to preview the feature, as shown in Figure 15-42. Choose the **OK** button; the final bend feature will be created, as shown in Figure 15-43.

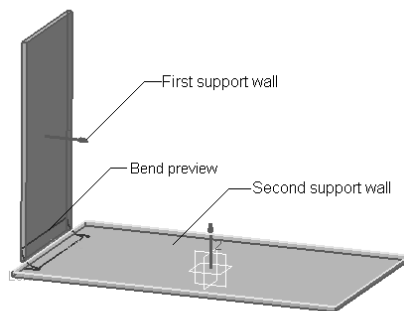


Figure 15-42 Preview of the bend feature

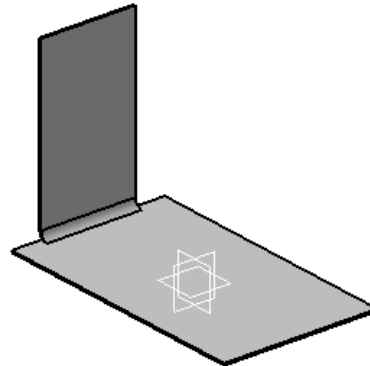


Figure 15-43 The resulting bend feature



**Tip.** You can create a bend even if the two support walls are not intersecting, provided the distance between the two walls is equal to or less than the default bend radius set in the **Sheet Metal Parameters** dialog box.

## Creating A Conical Bend

**Menu:** Insert > Bending > Conical Bend  
**Toolbar:** Bends > Conical Bend



The **Conical Bend** tool is used to create a bend of variable radius at the intersection of two walls.

The bend created using this tool will have a conical shape and varying radius from left to right of the wall, or vice-versa. To create a conical bent face, choose the **Conical Bend** button from the **Bends** toolbar; the **Bend Definition** dialog box will be displayed, as shown in Figure 15-44.

Choose two support walls in succession from the geometry area; the selected walls will be displayed in the **Support 1** and **Support 2** selection areas, respectively. Set the radius values in the **Left radius** and **Right radius** spinners, respectively. To add or

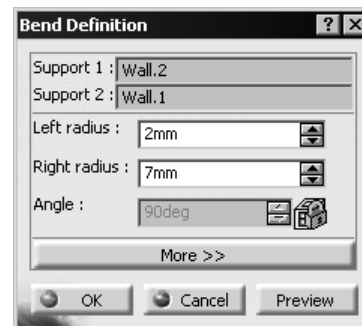


Figure 15-44 The Bend Definition dialog box

modify the type of relief in the bend, choose the **More** button; the **Bend Definition** dialog box will be expanded. The **Left Extremity** tab is chosen by default. Now, select the type of relief from the flyout of the **Select the extremity type** button. Next, choose the **Right Extremity** tab and select the type of relief for the right side of the wall.

You can see the preview of the sheet metal feature by choosing the **Preview** button from the **Bend Definition** dialog box, as shown in Figure 15-45. Choose the **OK** button; the resulting conical bend feature will be created, as shown in Figure 15-46.

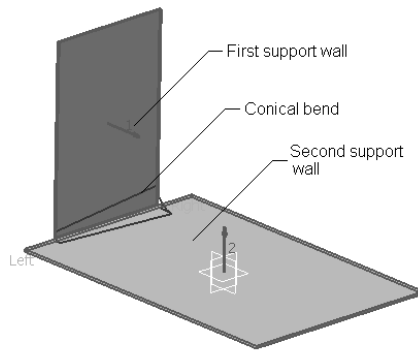


Figure 15-45 Preview of the conical bend feature

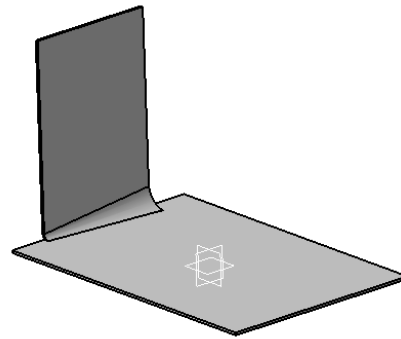


Figure 15-46 Resulting conical bend feature

## BEND FROM FLAT

**Menu:** Insert > Bending > Bend From Flat  
**Toolbar:** pBending > Bend From Flat



CATIA V5 allows you to bend a part of the sheet metal component by using the **Bend From Flat** tool. Remember that the sheet metal part will be folded along the non-continuous lines. Choose the **Bend From Flat** button from the **pBending** toolbar; the **Bend From Flat Definition** dialog box will be displayed, as shown in Figure 15-47. The options in the **Bend From Flat Definition** dialog box are discussed next.

### Profile

Select the sketch from the geometry area and it will be displayed in the **Profile** selection area. The selected sketch can contain more than a line, but these should be noncontinuous ones. You can also create the profile by using the **Sketch** button in the **Bend From Flat Definition** dialog box.

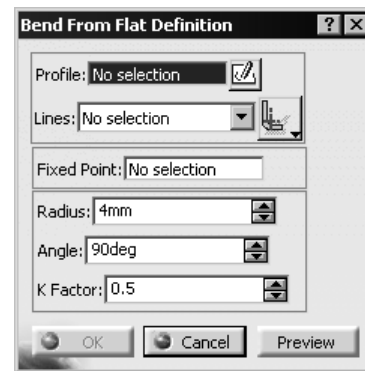


Figure 15-47 The **Bend From Flat Definition** dialog box

## Lines

If the selected profile has more than a line, then all lines will be listed in the **Lines** drop-down list. Select the line along which you need to create the bend from the **Lines** drop-down list. Next, select the down arrow on the button at the right of the **Lines** drop-down list; a flyout will be displayed with four options. The options in the flyout are discussed next.

### Axis



When you choose this button, the bend line will be placed at the centre of the bend and the bend will be created equally in both the directions of the bend line, as shown in Figure 15-48.

### BTL (Bent Tangent Line)



When you choose this button, the bend line will be placed at the start of the bend, as shown in Figure 15-49.

### IML (Inner Mold Line)

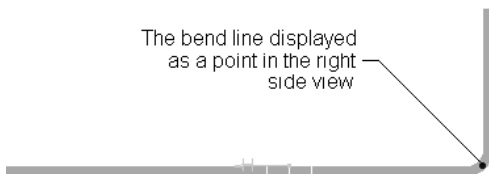


When you choose this button, the bend line will be placed at the apparent intersection of the inner surface of both the horizontal and vertical walls, as shown in Figure 15-50.

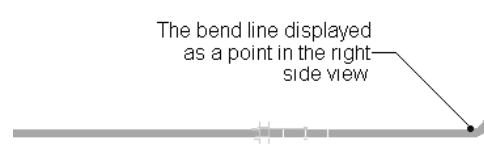
### OML (Outer Mold Line)



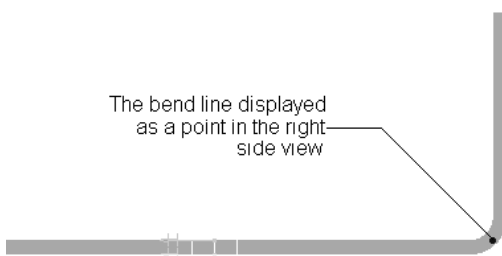
When you choose this button, the bend line will be placed at the apparent intersection of the outer surface of both the horizontal and vertical walls, as shown in Figure 15-51.



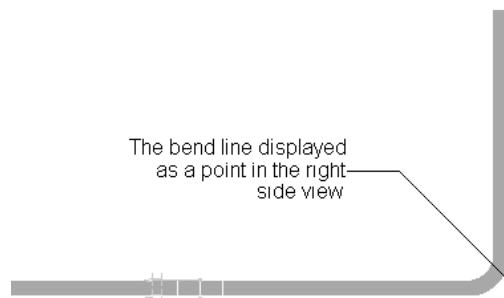
**Figure 15-48** Bend created using the **Axis** option



**Figure 15-49** Bend created using the **Bent Tangent Line** option



**Figure 15-50** Bend created using the **Inner Mold Line** option



**Figure 15-51** Bend created using the **Outer Mold Line** option



## Fixed Point

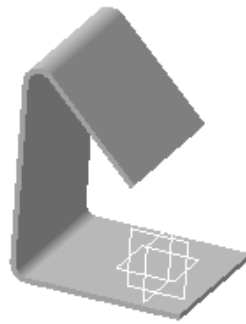
Once you select the sketch, the fixed point is automatically set on the face where the sketch is positioned. This means that no portion of the wall will move when the bend is created. You can change the fixed point by selecting another point on the wall. The fixed point is generally the start point of the sketch. To create a new fixed point, right click on the **Fixed Point** selection area, select the **Create Point** option, and define a point.

## Radius

By default, the value of the bend radius is the value that was set earlier in the **Sheet Metal Parameters** dialog box. You can also change the radius value in the **Radius** spinner.

## Angle

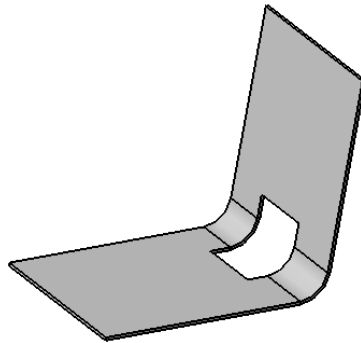
The **Angle** spinner is used to set the angle of the fold for the sheet metal component. The default value in this edit box is 90-degree. You can set a value below 180-degree in this spinner. Figure 15-52 shows the sheet metal part folded by an angle of 90-degree along Line 1 and 45-degree along Line 2. To bend a component using two lines, first you need to create a sketch consisting of two lines in the same sketching plane. Next, invoke the **Bend From Flat Definition** dialog box. In this dialog box, specify the angle value for these lines in their respective **Angle** spinners. Note that if you select **Line 1** in the **Lines** drop-down list, then you can only specify the angle value for Line 1. You can repeat the same procedure for specifying the angle value for the Line 2. To reverse the bending direction, click on the blue colored arrow in the preview and choose the **OK** button from the **Bend From Flat Definition** dialog box.



*Figure 15-52 Sheet metal part after folding it through an angle of 90-degree and 45-degree*

## FOLDING AND UNFOLDING SHEET METAL PARTS

Sometimes, you need to perform operations like cutout, hole, and so on across the bend face of the wall, as shown in Figure 15-53. You cannot perform these operations in bend condition. Therefore, you need to unfold the sheet metal part, add features, and fold it again. The folding and unfolding operations can be carried out using the **Unfolding** and **Folding** tools available in the **pBending** toolbar. These tools are discussed next.



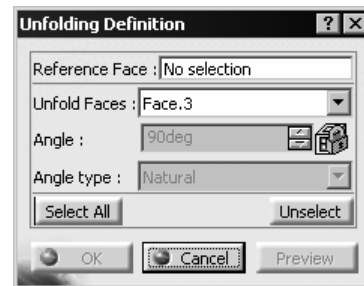
*Figure 15-53 Cutout operation performed across the bend face*

## Unfolding the Sheet Metal Parts

**Menu:** Insert > Bending > Unfolding  
**Toolbar:** Bending > Unfolding

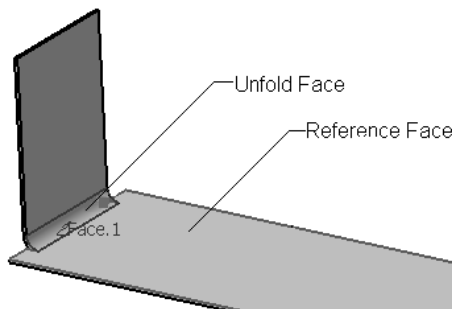


The **Unfolding** tool is used to unfold a bend face of the sheet metal part. Choose the **Unfolding** button from the **Bending** toolbar; the **Unfolding Definition** dialog box will be displayed, as shown in Figure 15-54.

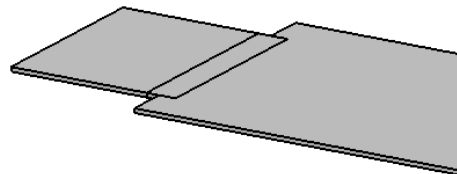


*Figure 15-54 The Unfolding Definition dialog box*

On invoking this dialog box, you will be prompted to select a reference face. Select a wall from the geometry area; it will be displayed in the **Reference Face** selection area. Next, you will be prompted to select a face to unfold. If you need to unfold a particular bend, select it from the drawing area, as shown in Figure 15-55. To unfold all the bends, choose the **Select All** button. Choose the **Unselect** button to clear the bends selected. Next, choose the **OK** button to unfold the part, as shown in Figure 15-56.



*Figure 15-55 The Reference Face and the Unfold Face to be selected*



*Figure 15-56 The unfolded part*

## Folding the Unfolded Parts

**Menu:** Insert > Bending > Folding  
**Toolbar:** Bending > Folding



The **Folding** tool is used to fold the wall back to its original position that was unfolded using the **Unfolding** tool. Choose the **Folding** button from the **Bending** toolbar; the **Folding Definition** dialog box will be displayed, as shown in Figure 15-57. Also, you will be prompted to select a reference face. Select the face of the wall from the geometry area; the selected face will be displayed in the **Reference Face** selection area. Next, select the bend section; it will be displayed in the **Fold Faces** drop-down list. The **Angle** spinner is used to specify the angle upto which the unfolded wall has to be folded. By default, the angular value will be the difference between 180 degrees and the bend angle of the original part. Select an option from the **Angle type** drop-down list to specify the angle type of the resulting folded part. The options in the **Angle type** drop-down list are discussed next.

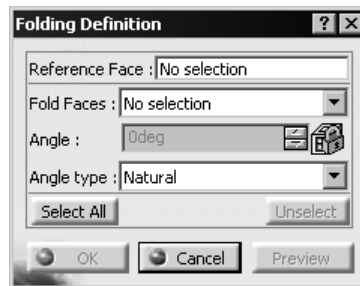


Figure 15-57 The *Folding Definition* dialog box

### Natural

By default, this option is selected and the wall will be folded back to its original position. So, the **Angle** spinner will not be available.

