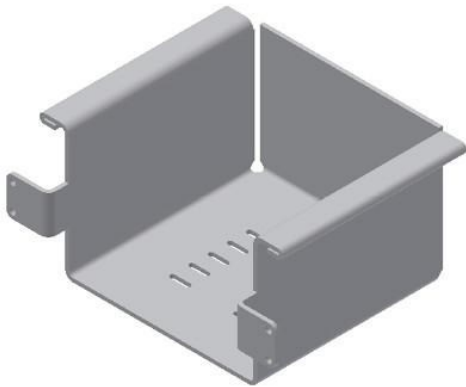# Chapter 6: Sheet Metal Modeling

This chapter will show you to:

- Create a face feature
- Create a Flange
- Create a Contour Flange
- Create a Corner Seam
- Create Punches
- Create a Bend Feature
- Create Corner Rounds
- Flat Pattern

TUTORIAL 1

In this tutorial, you create the sheet metal model shown in figure.

Starting a New Sheet metal File

1. To start a new sheet metal file, click **Get Started Launch > New** on the ribbon.
2. On the **Create New File** dialog, click the **Sheet Metal.ipt** icon, and then click **Create**.


Setting the Parameters of the Sheet Metal part

1. To set the parameters, click **Sheet Metal > Setup > Sheet Metal Defaults** on the ribbon; the **Sheet Metal Defaults** dialog appears.
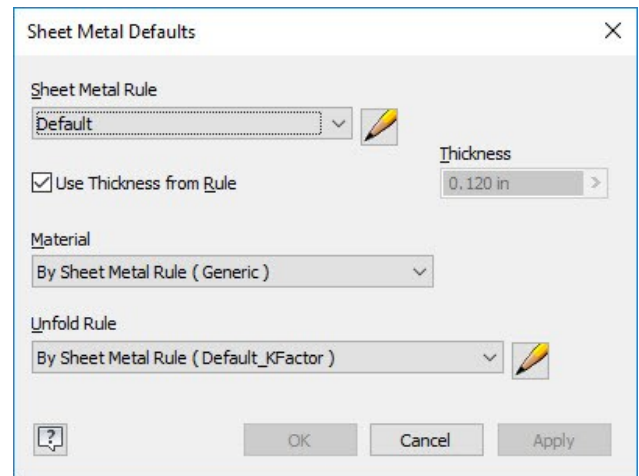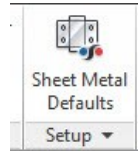

This dialog displays the default preferences of the sheet metal part such as sheet metal rule, thickness, material, and unfold rule. You can change these preferences as per your requirement.

2. To edit the sheet metal rule, click the **Edit Sheet Metal Rule** button on the dialog.
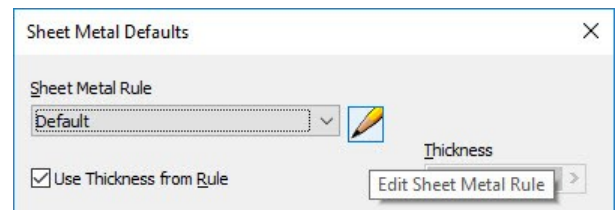

In the **Sheet** tab of the **Style and Standard Editor** dialog, you can set the sheet preferences such as sheet thickness, material, flat pattern bend angle representation, flat pattern punch representation and gap size.
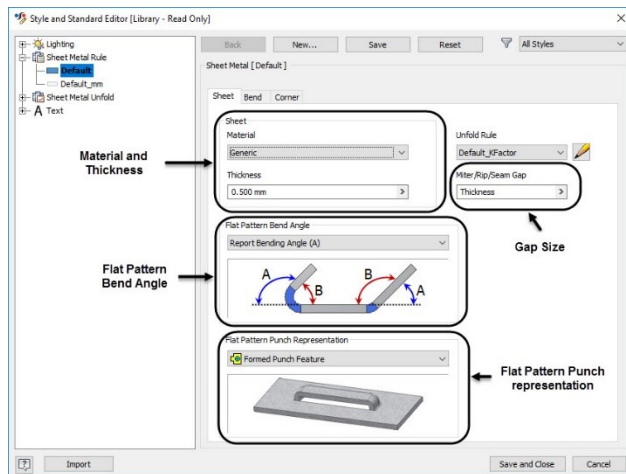
3. In the **Sheet** tab, set the **Thickness** to 0.12 and leave all the default settings.
4. Click the **Bend** tab.

In the **Bend** tab of this dialog, you can set the bend preferences such as bend radius, bend relief shape and size, and bend transition.
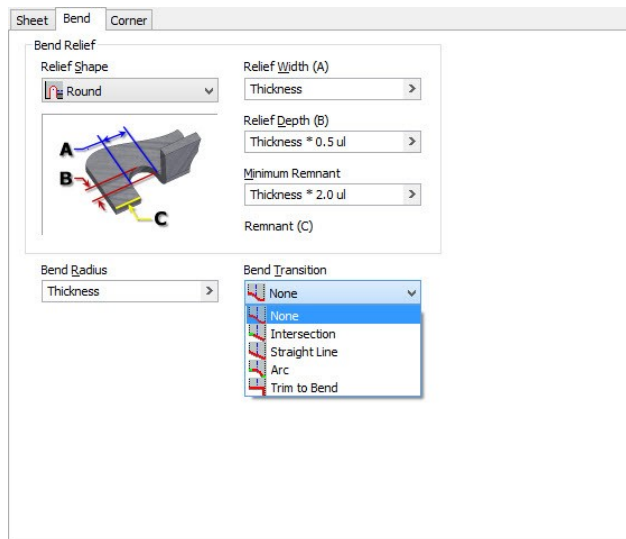
5. Set the **Relief Shape** to **Round**.
6. Click the **Corner** tab.

In the **Corner** tab, you can set the shape and size of the corner relief to be applied at the corners.
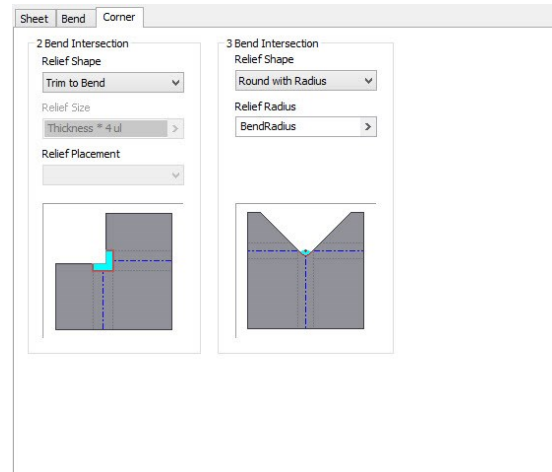

7. After setting the required preferences, click the **Save and Close** button.

The **Unfold Rule** option on the **Sheet Metal Defaults** dialog defines the folding/unfolding method of the sheet metal part. To modify or set a new Unfold Rule, click the **Edit Unfold Rule** button on the **Sheet Metal Defaults** dialog.

On the **Style and Standard Editor** dialog, select the required **Unfold Method**.
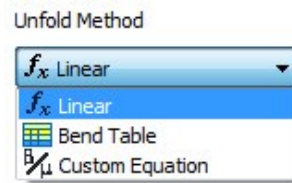
You can define the Unfold rule by selecting the **Linear** method (specifying the K factor), selecting a **Bend Table**, or entering a custom equation. Click **Save and Close** after setting the parameters.

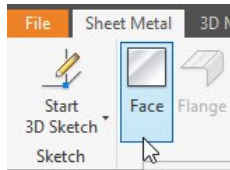8. Close the **Sheet Metal Defaults** dialog.

Creating the Base Feature

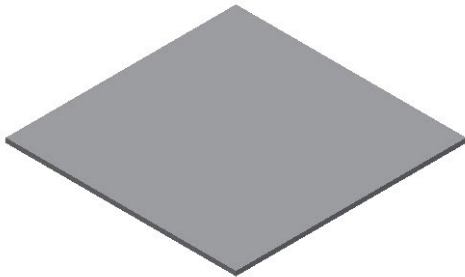1. Create the sketch on the XZ Plane, as shown in figure (Use the **Rectangle Two Point Center** tool).

437
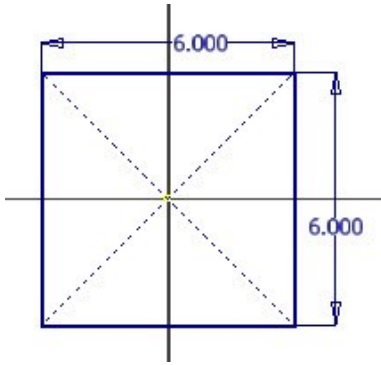
2.   Click **Finish Sketch**.
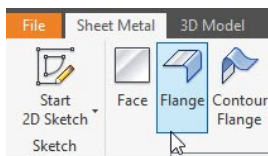3.   To create the base component, click **Sheet Metal > Create > Face** on the ribbon; the **Face** dialog appears.
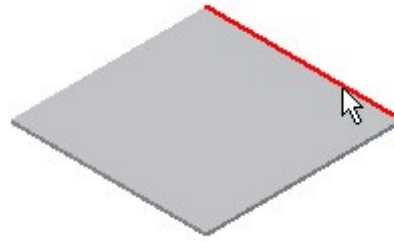

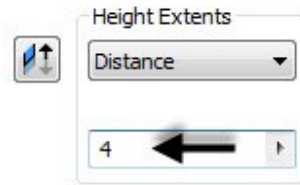4.   Click **OK** to create the tab feature.