### Defined

Select this option if you need to fold the wall to an angle other than its original position. Specify the angle in the **Angle** spinner.

### Spring back

Select this option if you need to fold the wall at an angle with respect to its original position. Specify the angle in the **Angle** spinner.

By choosing the **Select All** button, you can select all unfold walls. By choosing the **Unselect** button, you can unselect the selected unfold walls. Choose the **Preview** button; the preview of the feature will be displayed, as shown in Figure 15-58. Next, choose the **OK** button to create the final feature, as shown in Figure 15-59.

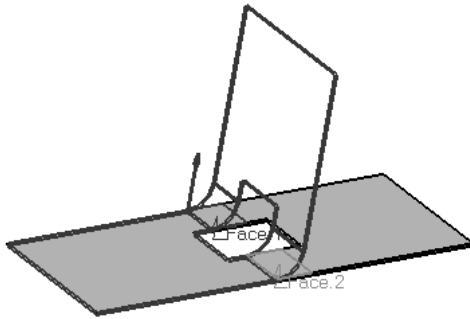


Figure 15-58 Preview of the folded part

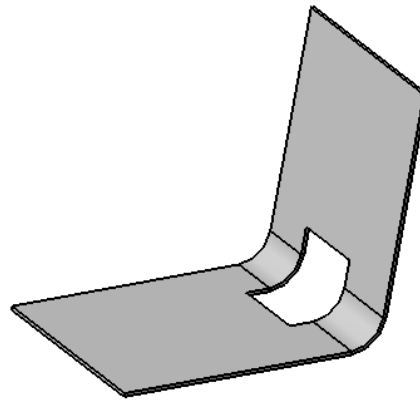


Figure 15-59 Resulting feature of the folded part

**Note**

After performing the required operation, if you want to fold the component, choose the **Select All** button from the **Folding Definition** dialog box.

## Mapping the Geometry

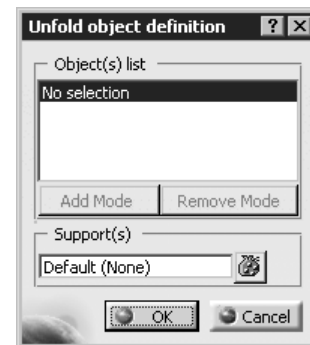
**Menu:** Insert > Bending > Shape Mapping  
**Toolbar:** pBending > Point or curve mapping

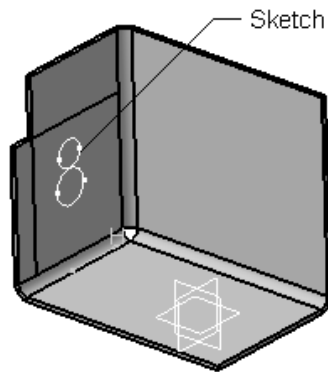


This tool allows you to create a geometrical object, such as a point or a curve in the unfolded view.

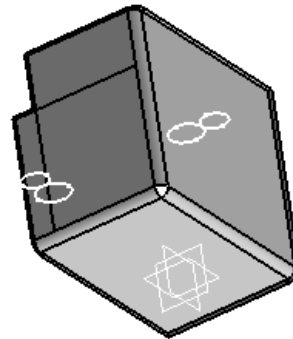
The **Point or curve mapping** tool is specially used for generating a logo, defining an area for chemical milling, or creating a cut on the components. To create a geometrical object, draw the required sketch in the **Sketcher** environment and then invoke the **Unfold object definition** dialog box by choosing the **Point or curve mapping** button from the **pBending** toolbar, refer to Figure 15-60.

Next, select the sketch from the geometry area; the selected sketch will be displayed in the **Object(s) list** selection area and its preview will be displayed in the geometry area. Figure 15-61 shows the sketch for mapping and Figure 15-62 shows the preview of the mapping. The **Add Mode** and **Remove Mode** buttons are used to add and remove elements from the **Object(s) list** selection area. The **Support(s)** selection area is used to select the supported wall with respect to the sketch.

Figure 15-60 The *Unfold object definition* dialog box



*Figure 15-61 Sketch for mapping*



*Figure 15-62 Preview of the mapping*

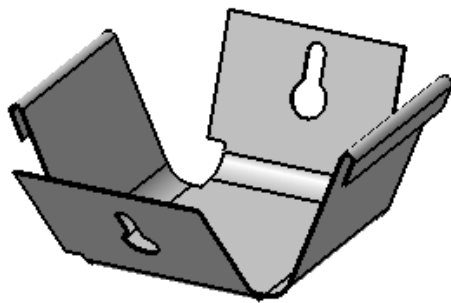
## CREATING FLAT PATTERNS OF SHEET METAL COMPONENTS

**Menu:** Insert > Views > Unfold/Fold  
**Toolbar:** Unfold > Unfold/Fold

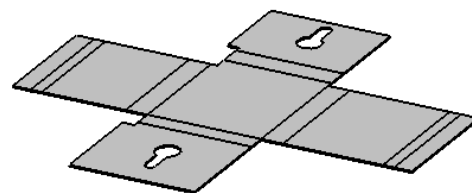


The flattened view of a sheet metal component plays a very important role during the manufacturing stage. While designing the tools for creating the sheet metal component, you need to flatten the sheet metal component.

Choose the **Unfold/Fold** button from the **Unfold** toolbar to view the unfolded sheet metal components. As this is a toggle tool, you can choose it again to fold back the sheet metal components. Note that you cannot make any modifications in the unfolded sheet metal component. Figure 15-63 shows a sheet metal component and Figure 15-64 shows the flat pattern of the same component.



*Figure 15-63 Sheet metal part*



*Figure 15-64 Flat pattern of the sheet metal part*

## VIEWING A SHEET METAL COMPONENT IN MULTIPLE WINDOWS

**Menu:** Insert > Views > Multi Viewer  
**Toolbar:** Unfold > Multi Viewer



The **Multi Viewer** tool is used to unfold a sheet metal component and display it in another window. To do so, choose the **Multi Viewer** button from the **Unfold** toolbar; the part which is currently folded part will be displayed as the unfolded part in the second window. To view both the windows on the same screen, choose **Window > Tile Horizontally** from the menu bar. If you modify any one view, the modification will be reflected in the other view also.

## USING VIEWS MANAGEMENT

**Menu:** Insert > Views > Views Management  
**Toolbar:** pViews > Views Management



The **Views Management** tool is used to manage the activation or deactivation views. To do so, choose the **Views Management** button from the **pViews** toolbar; the **Views** dialog box will be displayed, as shown in Figure 15-65. Alternatively, right-click on the **PartBody** feature from the specification tree; the contextual menu will be displayed. Choose **PartBody object > Views** from the contextual menu to display the **Views** dialog box.

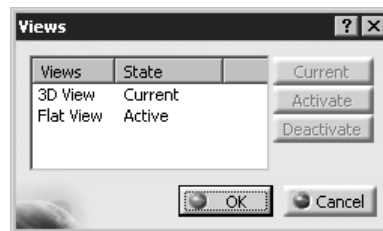


Figure 15-65 The Views dialog box

In this dialog box, the State of the **3D View** is **Current**, which means the part is folded currently. To make the State of the **Flat View** as **Current**, select it from the dialog box; the **Current** and **Deactivate** buttons will be enabled. Next, choose the **Current** button; the part is displayed as flat pattern. If you want to make the folded view again as the current view, first select the **3D View** and then choose the **Current** button. You can activate and deactivate the views by choosing the **Activate** and **Deactivate** buttons from this dialog box.



### Note

You can create holes, fillets, and chamfers features using the tools in the **Cutting/Stamping** toolbar. The procedure to create these features is similar to that of part modeling.

## STAMPING

Stamping is a metal working process in which a sheet is punched in a press tool. This process is used to create stamping features such as blanking, piercing, and forming on the sheet metal part. CATIA V5 provides a set of tools to design a sheet metal part that can be created by the stamping process. These tools are grouped in the **Stamping** toolbar, as shown in Figure 15-66. These tools are discussed next.

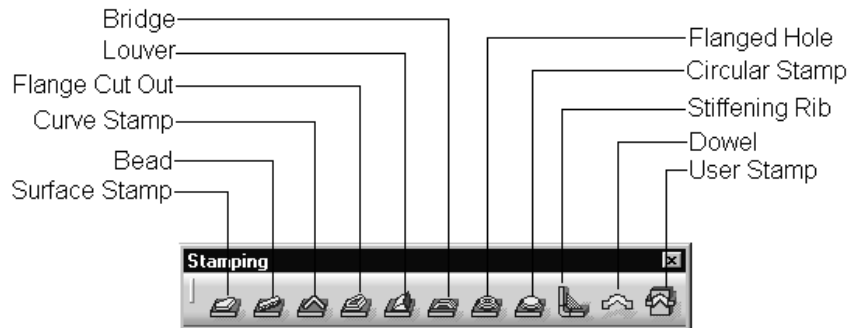


Figure 15-66 The **Stamping** toolbar



**Tip.** You can dislodge the **Stamping** toolbar from the flyout displayed by clicking on the down arrow available on the right of the **Surface Stamp** tool.

### Creating a Surface Stamp

**Menu:** Insert > Stamping > Surface Stamp  
**Toolbar:** Stamping > Surface Stamp



In CATIA V5, you can emboss an area enclosed by a closed profile using the **Surface Stamp** tool. To do so, you need to create a close sketch on a wall, as shown in Figure 15-67. Next, choose the **Surface Stamp** button from the **Stamping** toolbar; the **Surface Stamp Definition** dialog box will be displayed, as shown in Figure 15-68. Select the profile from the geometry area, if it is not already selected; it will be displayed in the **Profile** selection area. The other options in the **Surface Stamp Definition** dialog box are discussed next.

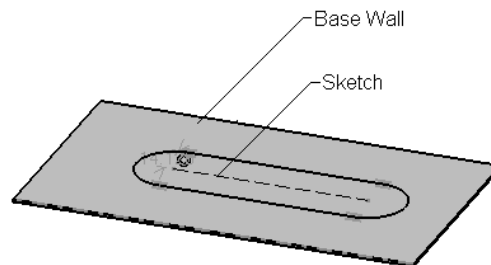


Figure 15-67 Sketch of the surface stamp feature

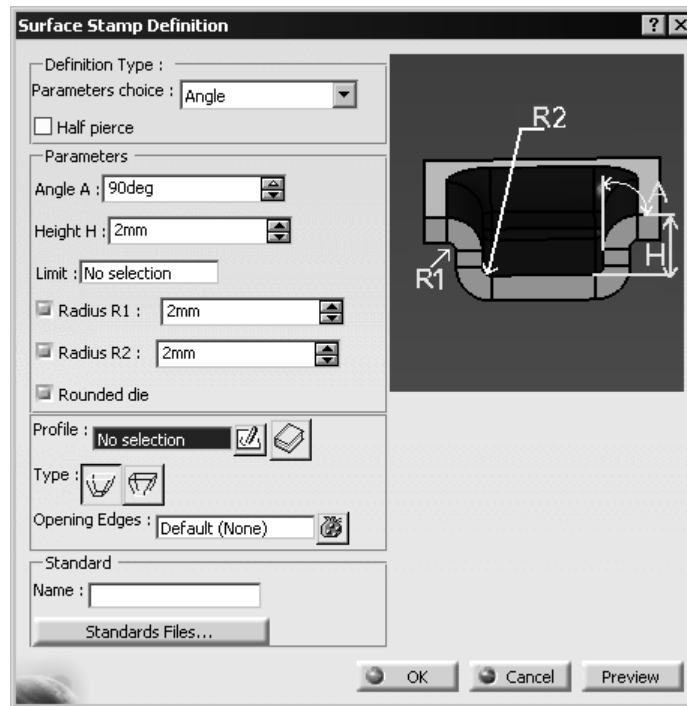


Figure 15-68 The Surface Stamp Definition dialog box

### Definition Type Area

The options in the **Definition Type** area are discussed next.

#### Parameters choice

The three options available in the **Parameters choice** drop-down list are the **Angle**, **Punch & Die**, and **Two profiles**. By default, the **Angle** option is selected in the **Parameters choice** drop-down list.

#### Half pierce

On selecting this check box, the stamp will be created such that its height will be half of the thickness of the sheet metal. If you select this check box, the **Angle A**, **Radius R1**, **Radius R2**, **Rounded die**, and **Opening Edges** options will not be available in the dialog box.

### Parameters Area

When the **Angle** option is selected in the **Parameters choice** drop-down list, the following options are available in the **Parameters** area:

#### Angle A

Set the angular value of the stamp feature in this spinner. The angle will be provided between the wall of the surface stamp and the face where the profile is drawn. The default value in the **Angle A** spinner is **90deg**.



**Height H**

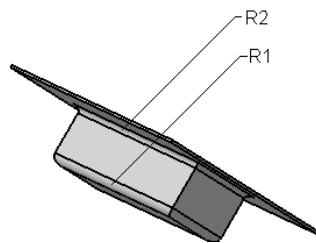
The height of the stamp feature is specified in the **Height H** spinner. Note that this spinner will not be available if you specify the height using the **Limit** option.

**Limit**

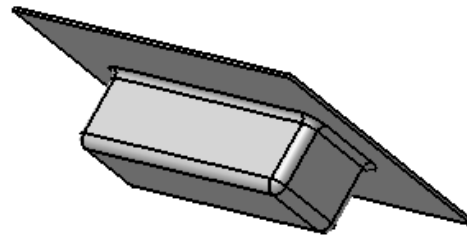
You can also specify the height of the stamp feature by specifying the limits. The limits specified can be a surface or plane. To create a plane, right-click in the **Limit** selection area and then choose the **Create Plane** option from the contextual menu.