Creating the flange

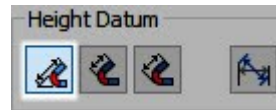1.   To create the flange, click **Sheet Metal > Create > Flange** on the ribbon; the **Flange** dialog appears**.**
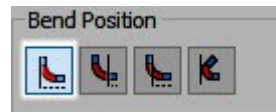

2.   Select the edge on the top face, as shown.
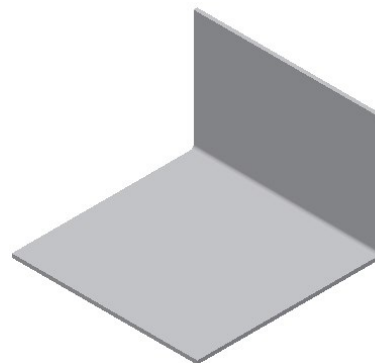

3.   Set the **Distance** to 4.

4.   Click on the **Bend from the intersection of the two outer faces** icon in the **Height Datum** section. This measures the flange height from the outer face.

5.   Under the **Bend Position** section, click the **Inside of the Bend extents** icon.
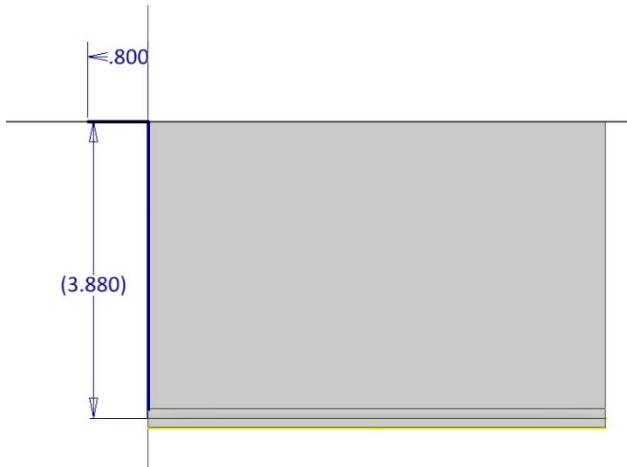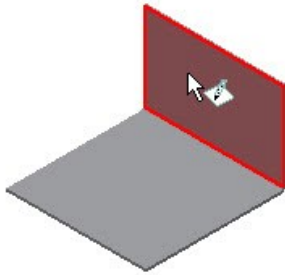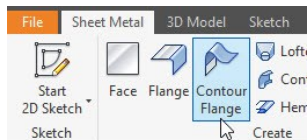