**Radius R1, Radius R2, and Rounded Die**

Select the **Radius R1** and **Radius R2** check boxes to create the fillets at the top and bottom surfaces of the stamp, respectively, as shown in Figure 15-69. Set the fillet radii in the **Radius R1** and **Radius R2** spinners. If you select the **Rounded die** check box, the side edges of the stamp feature will be filleted. Figure 15-69 shows the surface stamp feature with the **Rounded die** check box cleared. Figure 15-70 shows the surface stamp feature with the **Rounded die** check box selected.



**Figure 15-69** Stamp feature without side edges filleted



**Figure 15-70** Stamp feature with side edges filleted

**Profile Selection Area**

The **Profile** selection area displays the profile selected for the surface stamp. You can sketch the profile of the surface stamp by using the **Sketch** button available on the right of the **Profile** selection area.

**Type buttons**

When you choose the first button in this area, the dimensions of the top face of the stamp feature will be equal to the sketch selected initially, as shown in Figure 15-71. On choosing the second button, the dimensions of the bottom of the stamp feature will be equal to the sketch selected initially, as shown in Figure 15-72.

**Opening Edges**

You can also create a stamp feature with the open wall. To do so, click in the **Opening Edges** area, select a profile from the sketch; the wall relative to the selected profile will not be created. Figure 15-73 shows the stamp feature with an open wall. To see the preview of the stamp, choose the **Preview** button. Choose the **OK** button from the **Surface Stamp Definition** dialog box; the surface stamp feature will be created, as shown in Figure 15-74.

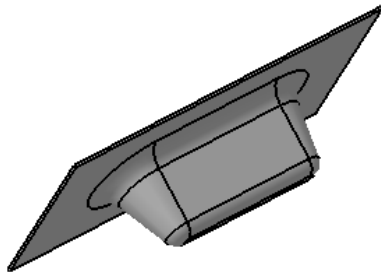


Figure 15-71 Type 1

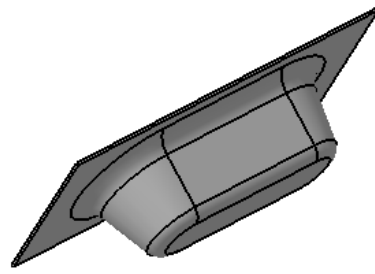


Figure 15-72 Type 2

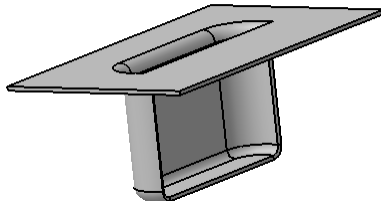


Figure 15-73 The stamp feature with the open wall

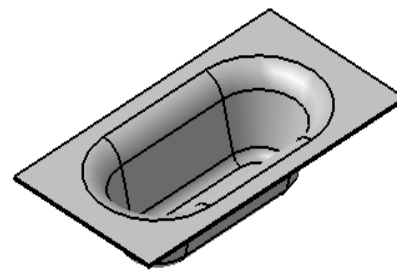


Figure 15-74 The surface stamp feature

## Creating a Bead Stamp

**Menu:** Insert > Stamping > Bead  
**Toolbar:** Stamping > Bead



The **Bead** tool is used to create an embossed or an engraved bead on a sheet metal part along an open sketch. If the sketch has multiple continuous entities, they should be tangentially connected. Else, you will get an error message. Choose the **Bead** button; the **Bead Definition** dialog box will be displayed, as shown in Figure 15-75.

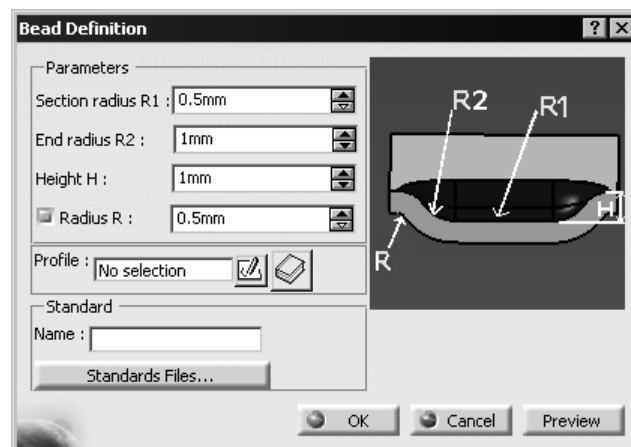
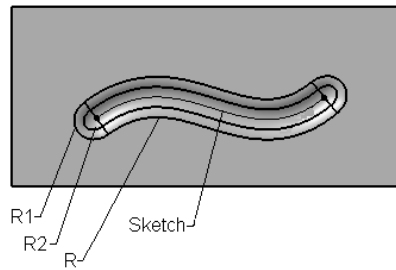
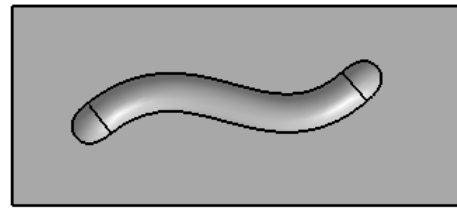


Figure 15-75 The Bead Definition dialog box

You will be prompted to select the profile. Select it from the geometry area. Next, set the values of the section radius and end radius in their respective spinners. Note that the end radius should be greater than the section radius. Set the height of the stamp in the **Height H** spinner. Various parameters of the bead feature are shown in Figure 15-76. If you do not select the **Radius R** check box, the resulting stamp feature will be created without a fillet, as shown in Figure 15-77. You can change the direction of the bead by clicking on the orange pointer in the geometry area. To see the preview of the feature, choose the **Preview** button. Next, choose the **OK** button; the stamping feature will be created.



*Figure 15-76 Parameters of bead feature*



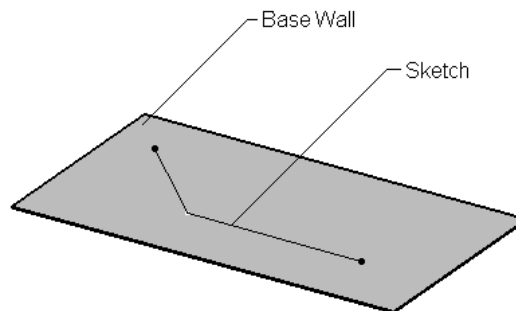
*Figure 15-77 Stamp feature without fillet*

## Creating a Curve Stamp

**Menu:** Insert > Stamping > Curve Stamp  
**Toolbar:** Stamping > Curve Stamp



The **Curve Stamp** tool is used to create an embossed feature along the entities that are not connected tangentially. To create a curve stamp feature, you need to create an open or a closed profile on the wall of the sheet metal component, as shown in Figure 15-78. Next, choose the **Curve Stamp** button from the **Stamping** toolbar; the **Curve stamp definition** dialog box will be displayed, as shown in Figure 15-79. The options in this dialog box are discussed next.



*Figure 15-78 Sketch of a curve stamp*

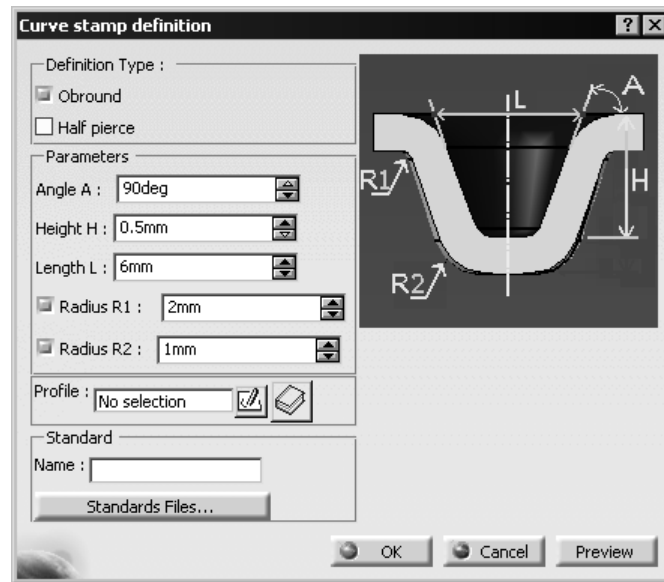


Figure 15-79 The Curve stamp definition dialog box

### Definition Type Area

The options in this area are discussed next.

#### Obround

By selecting the **Obround** check box, you can round-off the end of the edges of the curve stamp shown in Figure 15-80. Figure 15-81 shows the curve stamp feature with the **Obround** check box cleared.

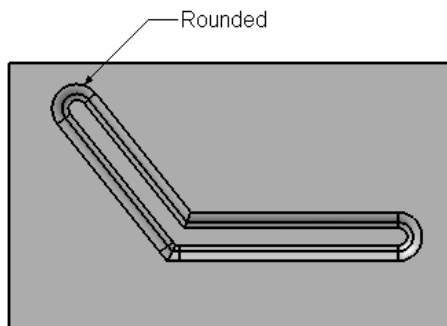


Figure 15-80 Curve stamp feature with the **Obround** check box selected

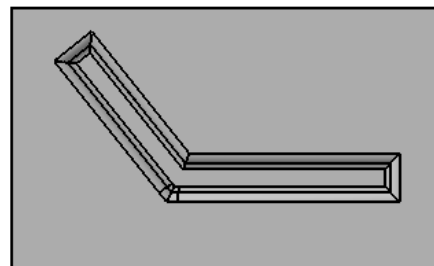


Figure 15-81 Curve stamp feature with the **Obround** check box cleared

#### Half pierce

On selecting this check box, the stamp feature will be created such that the height of the stamp feature will be half of the sheet metal.

### Parameters Area

Set the angular value in the **Angle A** spinner to provide a draft to the stamp wall. The angular value can range from 0.1 to 90-degree.

Next, set the height and length of the stamp feature in the **Height H** and **Length L** spinners, respectively. The **Radius R1** spinner is used to create a fillet on the surface where you have created the profile and the **Radius R2** spinner is used to create a fillet on the other surface, where the stamp ends. To activate the **Radius R1** and **Radius R2** spinners, select their corresponding check boxes.

### Profile

If the sketch is already drawn, select it from the geometry area; the preview of the curve stamp will be displayed. If the sketch is not drawn, choose the **Sketch** button on the right of the **Profile Selection** area, select the surface of the base feature, and create a sketch.

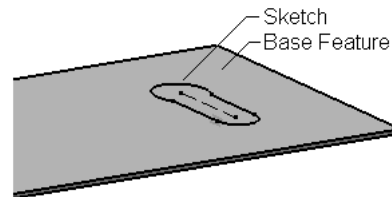
To see the preview of the stamp feature, choose the **Preview** button. Choose the **OK** button to create a curve stamp feature.

### Creating a Flanged Cut Out Stamp

**Menu:** Insert > Stamping > Flanged Cut Out  
**Toolbar:** Stamping > Flanged Cut Out

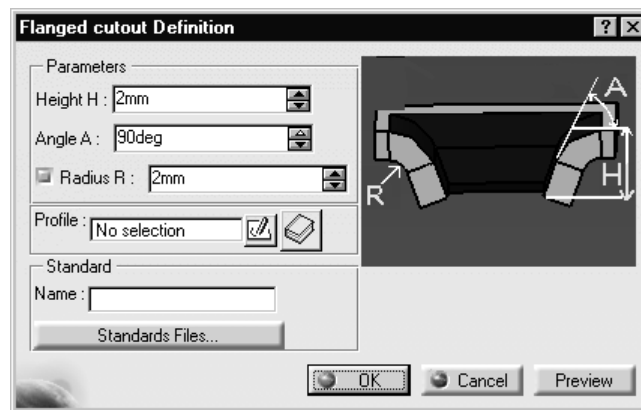


To create a flanged cut out stamp, create a closed profile on a wall, as shown in Figure 15-82. Choose the **Flange Cut Out** button from the **Stamping** toolbar; the **Flanged cutout Definition** dialog box will be displayed, as shown in Figure 15-83.



**Figure 15-82** Sketch of the flanged cutout feature

You will be prompted to select the profile. Select the closed profile from the geometry area, if it is not already selected. Set the height of the flange



**Figure 15-83** The *Flanged cutout Definition* dialog box

cutout feature in the **Height H** spinner. The height is specified normal to the face on the which the sketch is drawn. Next, set the angle in the **Angle A** spinner. If you select the **Radius R** check box, the edges of the profile get filleted. You can set the fillet radius in the **Radius R** spinner. Choose the **Preview** button; the preview of the flange cutout stamp will be displayed. Choose the **OK** button; the flanged cut out feature will be created, as shown in Figure 15-84.

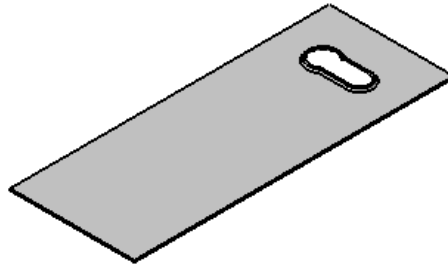


Figure 15-84 The flanged cutout feature

## Creating a Louver Stamp

**Menu:** Insert > Stamping > Louver  
**Toolbar:** Stamping > Louver



Louvers are created in a sheet metal part to provide openings for the purpose of ventilation. Generally, a rectangular pattern of louvers will be created on the top face of a cover. To create a louver stamp feature, you need to create a closed sketch on the base feature. Next, choose the **Louver** button from the **Stamping** toolbar; the **Louver Definition** dialog box will be displayed, as shown in Figure 15-85.