6.   Click **OK** to create the flange.
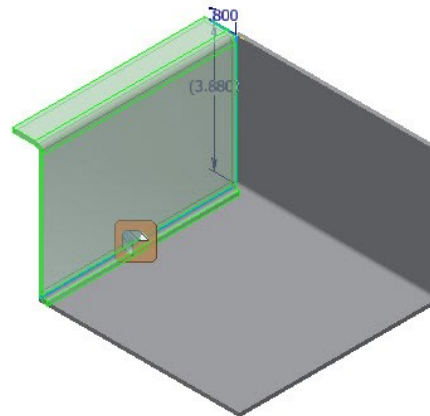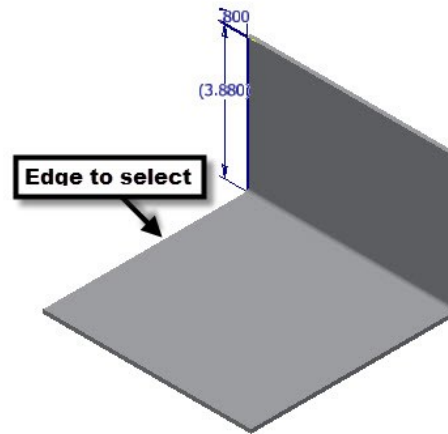

Creating the Contour Flange

1.   Draw a sketch on the front face of the flange, as shown in figure.
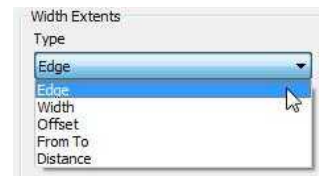
2. Click **Finish Sketch**.
3. To create the contour flange, click **Sheet Metal > Create > Contour Flange** on the ribbon; the **Contour Flange** dialog appears.
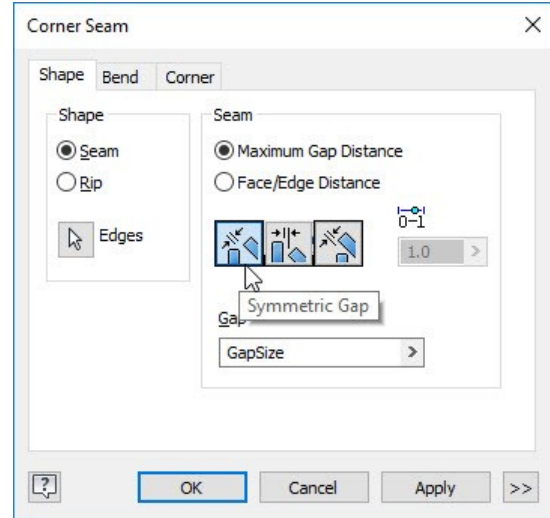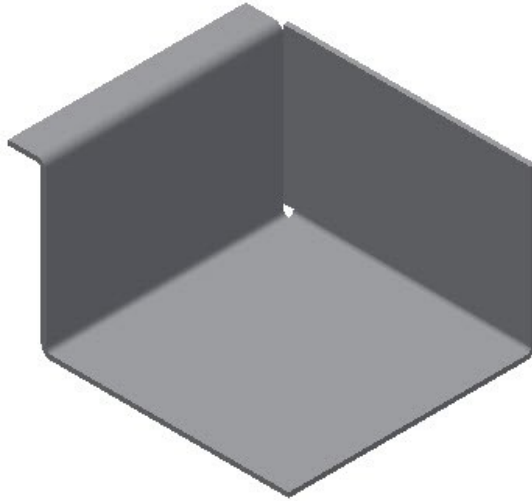

4. Select the sketch from the model.
5. Select the edge on the left side of the top face; the contour flange preview appears.
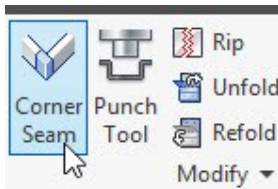
6. Select **Edge** from the **Type** drop-down.

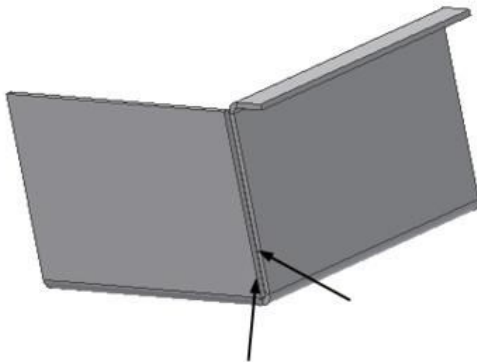7. Click **OK** to create the contour flange.

Creating the Corner Seam

1. To create the corner seam, click **Sheet Metal> Modify > Corner Seam** on the ribbon; the **Corner Seam** dialog appears.

2. Rotate the model.
3. Select the two edges forming the corner.

4. Set the parameters in the **Shape** tab of the dialog, as shown.
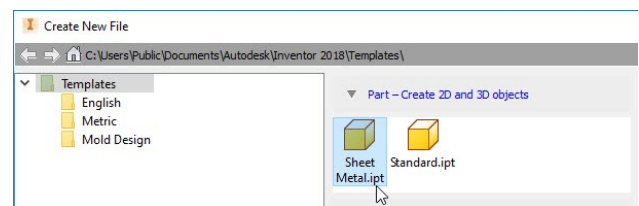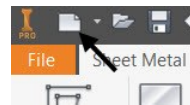

5. Click the **Bend** tab and make sure that the **Default** option is selected in the **Bend Transition** drop-down.
6. Click the **Corner** tab and set the **Relief Shape** to **Round**.

You can also apply other types of relief using the options in the **Relief Shape** drop-down.
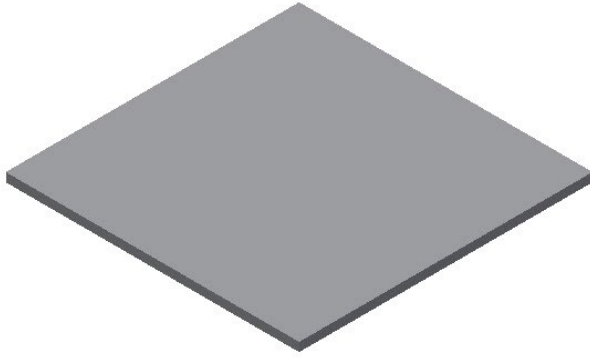
7. Click **OK**.

Creating a Sheet Metal Punch iFeature

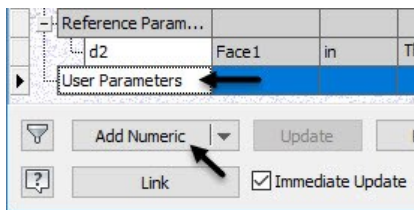1. Open a new sheet metal file using the **Sheet Metal.ipt** template.
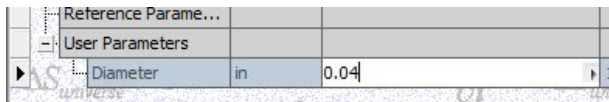

2. Create a sheet metal face of the dimensions 4x4.

3. Click **Manage > Parameters > Parameters** f_x on the ribbon; the **Parameters** dialog appears.

4. Select the **User Parameters** row and click the **Add Numeric** button on the dialog. This adds a new row.

5. Enter **Diameter** in the new row.

6. Set **Unit Type** to **in** and type-in 0.04 in the **Equation** box.

7. Likewise, create a parameter named **Length** and specify its values, as shown below.
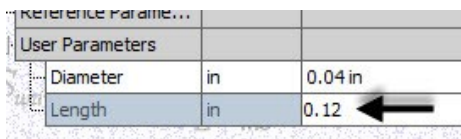

8. Click **Done**.

9. Click **Sheet Metal > Sketch > Start 2D Sketch** on the ribbon.

10. Select the top face of the base feature.

11. On the ribbon, click **Sketch > Create > Rectangle drop-down > Slot Center to Center**.

12. Click to define the first center point of the slot.

13. Move the cursor horizontally and click to define the second center point of the slot.

14. Move the cursor outward and click to define the slot radius.
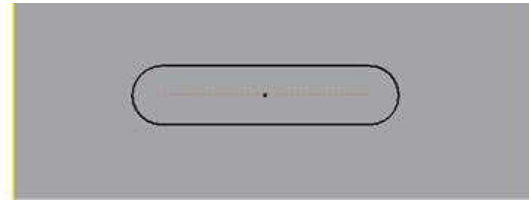
15. Click **Dimension** on the **Constrain** panel and select the round end of the slot.

16. Click to display the **Edit Dimension** box.

17. Click the arrow button on the box and select **List Parameters** from the shortcut menu; the **Parameters** list appears.
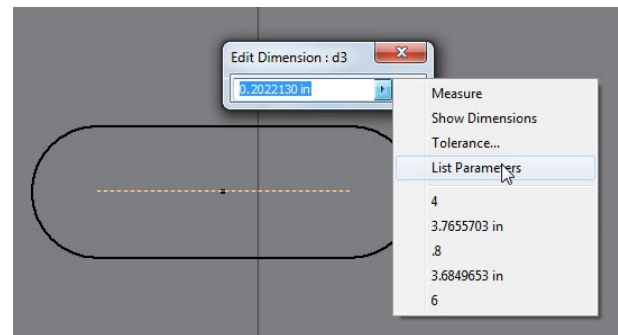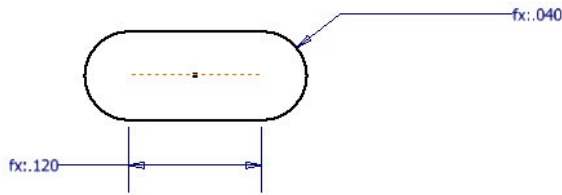

18. Select **Diameter** from the list and click the green check on the **Edit Dimension** box.
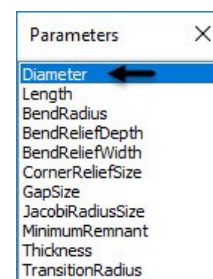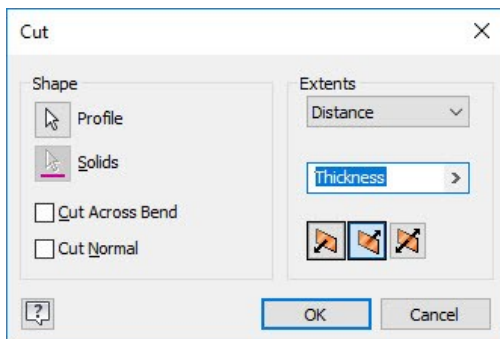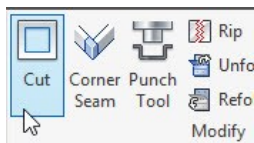

19. Likewise, dimension the horizontal line of the slot and set the parameter to **Length**.
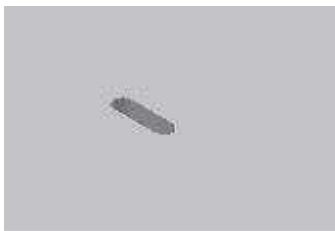
---

20. Click the **Point** button on the **Create** panel and place it at the center of the slot.
21. Delete any projected edges (yellow lines) from the sketch.
22. Click **Finish Sketch**.
23. Click **Sheet Metal > Modify > Cut** on the ribbon; the **Cut** dialog appears.