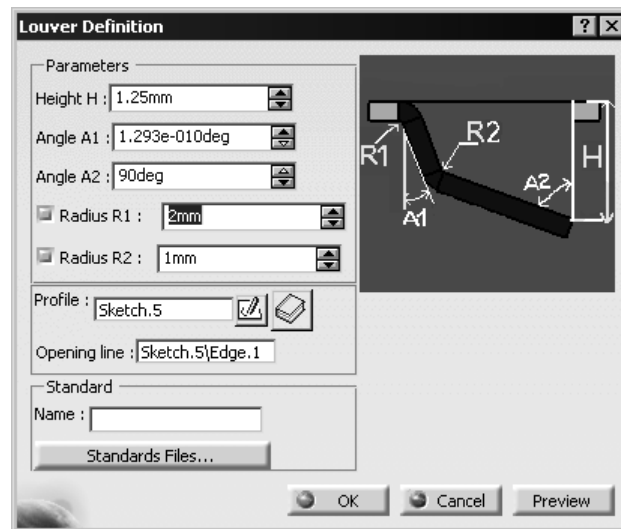
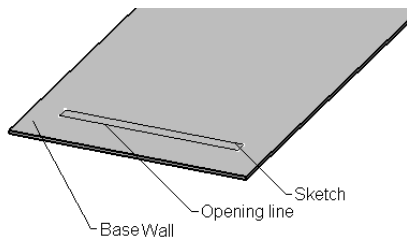
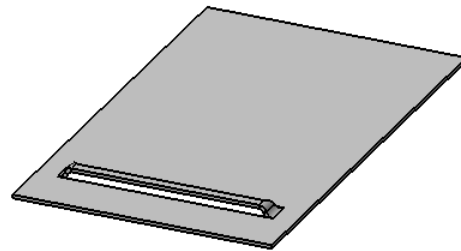


Figure 15-85 The Louver Definition dialog box

Select the sketch from the geometry area, if it is not already selected. Next, set the height of the louver in the **Height H** spinner. If you need to create the side walls of the louver at an angle to the face on which it is placed, set the angular value in the **Angle A1** spinner. By default, the angular value will be **0deg**. The angle between the top face of the louver and the plane where the opening is provided is specified in the **Angle A2** spinner. By default, its value will be **90deg**. Next, specify the opening edge of the louver from the sketch. The selected entity will be displayed in the **Opening line** selection area. Next, choose **OK** to create the louver feature. Figure 15-86 shows the sketch to create the louver feature and Figure 15-87 shows the resulting feature.



*Figure 15-86 Sketch for the louver feature*



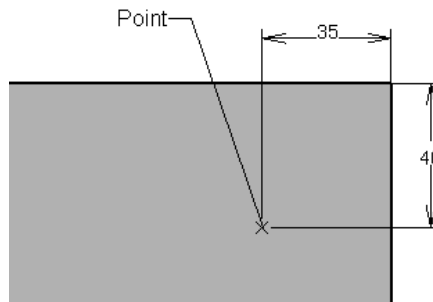
*Figure 15-87 Resulting louver feature*

## Creating a Bridge Feature

**Menu:** Insert > Stamping > Bridge  
**Toolbar:** Stamping > Bridge



The **Bridge** tool is used to create the features used for the purpose of holding and gripping. To create a bridge feature, you need to create a point on the base wall, as shown in the Figure 15-88. Select the point and the surface on which the Bridge feature has to be created. Next, choose the **Bridge** button from the **Stamping** toolbar; the **Bridge Definition** dialog box will be displayed, as shown in Figure 15-89.



*Figure 15-88 Dimensioned sketch point*

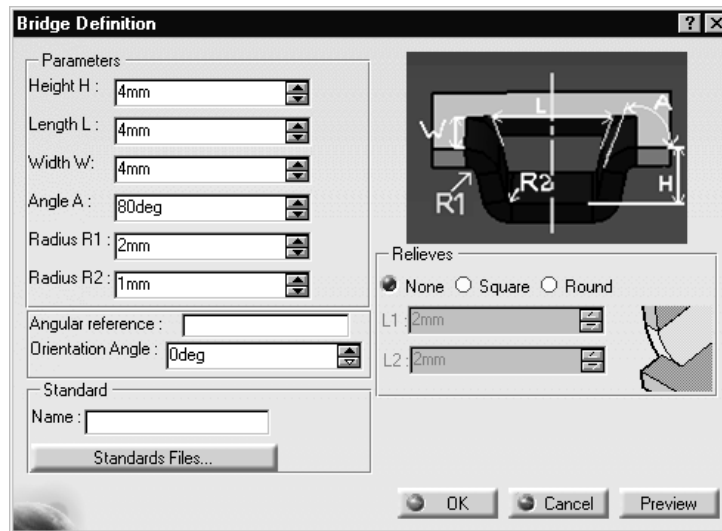


Figure 15-89 The *Bridge Definition* dialog box

### Parameters Area

The options in the **Parameters Area** are discussed next.

#### Height H and Length L

Set the height and length of the bridge in the **Height H** and **Length L** spinners, respectively.

#### Width W

Set the value for the width of the bridge in the **Width W** spinner.

#### Angle A

If you need the walls of the bridge at an angle, set the value in the **Angle A** spinner.

#### Radius R1 and Radius R2

Set the values of the outer bend radius and inner bend radius of the bridge in the **Radius R1** and **Radius R2** spinners, respectively.

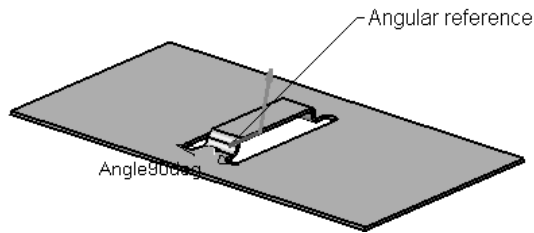
#### Angular reference

Select the angular reference about which the bridge rotates from the geometry area.

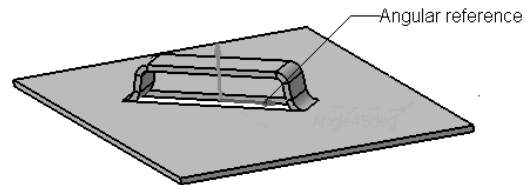
#### Orientation Angle

If you need the bridge feature to be created at an angle with respect to the horizontal plane, set the value in the **Orientation Angle** spinner. Figure 15-90 shows the bridge at an angle of 90-degree and Figure 15-91 shows the bridge at an angle of 45-degree.





**Figure 15-90** Bridge at an angle of 90-degree



**Figure 15-91** Bridge at an angle of 45-degree

### Relieves Area

There are three options in the **Relieves** area. They are discussed next.

If you select the **None** radio button, the bridge will not have any relief. If you need a square relief, select the **Square** radio button; the **L1** and **L2** spinners will be highlighted. Set the limits in the **L1** and **L2** spinners, respectively. You can create a round relief by selecting the **Round** radio button.

You can change the direction of the material by clicking the orange pointer in the geometry area. Choose the **OK** button to create the bridge stamp.

### Creating a Flanged Hole

<b>Menu:</b>	Insert > Stamping > Flanged Hole
<b>Toolbar:</b>	Stamping > Flanged Hole



To create a flanged hole, you need to create a point on a wall. Then select the point and the surface on which the hole has to be created. Next, choose the **Flanged Hole** button from the **Stamping** toolbar; the **Flanged Hole Definition** dialog box will be displayed, as shown in Figure 15-92.

You will notice that the **Major Diameter** option is selected by default in the **Parameters choice** drop-down list under the **Definition Type** area. Next, set the values for the height of the flanged hole, fillet radius, angle of the flanged hole, and the diameter of the flanged hole in the **Height H**, **Radius R**, **Angle A** and **Diameter D** spinners, respectively. The location of the flanged hole will be indicated by the pointer you place on the surface. If you do not want a protruded hole, select the **Without cone** radio button in the **Definition Type** area. Figure 15-93 shows the point placed on a wall. Figure 15-94 shows the flanged hole created with the **With cone** radio button selected.

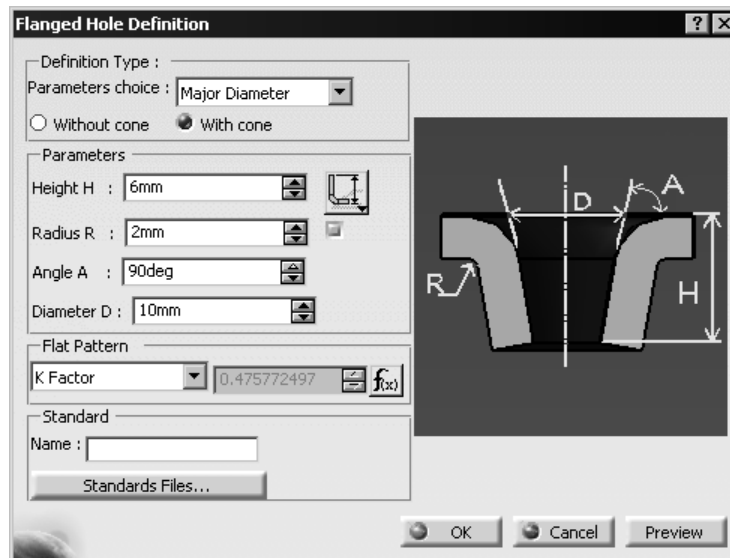


Figure 15-92 The Flanged Hole Definition dialog box

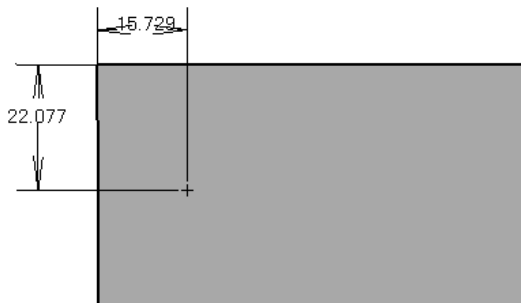


Figure 15-93 Dimensioned sketch point

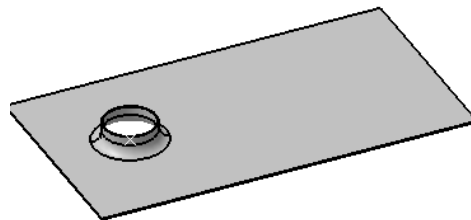


Figure 15-94 Flanged hole feature with the With cone radio button selected

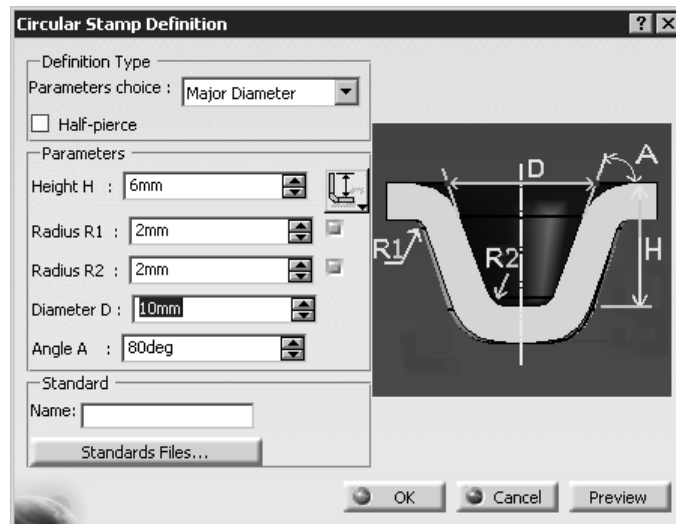
## Creating a Circular Stamp

**Menu:** Insert > Stamping > Circular Stamp  
**Toolbar:** Stamping > Circular Stamp



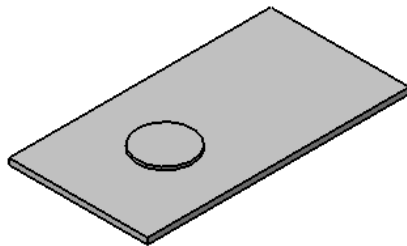
The **Circular Stamp** tool is used to create a circular stamp by specifying the parameters of the punch. To create a circular stamp, first you need to create a point on the wall. Next, select the point and the surface on which the hole has to be created and then choose the **Circular Stamp** button from the **Stamping** toolbar; the **Circular Stamp Definition** dialog box will be displayed, as shown in Figure 15-95.

In the **Circular Stamp Definition** dialog box, the **Major Diameter** option in the **Parameters choice** drop-down list under the **Definition Type** area is selected by default. The **Half-pierce**

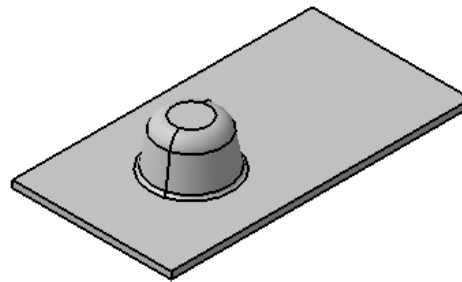


*Figure 15-95 The Circular Stamp Definition dialog box*

check box is selected to create a stamp such that its height is half the thickness of the sheet metal. If you select this check box, the **Radius R1**, **Radius R2**, and **Angle A** spinners will not be enabled in this dialog box. Figure 15-96 shows the feature created with the **Half-pierce** check box selected and Figure 15-97 shows the feature created with the **Half-pierce** check box cleared.



*Figure 15-96 Feature created with the Half-pierce check box selected*



*Figure 15-97 Feature created with the Half-pierce check box cleared*

## Creating a Stiffening Rib

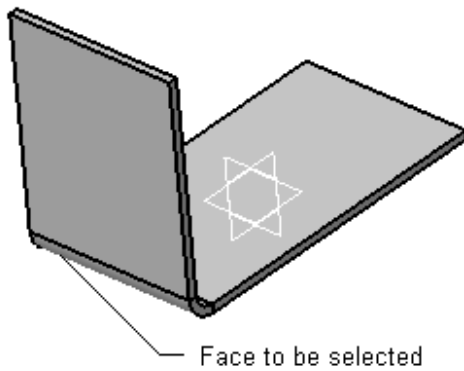
**Menu:** Insert > Stamping > Stiffening Rib  
**Toolbar:** Stamping > Stiffening Rib



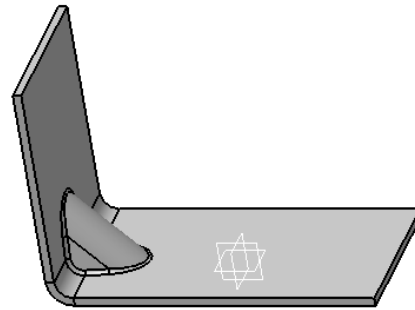
The **Stiffening Rib** tool is used to create a stiffness rib by specifying the geometrical parameters. To create a stiffness rib, choose the **Stiffening Rib** button from the **Stamping** toolbar; you will be prompted to select a position for the stiffness rib.