24. Accept the default values and click **OK** to create the cut feature.


25. Click **Manage > Author > Extract iFeature** on the ribbon; the **Extract iFeature** dialog appears.
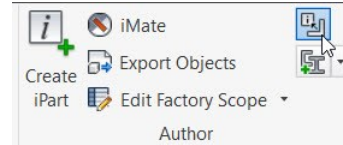

26. On the dialog, select **Type > Sheet Metal Punch iFeature**.
27. Select the cut feature from the model geometry or from the Browser window. The parameters of the cut feature appear in the **Extract iFeature** dialog.

Next, you must set the **Size Parameters** of the iFeature.

28. Set the **Limit** of the **Diameter** value to **Range**. The **Specify Range** dialog appears**.**
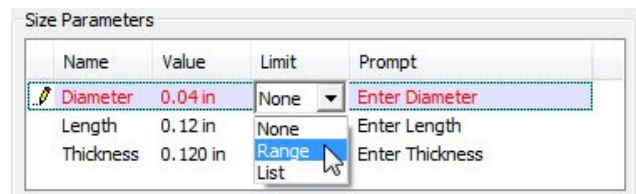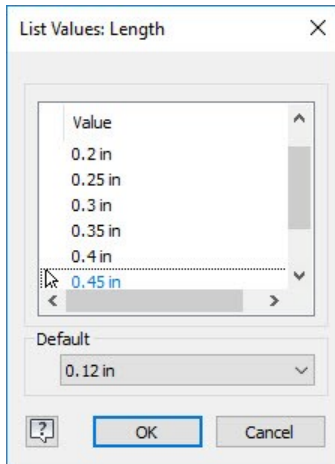

29. Set the values in the **Specify Range** dialog, as shown below and click **OK**.
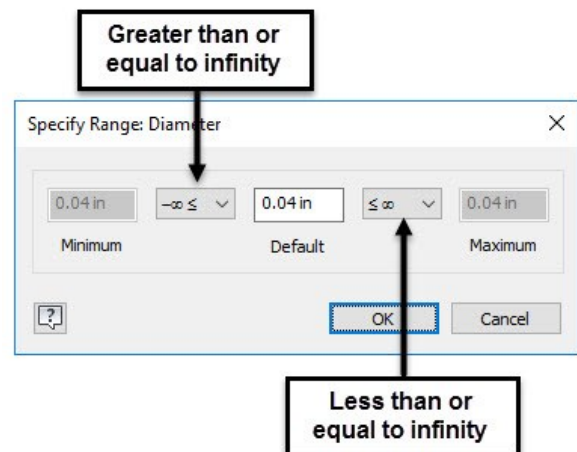
30. Set the **Limit** of the **Length** value to **List**. The **List Values** dialog appears.
31. Click on **Click here to add value** and enter 0.2 as value.

32. Likewise, type-in values in the **List Values** dialog, as shown below.

33. Click **OK**.
34. Set the **Limit** of the **Thickness** value to **Range**. The **Specify Range** dialog appears.
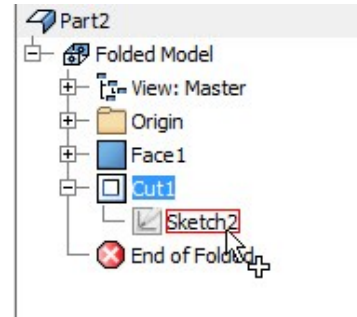35. Set the values in the **Specify Range** dialog, as shown below. Next, click **OK**.
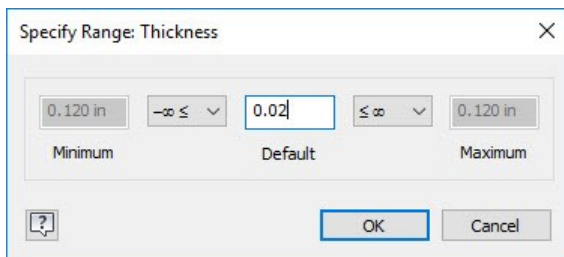
Next, you need to select the center point of the slot. This point will be used while placing the slot.
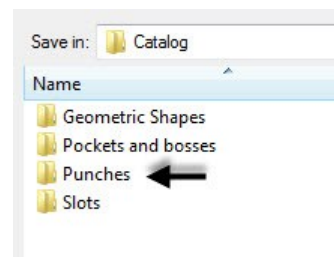
36. Click the **Select Sketch** button on the **Extract iFeature** dialog.
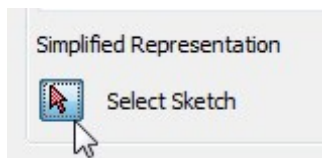
37. Select the sketch of the cut feature from the Browser window.

38. Click **Save** on the dialog; the **Save As** dialog appears.
39. Browse to the **Punches** folder and enter **Custom slot** in the **File name** box.
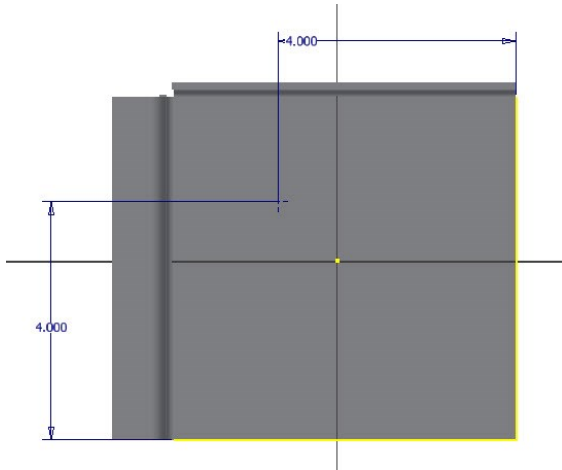
40. Click **Save** and **Yes**.
41. Click **File Menu > Save**.
42. Save the sheet metal part file as Custom slot.
43. Switch to the sheet metal file of the current tutorial.

Creating a Punched feature
1. Start a sketch on the top face of the base sheet.
2. On the ribbon, click **Sketch > Create > Point**.
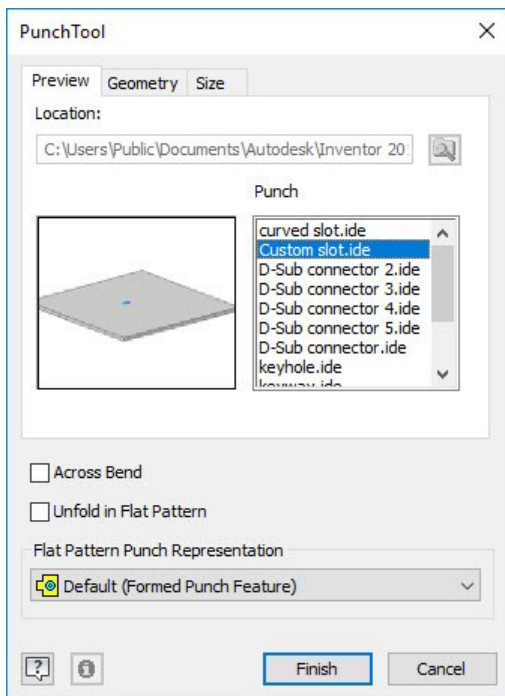3. Place a point and add dimensions to it, as shown below.

4. Click **Finish Sketch**.

5. To create the punch, click **Sheet Metal > Modify > Punch Tool** on the ribbon; the **PunchTool Directory** dialog appears.
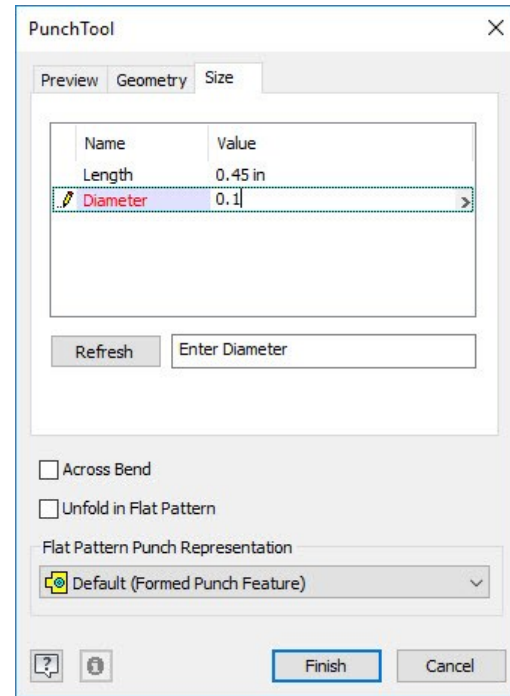