While selecting the position of the stiffness rib make sure to select the external face of the feature, as shown in Figure 15-98. On selecting the position of the stiffness rib, the preview of the stiffness rib will be displayed in the geometrical area and the **Stiffening Rib Definition** dialog box will also be displayed.

The options in the **Stiffening Rib Definition** dialog box are used to specify the length, radius, and angle for the stiffness rib in their respective spinners. Choose the **OK** button to create the rib feature, as shown in Figure 15-99.




**Figure 15-98** Face selected for creating the stiffness rib

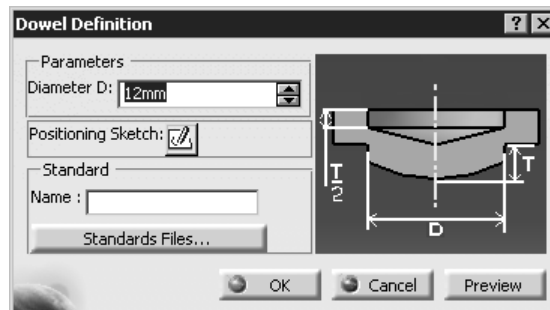


**Figure 15-99** Model after creating the stiffness rib feature

## Creating a Dowel

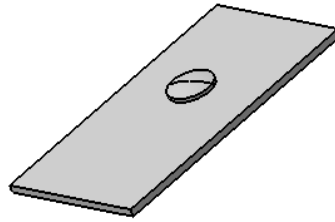
**Menu:** Insert > Stamping > Dowel  
**Toolbar:** Stamping > Dowel

 To create a dowel, choose the **Dowel** button from the **Stamping** toolbar; you will be prompted to select a face, a plane, or a point for the positioning the dowel. Select the face of the feature for positioning the dowel; the **Dowel Definition** dialog box will be displayed, as shown in Figure 15-100.



**Figure 15-100** The **Dowel Definition** dialog box

You can change the position of the dowel by using the **Positioning Sketch** button in the **Dowel Definition** dialog box. You can also increase or decrease the diameter of the dowel by using the **Diameter D** spinner available in the dialog box. Choose the **OK** button to create the dowel feature, as shown in Figure 15-101.



*Figure 15-101 Model after creating the dowel feature*

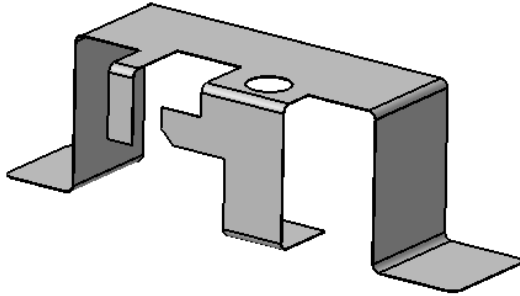
## TUTORIALS

### Tutorial 1

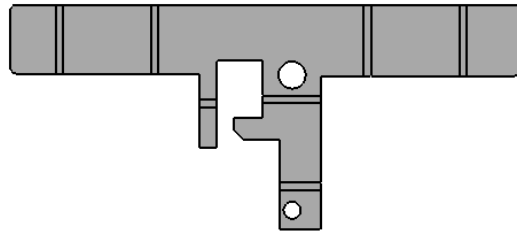
In this tutorial, you will create a sheet metal component of the Holder Clip, as shown in Figure 15-102. The flat pattern of the component is shown in Figure 15-103. Its dimensions are shown in Figures 15-104 and 15-105. The thickness of the sheet is 1 mm. After creating the sheet metal component, create its flat pattern. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

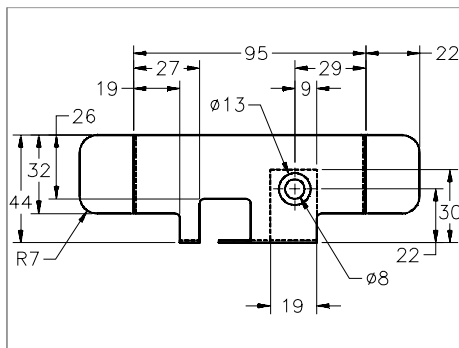
- Start a new metric sheet metal file and then draw the sketch of the top face of the sheet metal component.
- Set the parameters in the **Sheet Metal Parameters** dialog box and convert the sketch into a sheet metal face.
- Add the flange on the right and the left faces of the top feature.
- Add the flange on the front face of the feature.
- Create a cut feature on the front face of the new flange and then add another face and chamfer to it.
- Create the last flange and then create the hole. Finally, create the flat pattern.
- Save the sheet metal component.



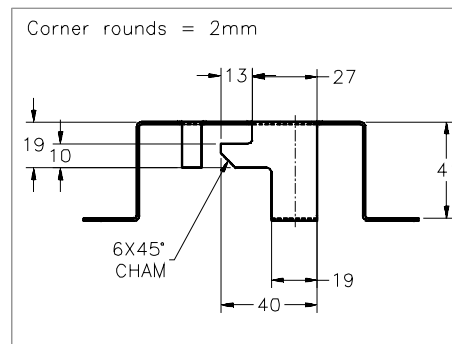
**Figure 15-102** Sheet metal model of the Holder Clip



**Figure 15-103** Flat pattern of the component



**Figure 15-104** Top view of the Holder Clip



**Figure 15-105** Front view of the Holder Clip

### Starting a New Part File

1. To start a new file in the **Generative Sheetmetal Design** workbench, choose **Start > Mechanical Design > Generative Sheetmetal Design** from the menu bar; the **New Part** dialog box is displayed. Choose the **OK** button in the **New Part** dialog box; a new file gets started in the **Generative Sheetmetal Design** workbench.

### Setting the Values in the Sheet Metal Parameters Dialog Box


1. Choose the **Sheet Metal Parameters** button from the **Walls** toolbar; the **Sheet Metal Parameters** dialog box is displayed.
2. Set **1mm** in the **Thickness** spinner and **2mm** in the **Default Bend Radius** spinner of the **Sheet Metal Parameters** dialog box. Choose the **OK** button.

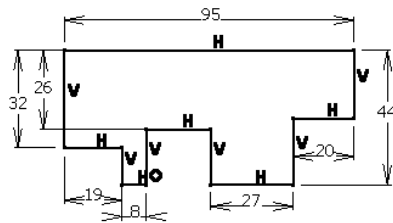


### Drawing the Sketch for the Top Face

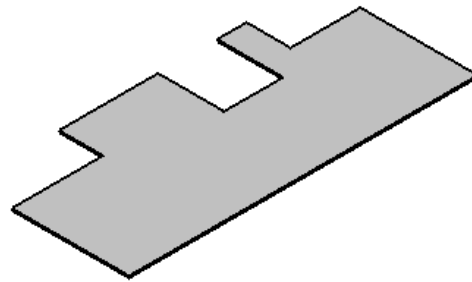
1. Invoke the **Sketcher** workbench by selecting the xy plane as the sketching plane and draw the sketch for the base wall, as shown in Figure 15-106.
2. Exit the **Sketcher** workbench.

### Converting the Sketch into a Base Wall

1. Choose the **Wall** button from the **SmdNewDesign** toolbar; the **Wall Definition** dialog box is displayed. 
2. Select the sketch from the geometry area, if it is not already displayed in the **Profile** selection area. The preview of the base feature is displayed in the geometry area after selecting the sketch.
3. Choose the **OK** button to get the final feature of the base wall, as shown in Figure 15-107.





**Figure 15-106** Sketch for the base wall of the Holder Clip



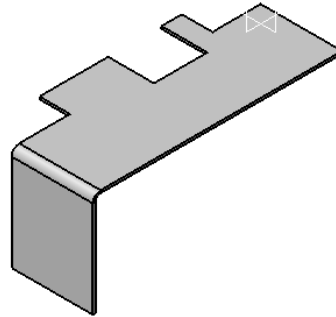
**Figure 15-107** Base wall of the Holder Clip

### Creating the Flanges

1. Choose the **Flange** button from the **SmdNewDesign** toolbar; the **Flange Definition** dialog box is displayed. 
2. Select the edge on which you need to create the flange, as shown in Figure 15-108; it is displayed in the **Spine** selection area.
3. Next, select the **Basic** option in the **Flange type** drop-down list, if it is not already selected.
4. Set the value **41mm** in the **Length** spinner. Choose the down arrow on the **Length type** button; a flyout is displayed. Choose the third button in the flyout. 
5. Set **90deg** in the **Angle** spinner and **2mm** in the **Radius** spinner. Choose the **Reverse Direction** button, if required.
6. Accept the remaining default options and choose the **OK** button to create the flange. Figure 15-109 shows the resultant flange.

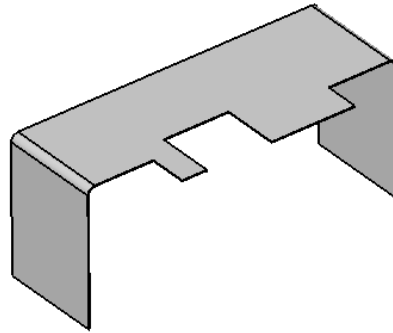


**Figure 15-108** Selecting the edge for creating the flange




**Figure 15-109** Sheet metal component after creating the flange

7. Create another flange of 41 mm length on the other edge, as shown in Figure 15-110. For creating the flange, follow the same procedure as explained in the previous steps. Choose the **Reverse Direction** button, if required.

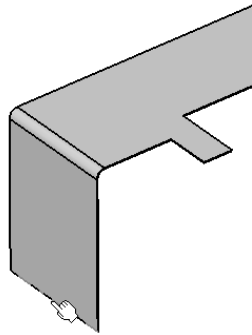


**Figure 15-110** Sheet metal component after creating the flanges

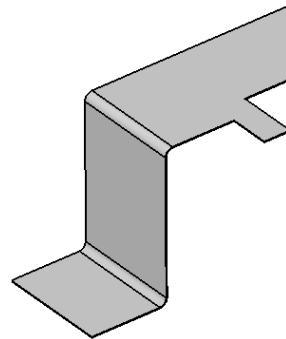
### Creating Other Flanges

1. Choose the **Flange** button from the **SmdNewDesign** toolbar; the **Flange Definition** dialog box is displayed. 
2. Set the value **22mm** in the **Length** spinner, **90deg** in the **Angle** spinner, and **2mm** in the **Radius** spinner.
3. Select the edge of the base wall, as shown in Figure 15-111.
4. Choose the **Reverse Direction** button, if required and accept the remaining default options. Next, choose the **OK** button to create the flange. The sheet metal component after creating the flange is shown in Figure 15-112.





**Figure 15-111** Selecting the edge for creating the flange

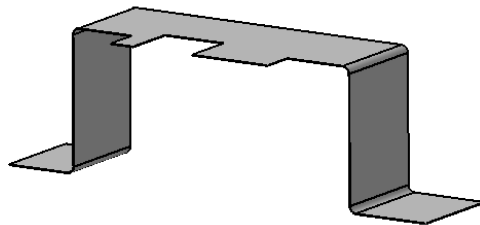


**Figure 15-112** Sheet metal component after creating the flange

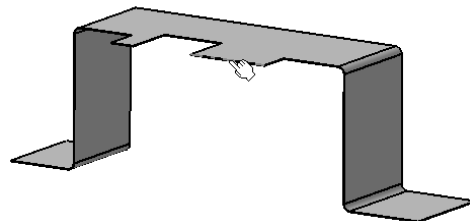
5. Similarly, create another flange of 22mm length on the edge, as shown in Figure 15-113.

### Creating a Wall on Edge

1. Choose the **Wall On Edge** button from the **SmdNewDesign** toolbar; the **Wall On Edge Definition** dialog box is displayed. Select the **Sketch Based** option from the **Type** drop-down list in the dialog box.
2. Select the lower edge of the base wall, as shown in Figure 15-114.



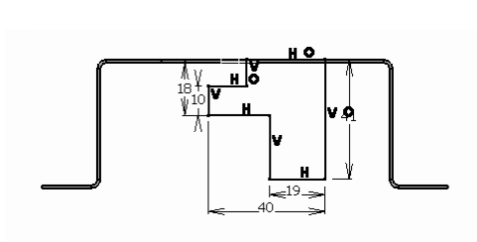
**Figure 15-113** Sheet metal component after creating the flange



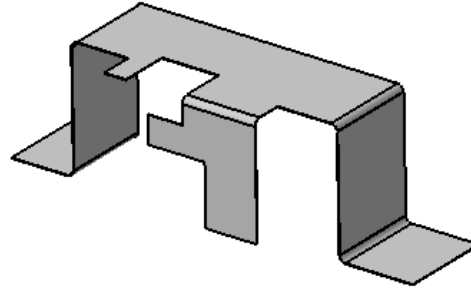
**Figure 15-114** Selecting the lower edge to create the wall on edge

3. Choose the **Sketch** button on the right side of the **Profile** selection area; you are prompted to select the sketching plane.
4. Select the surface representing the thickness of the base wall. Draw the sketch of the wall on edge, as shown in Figure 15-115.
5. Exit the **Sketcher** workbench.

- Accept the remaining default options and choose the **OK** button to create the wall on the edge. Figure 15-116 shows the resultant wall on the edge.



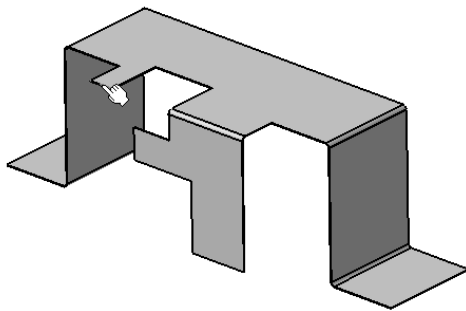
*Figure 15-115 Sketch of the wall on edge feature*



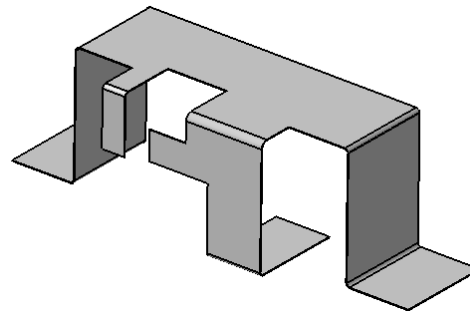
*Figure 15-116 Sheet metal component after creating the wall on the edge*

### Creating other Flanges

- Choose the **Flange** button from the toolbar; the **Flange Definition** dialog box is displayed.
- Set the value **19mm** in the **Length** spinner, **90deg** in the **Angle** spinner, and **2mm** in the **Radius** spinner. Select the edge of the base wall, as shown in Figure 15-117.
- Accept the remaining default options and choose the **OK** button to create the flange. The sheet metal component after creating the flange is shown in Figure 15-118.
- Similarly, create another flange of 30mm length on the edge, refer to Figure 15-118.



*Figure 15-117 Selecting an edge*



*Figure 15-118 The flange created*

### Creating the Remaining Feature

- Create the holes using the **Cut Out** tool from the **pCuttingStamping** toolbar.
- Create the fillets by using the **Corner** tool.

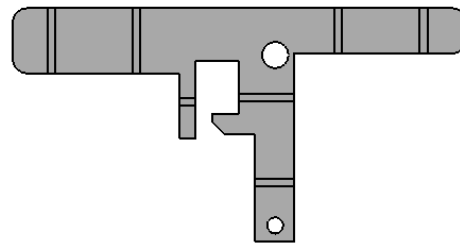
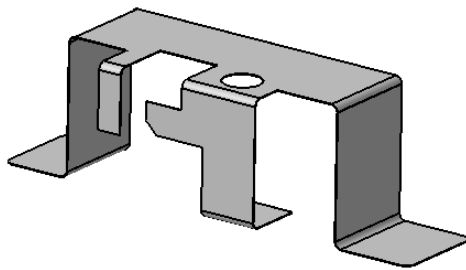


3. Invoke the **Chamfer** tool and create the chamfers on the walls. Figure 15-119 shows the sheet metal component after creating the holes, fillets, and chamfer. For the dimension of holes, fillets, and chamfer, refer to Figures 15-104 and 15-105. This completes the sheet metal component of the Holder Clip. The final sheet metal component of the Holder Clip is shown in Figure 15-119.



### Creating the Flat Pattern

1. Choose the **Fold/Unfold** button from the **pViews** toolbar; the flattened view of the sheet metal part is created, as shown in Figure 15-120.



**Figure 15-119** Final model of the Holder Clip

**Figure 15-120** Flat Pattern of the Holder Clip

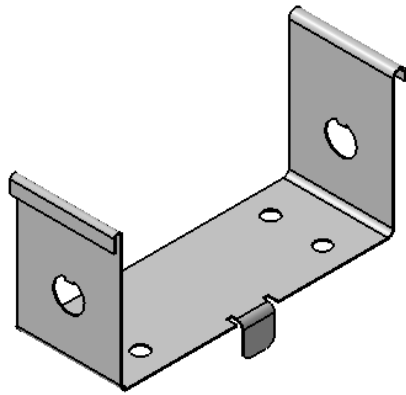
2. Save the sheet metal component with the name *c15tut1*.

## Tutorial 2

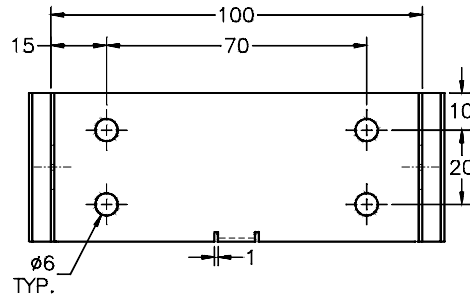
In this tutorial, you will create the sheet metal component shown in Figure 15-121. Its dimensions are given in Figures 15-122 through 15-124. The flat pattern of the component is shown in Figure 15-125. The thickness of the sheet is 1 mm. The length of the hem on the two faces is 5 mm and its radius is 3 mm. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

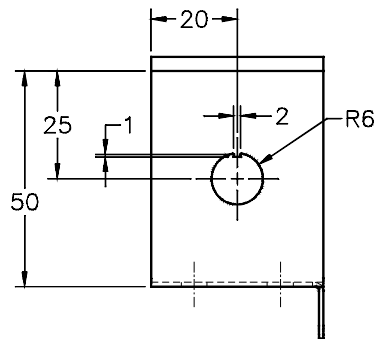
- a. Start a new file in **Generative Sheetmetal Design** workbench.
- b. Draw the sketch and convert it into a base wall by using the **Wall** tool, refer to Figure 15-126.
- c. Create one hole and then pattern it to create the remaining three instances, refer to Figure 15-127.
- d. Create the flange on the left and right edges of the base wall, refer to Figure 15-129.
- e. Create hems on the two flanges and then create the two keyways by using the cutout tool, refer to Figure 15-130 and 15-131.
- f. Create the wall on the edge on the front face of the base, refer to Figure 15-132.
- g. Create the flat pattern, refer to Figure 15-133.
- h. Save the model.



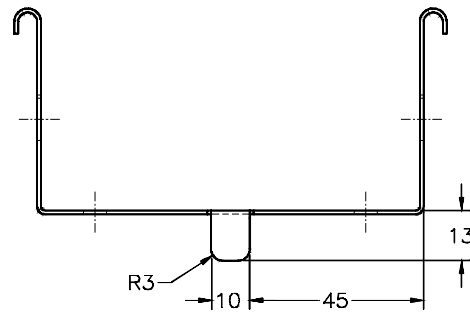
**Figure 15-121** Sheet metal component for Tutorial 2



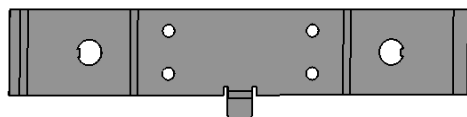
**Figure 15-122** Top view of the model



**Figure 15-123** Left-side view of the model



**Figure 15-124** Front view of the model



**Figure 15-125** Flat pattern of the Sheet metal component

### Starting a New File and Setting the Sheetmetal Part Parameters

1. Choose **New** from the **File** options in the menu bar and start a new file in the **Generative Sheetmetal Design** workbench.
2. Choose the **Sheet Metal Parameters** button from the **SmdNewDesign** toolbar; the **Sheet Metal Parameters** dialog box is displayed.




3. Set **1mm** in the **Thickness** spinner and **2mm** in the **Default Bend Radius** spinner.
4. Choose the **Bend Extremities** tab and select the **Square relief** option from the drop-down list. Set **1mm** and **2mm** in the **L1** and **L2** spinners, respectively and then choose the **OK** button from the **Sheet Metal Parameters** dialog box to exit it.

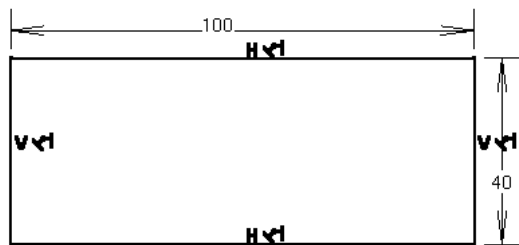
### Drawing the Sketch for the Base Feature

1. Invoke the **Sketcher** workbench by selecting the xy plane as the sketching plane to draw the sketch, as shown in Figure 15-126.
2. Exit the **Sketcher** workbench.

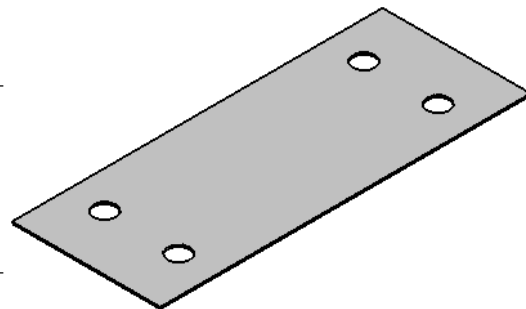
### Converting the Sketch into a Base Wall

As the parameters are set in the **Sheet Metal Parameters** dialog box, you need to convert the sketch into a base wall.

1. Choose the **Wall** button from the **SmdNewDesign** toolbar; the **Wall Definition** dialog box is displayed. Select the sketch from the geometry area, if it is not already selected; it is displayed in the **Profile** selection area. Choose **OK** to create the base wall. 
2. Choose the **Hole** button from the **Holes** toolbar and create a hole at the lower left corner of the base wall. For dimensions of the hole, refer to Figure 15-122.
3. Create the rectangular pattern of the hole. Change the current view to isometric view. The base of the sheet metal component, after creating the hole pattern, is shown in Figure 15-127.



*Figure 15-126 Sketch of the base wall*

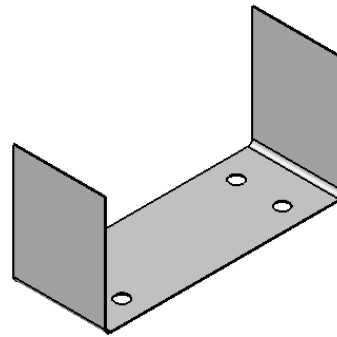
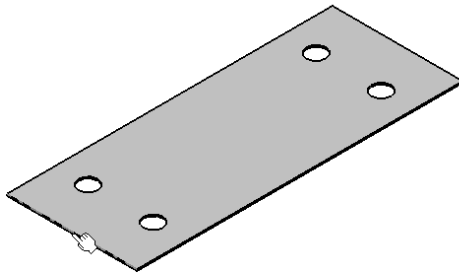


*Figure 15-127 Base wall with the hole feature*

### Creating Two Flanges


1. Choose the **Flange** button from the **SmdNewDesign** toolbar; the **Flange Definition** dialog box is displayed.

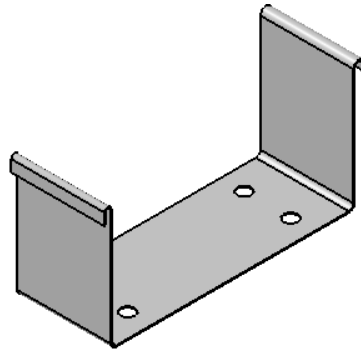
2. Select the edge of the base wall, as shown in Figure 15-128. You will notice that the preview of the flange is displayed in the downward direction.
3. Set the value **50mm** in the **Length** spinner. Choose the down arrow on the **Length type** button; a flyout is displayed. Choose the third button in the flyout.
4. Set **90deg** in the **Angle** spinner and **2mm** in the **Radius** spinner.
5. Choose the **Reverse Direction** button, if required and choose **OK**; the flange is created.
6. Similarly, create the flange on the other side of the base feature. The sheet metal component, after creating the flanges, is shown in Figure 15-129.



*Figure 15-128 Selecting the edge for flange      Figure 15-129 Model after creating the two flanges*

### Creating the Hems


1. Choose the **Hem** button from the **Swept Walls** toolbar; the **Hem Definition** dialog box is displayed. You are prompted to select an edge to create the hem. 
2. Select the inner edge of the right flange; it is displayed in the **Spine** selection area.
3. Select the **Basic** option from the **Flange type** drop-down list.
4. Set **5mm** value in the **Length** spinner and **1mm** in the **Radius** spinner.
5. Accept the remaining default parameters in the **Hem** dialog box and choose the **OK** button to create the hem.
6. Similarly, create the hem on the other flange. The sheet metal component, after creating the two hems is shown in Figure 15-130.

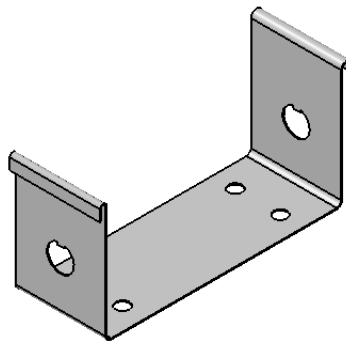


*Figure 15-130 Model after creating two hems*

### Creating a Cut Feature


Next, a cut feature with a keyway needs to be created. This feature will be created by using the **Cut Out** tool.

1. Define a new sketching plane on the outer face of the right flange. Draw the sketch by referring to the dimensions given in Figure 15-123.
2. Exit the **Sketcher** workbench.
3. Choose the **Cut Out** button from the **pCuttingStamping** toolbar; the **Cutout Definition** dialog box is displayed. 
4. Select the **Up to Last** option in the **Type** drop-down list from the **End Limit** area.
5. Accept the remaining default parameters in the **Cutout Definition** dialog box and choose **OK** to create the cutout.
6. The model, after creating the two keyways, is shown in Figure 15-131.




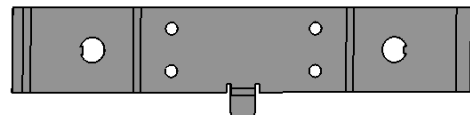
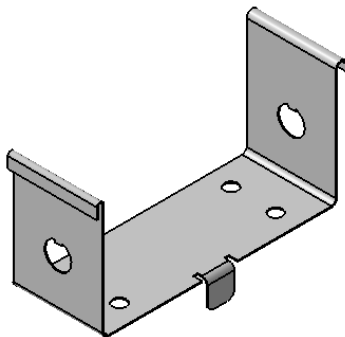
*Figure 15-131 Sheet metal after creating the two keyways*

### Creating the Side Wall with Corner and Filletig the Edges

1. Choose the **Wall on Edge** button from the **SmdNewDesign** toolbar; the **Wall On Edge Definition** dialog box is displayed. 
2. Choose the **Height and Inclination** tab in the **Wall On Edge Definition** dialog box, if it is not already chosen and set **13mm** in the **Height** spinner and **90deg** in the **Angle** spinner. Select the **With Bend** check box.
3. Choose the **Extremities** tab from this dialog box. Set **-44mm** in both the **Left offset** and **Right offset** spinners.
4. Select the front edge of the base feature to create a wall on edge.
5. Accept the remaining default options and choose **OK**; the side wall feature is created. Create the corner feature of dimensions given in Figure 15-124.
6. Create the fillets by using the **Corner** tool. For dimensions of the fillets, refer to Figure 15-124. The final sheet metal component is shown in Figure 15-132.

### Creating the Flat Pattern

1. Choose the **Unfold/Fold** button from the **Views** toolbar, the flat pattern of the sheet metal component is created, as shown in Figure 15-133. 
2. Save the sheet metal component with the name *c15tut2*.



**Figure 15-132** Final sheet metal component    **Figure 15-133** Flat pattern of the sheet metal component

## Tutorial 3

In this tutorial, you will create the sheet metal component of the Slide Cover, as shown in Figure 15-134. Its dimensions are shown in Figure 15-135. **(Expected time: 30 min)**



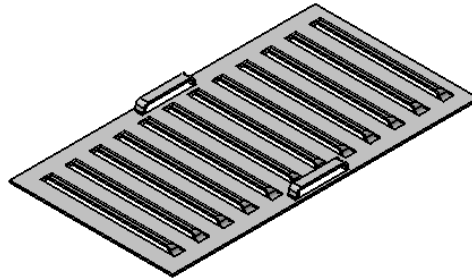


Figure 15-134 Model for Tutorial 3

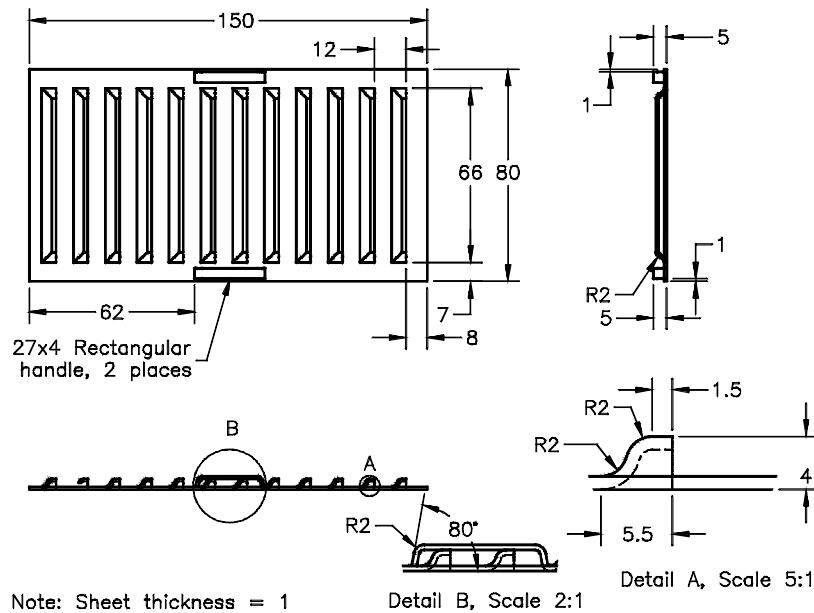


Figure 15-135 Dimensions for Tutorial 3

The following steps are required to complete this tutorial:

- Start a new file in **Generative Sheetmetal Design** workbench.
- Draw the sketch and convert it into a base wall by using the **Wall** tool, refer to Figure 15-136 and Figure 15-137.
- Create the louver feature and pattern it, refer to Figures 15-138 through 15-140.
- Create the handles, refer to Figures 15-141 and 15-142.
- Save the model

### Starting a New Part File

1. Start a new file in the **Generative Sheetmetal Design** workbench.
2. Set the sheet metal parameters in the **Sheet Metal Parameters** dialog box.

### Drawing the Sketch of the Base Wall

1. Invoke the **Sketcher** workbench by selecting the xy plane as the sketching plane and draw the sketch, as shown in Figure 15-136.
2. Exit the **Sketcher** workbench.

### Converting the Sketch into a Base Wall

1. Choose the **Wall** button from the **SmdNewDesign** toolbar; the **Wall Definition** dialog box is displayed.
2. Select the sketch from the geometry area, if it was not selected earlier.
3. Choose the **OK** button to create the base wall, as shown in Figure 15-137.

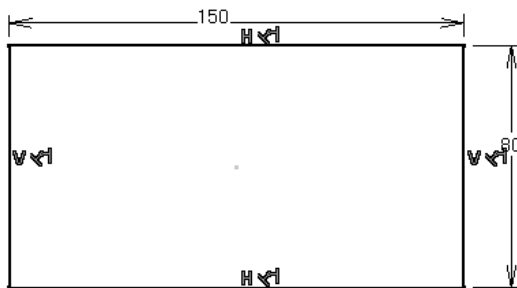


Figure 15-136 Sketch of the base wall

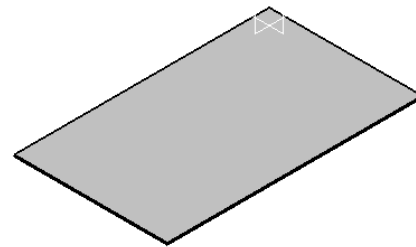


Figure 15-137 The resultant base wall

### Drawing the Sketch of the Ventilation Slots

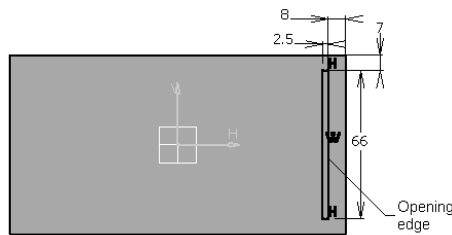
1. Choose the **Sketch** button and then select the top face of the base wall as the sketching plane; the **Sketcher** workbench is invoked. Draw the sketch of the Louver, as shown in Figure 15-138.
2. Exit the **Sketcher** workbench.

### Creating Ventilation Slots

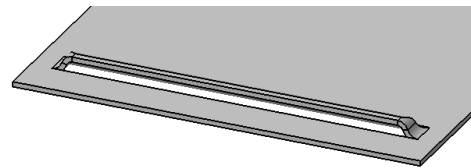
1. Choose the **Louver** button from the **Stamping** toolbar; the **Louver Definition** dialog box is displayed. Set **2mm** in the **Height** spinner.
2. Set **0deg** in the **Angle A1** spinner and **90deg** in the **Angle A2** spinner. Next, set **2mm** and **1mm** in the **Radius R1** and **Radius R2** spinners, respectively.



3. Select the sketch from the geometry area; if it is not displayed in the **Profile** selection area.
4. Select the line nearest to the edge as the open end of the louver, refer to Figure 15-138.
5. Reverse the direction of the louver, if required.
6. Choose the **OK** button; the Louver stamp feature is created, as shown in Figure 15-139.



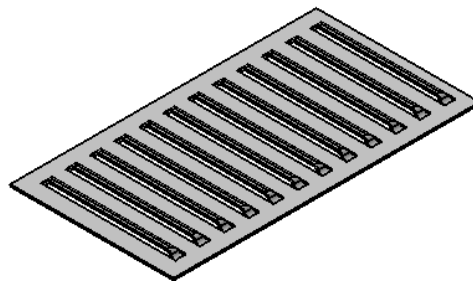
*Figure 15-138 Sketch of the Louver stamp*



*Figure 15-139 The louver stamp feature created*

### Creating the Pattern of the Louver Feature

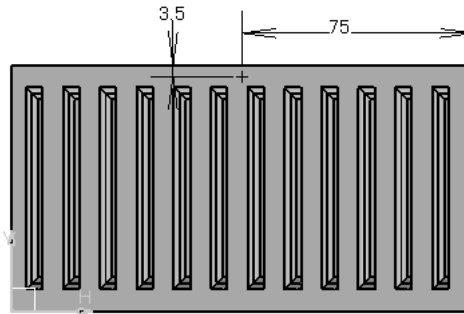
1. Select the louver feature from the specification tree and choose the **Rectangular Pattern** button; the **Rectangular Pattern Definition** dialog box is displayed.
2. Set **12** in the **Instances** spinner and **12mm** in the **Spacing** spinner.
3. Select the base wall as the reference element.
4. Choose the **Reverse** button, if required.
5. Choose the **Second Direction** tab and make sure the number of instances specified in the **Instances** spinner is 1.
6. Choose the **OK** button; the pattern of the louver feature is created, as shown in Figure 15-140.



*Figure 15-140 Pattern of the louver feature*

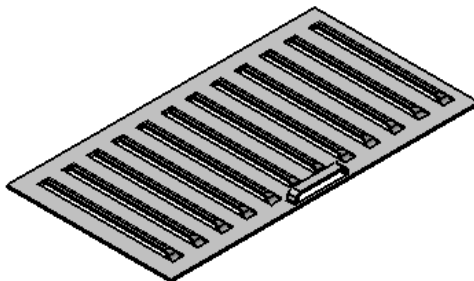
### Creating the Handles

1. Create a point on the top face of the base wall, as shown in Figure 15-141. Select the point and the top face of the sheet metal part. Then, choose the **Bridge** button from the **Stamping** toolbar; the **Bridge Definition** dialog box is displayed.

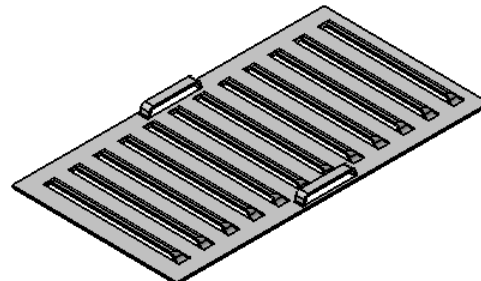


*Figure 15-141 Creating the point for the Bridge feature*

2. Set **4mm** in the **Height** spinner and **27mm** in the **Length** spinner.
3. Next, set **4mm** in the **Width** spinner and **80deg** in the **Angle** spinner.
4. Set **0.5mm** in the **Radius R1** spinner and **1mm** in the **Radius R2** spinner.
5. Change the direction, if required, and choose **OK**; the Bridge feature is created, as shown in Figure 15-142.
6. Similarly, create another handle on the other side of the feature. The final model for Tutorial 3 is shown in Figure 15-143.
7. Save the sheet metal component with the name *c15tut3*.

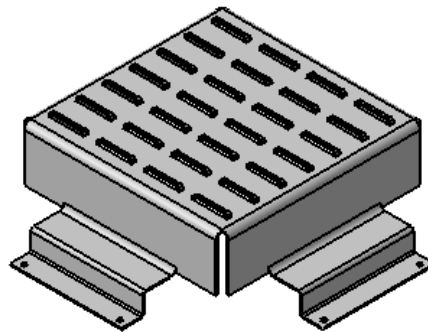


*Figure 15-142 The Bridge feature created*

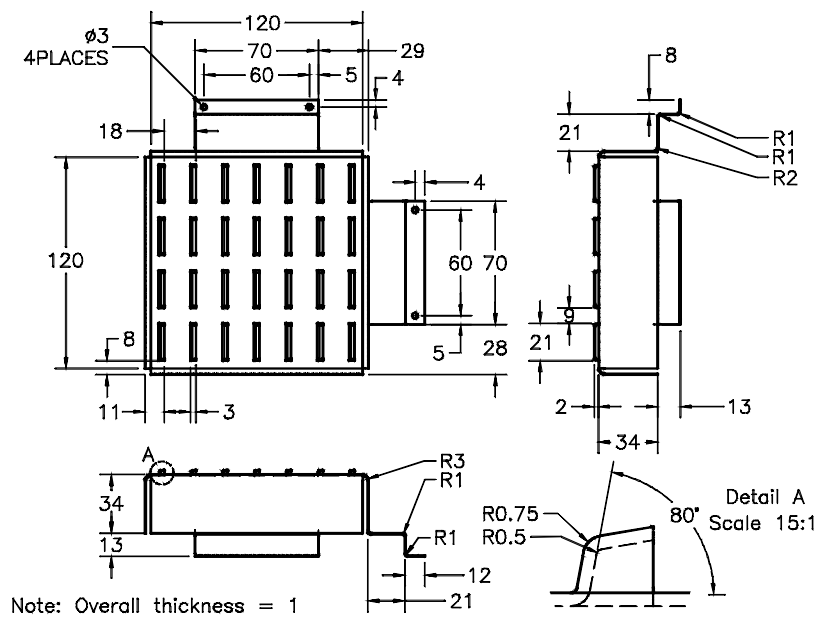


*Figure 15-143 The final model for Tutorial 3*

In this tutorial, you will create the sheet metal component, as shown in Figure 15-144. Its dimensions are shown in Figure 15-145. The thickness of the sheet metal component is 1 mm and bend radius is 2 mm. **(Expected time: 45 min)**



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**Figure 15-145** Views and dimensions of the sheet metal component for Tutorial 4

The following steps are required to complete this tutorial:

- Start a new file in **Generative Sheetmetal Design** workbench.
- Draw the sketch and convert it into a base wall by using the **Wall** tool, refer to Figure 15-147.
- Create the flange on the four edges of the base wall, refer to Figure 15-150.
- Create the wall on the flange, refer to Figure 15-152.
- Create the cut out feature.
- Create the louver on the top face and pattern it, refer to Figure 15-157. Save the model.

### Starting a New File and Setting the Sheet Metal Parameters

- Start a new file in the sheetmetal environment, invoke the **Sheet Metal Parameters** dialog box, and then set the parameters of the sheet metal component in this dialog box.

### Drawing the Sketch of the Base Wall

- Invoke the **Sketcher** workbench by selecting the xy plane and draw the sketch, as shown in Figure 15-146.
- Exit the **Sketcher** workbench.

### Converting the Sketch into the Base Plate

- Choose the **Wall** button from the **SmdNewDesign** toolbar; the **Wall Definition** dialog box is displayed.
- Select the sketch from the geometry area, if it is not already selected.
- Choose the **OK** button; the base wall is created, as shown in Figure 15-147.

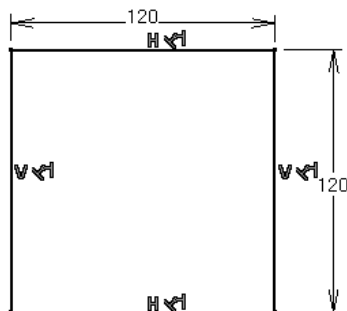


Figure 15-146 Sketch for the base wall feature

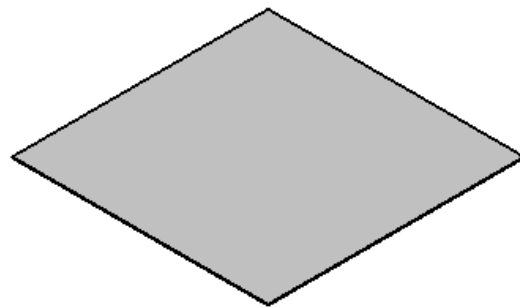
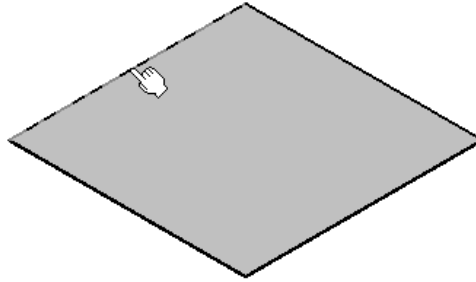


Figure 15-147 The base wall feature


### Creating the Flange on the Edge of the Base Plate

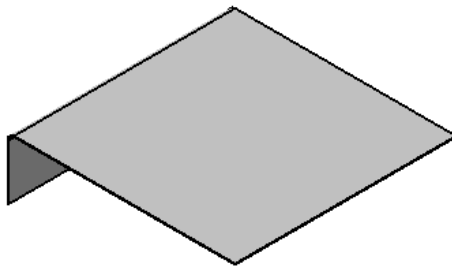
- Select the edge on which the flange has to be created, as shown in Figure 15-148.
- Choose the **Flange** button from the **SmdNewDesign** toolbar; the **Flange Definition** dialog box is displayed.



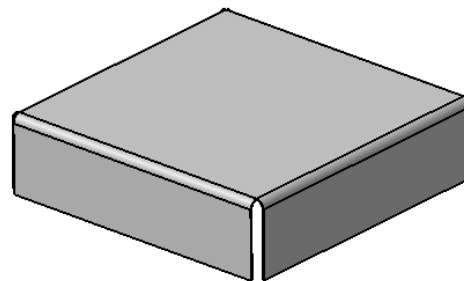


*Figure 15-148 Edge selected for creating the flange*

3. Choose the **Basic** option from the **Flange type** drop-down list, if it is not already selected.
4. Set the value **34mm** in the **Length** spinner. Choose the down arrow on the **Length type** button; a flyout is displayed. Choose the third button in the flyout. 
5. Set **90deg** in the **Angle** spinner and **2mm** in the **Radius** spinner.
6. Choose the **Reverse Direction** button, if required and choose the **OK** button. Figure 15-149 shows the resultant flange.
7. Similarly, create three more flanges on the other edges of the base wall. Figure 15-150 shows the sheet metal component after creating other flanges.



*Figure 15-149 Resulting flange feature*

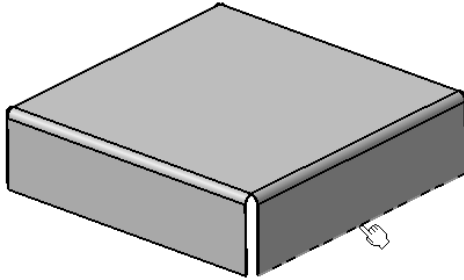


*Figure 15-150 Component with all flanges*

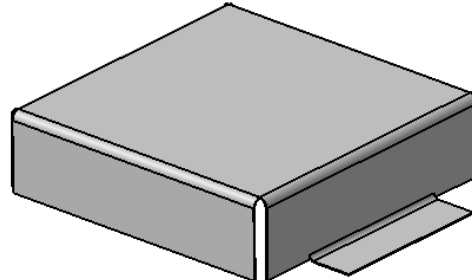
### Creating the Wall on the Flange

1. Choose the **Wall On Edge** button from the **SmdNewDesign** toolbar; the **Wall On Edge Definition** dialog box is displayed. Select the **Automatic** option from the **Type** drop-down list.
2. Select the edge of the flange, as shown in Figure 15-151.
3. Set **21mm** in the **Height** spinner and **90deg** in the **Angle** spinner. Select the **With Bend** check box.

4. Choose the **Extremities** tab. Set **-25mm** value both in the **Left offset** and **Right offset** spinners.
5. Choose the **OK** button; the side wall feature is created, as shown in Figure 15-152.




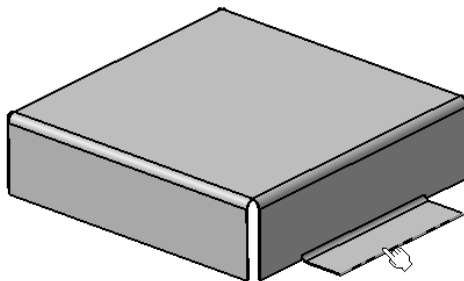
*Figure 15-151* Selecting the edge for creating the wall on the flange



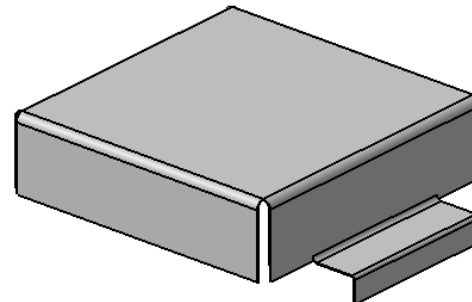
*Figure 15-152* The side wall feature created

### Creating the Flange on the Wall

1. Select the edge of the wall created in the previous step, as shown in Figure 15-153.
2. Choose the **Flange** button; the **Flange Definition** dialog box is displayed. Select the **Basic** option in the **Flange type** drop-down list. 
3. Set **13mm** in the **Length** spinner, **90deg** in the **Angle** spinner, and **1mm** in the **Radius** spinner.
4. Choose the **Reverse Direction** button, if required. Choose the **OK** button; the flange is created, as shown in Figure 15-154.



*Figure 15-153* Selecting the edge for creating the flange



*Figure 15-154* The flange created

5. Next, create another flange on the wall created in the previous step. Make sure that it is in the horizontal direction and also away from the vertical wall.
6. Similarly, create the wall on edge and flange feature on the other side of the sheet metal component, as shown in Figure 15-155. Its parameters are the same as in the previous case.

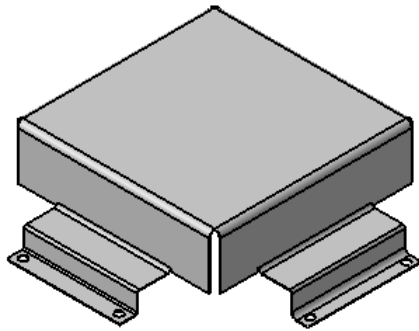


### Creating the Cut Outs

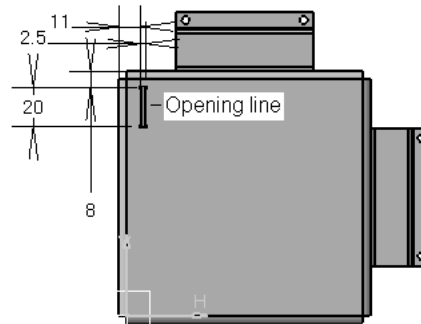
1. Create all cut outs by using the **Cut Out** tool from **pCuttingStamping** toolbar, refer to Figure 15-155. For dimensions of the cut outs, refer to Figure 15-145.

### Creating the Louver


1. Select the top face of the base wall as a sketching plane and draw the sketch of the louver, as shown in Figure 15-156.




**Figure 15-155** The component after creating the Cut out features

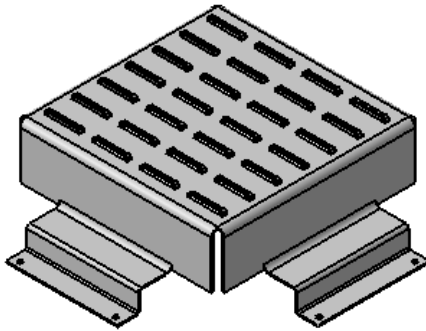


**Figure 15-156** Sketch of the Louver

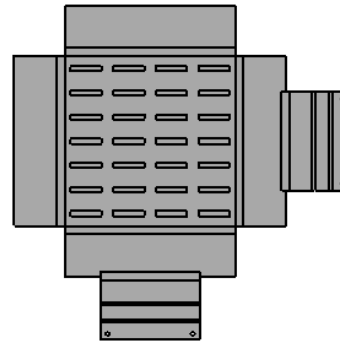
2. Choose the **Louver** button from the **Stamping** toolbar. Set **2mm** in the **Height** spinner and **10deg** in the **Angle A1** spinner, and **80deg** in the **Angle A2** spinner. 
3. Set **0.25mm** in both the **Radius R1** and **Radius R2** spinners. Select the sketch from the geometry area.
4. Select the line that is near to the cut out feature as the open end of the louver feature.
5. Change the direction, if required. Choose the **OK** button; the louver feature is created.
6. Create the pattern of the louver feature, refer to Figure 15-157.

### Creating the Flat Pattern

1. Choose the **Fold/Unfold** button from the **Views** toolbar; the sheet metal component is unfolded, as shown in Figure 15-158. To fold it again, choose the same button. 
2. Save the sheet metal component with the name *c15tut4*.



**Figure 15-157** Pattern of the louver feature created



**Figure 15-158** Flat pattern of the component

## SELF-EVALUATION TEST

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

1. The sheet metal files are saved as the *.CATPart* file. (T/F)
2. When you invoke a new sheet metal file, you are in the **Sketcher** environment by default. (T/F)
3. The **Linear** option provides a round relief between the supporting walls of the sheetmetal. (T/F)
4. The **Extrusion** tool is applicable only for the open profiles. (T/F)
5. You can unfold a sheet metal component by choosing the \_\_\_\_\_ button.
6. Tear Drop is a type of Hem. (T/F)
7. The **Closed relief** option provides relief to the intersection between the bends of two supporting walls. (T/F)
8. \_\_\_\_\_ are created in a sheet metal part for the purpose of ventilation.
9. You can also create the stamp feature with open walls. (T/F)
10. You can fillet all corners of the feature by using the \_\_\_\_\_ tool.

## REVIEW QUESTIONS

Answer the following questions:

1. The \_\_\_\_\_ tool is used to create a new bent face between the two existing walls.
2. The \_\_\_\_\_ walls are created by sweeping a profile along the selected edge.
3. Which of the following options is not available in the **Lines** drop-down list of the **Bend From Flat Definition** dialog box.
  - (a) **Bent Tangent Line**
  - (b) **Inner Mold Line**
  - (c) **Outer Mold Line**
  - (d) **Middle Mold Line**
4. You can change the values of \_\_\_\_\_ parameters in the **Sheet Metal Parameters** dialog box at any time. (T/F)
5. You can set the material for the sheet metal component from the **Sheet Metal Parameters** dialog box. (T/F)
6. The \_\_\_\_\_ feature is created for holding and gripping a sheet metal component.
7. Which of the following options provides relief between the extreme edges of two supporting walls?
  - (a) **Tangent**
  - (b) **Closed**
  - (c) **Maximum**
  - (d) **Linear**
8. The \_\_\_\_\_ option provides a round relief between the supporting walls of the sheet metal.
9. While creating the Bead feature, if the sketch has multiple continuous entities, they should be tangentially connected. (T/F)
10. Flange is the bend section of the sheet metal. (T/F)

## EXERCISE

### Exercise 1

Create the sheet metal component shown in Figure 15-159. The dimensions of the model are shown in Figures 15-160 and 15-161. Assume the missing dimensions.

(Expected time: 30 min)

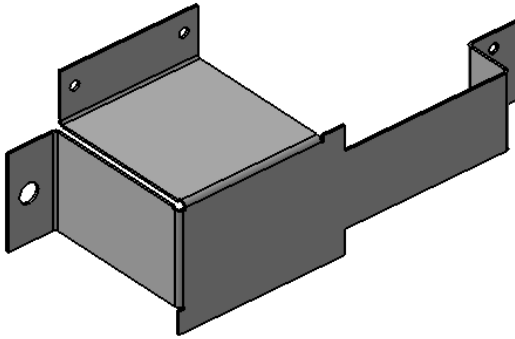


Figure 15-159 Sheet metal part for Exercise 1

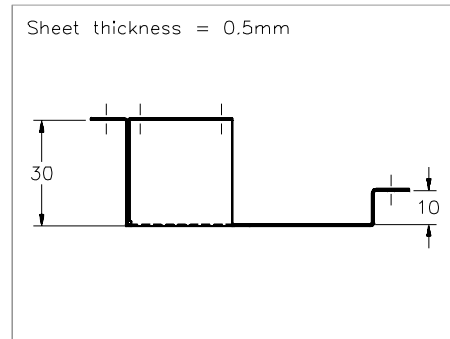


Figure 15-160 Top view of the component

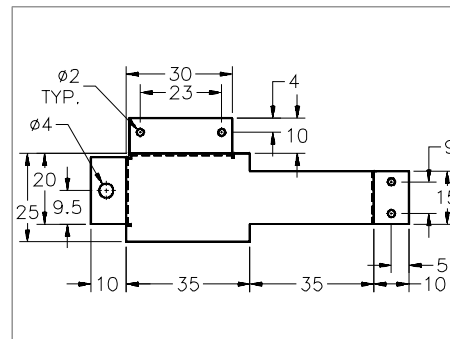


Figure 15-161 Front view of the component

### Answers to Self-Evaluation Test

1. T, 2. F, 3. F, 4. T, 5. Unfold/Fold, 6. F, 7. T, 8. Louvers, 9. T, 10. Corner