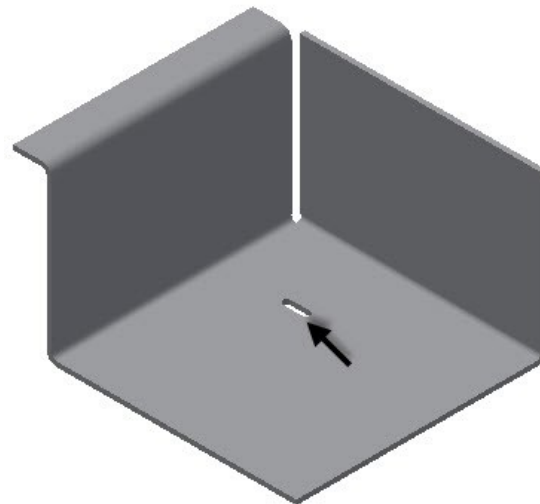6. Select Custom slot.ide from the dialog and click **Open**; the **PunchTool** dialog appears.

7. Click the **Size** tab on the **PunchTool** dialog.

8. Set **Length** to 0.45 and **Diameter** to 0.1.

9. Click **Refresh** to preview the slot.
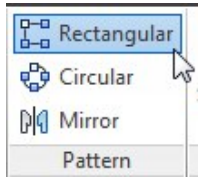10. Click **Finish** to create the slot.

Note: If the slot is not oriented as shown in figure, then click the **Geometry** tab on the **PunchTool** dialog and type-in **90** in the **Angle** box.

Creating the Rectangular Pattern

1. Click **Sheet Metal > Pattern > Rectangular**

Pattern on the ribbon. The **Rectangular Pattern** dialog appears.
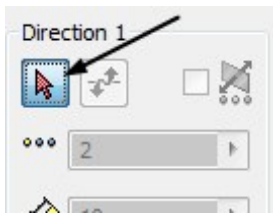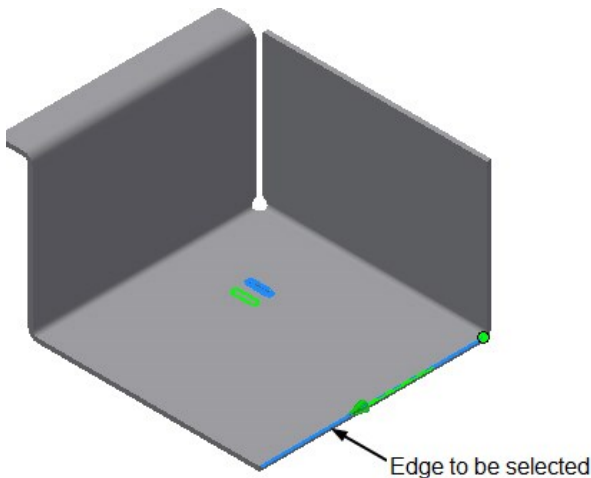

2. Select the slot feature.

*You can also select multiple solid bodies from the graphics window using the **Pattern Solids** option.*
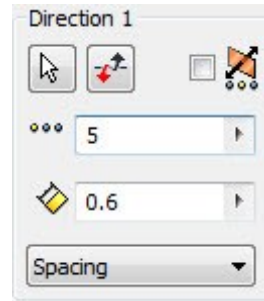
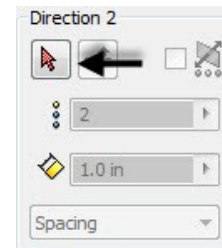3. Click the **Direction 1** button on the dialog.

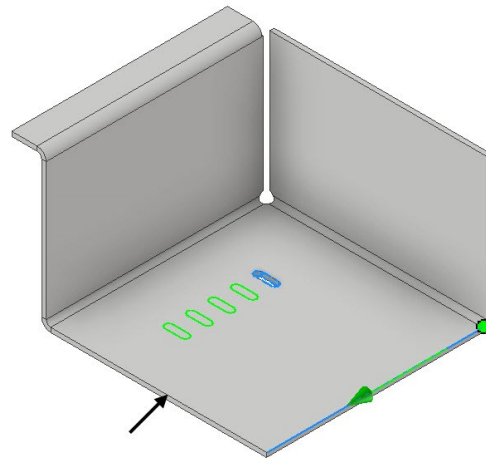4. Select the edge of the base feature, as shown below.

Edge to be selected

5. Select **Spacing** from the drop-down located in the **Direction 1** group.
6. Specify **Column Count** as 5.
7. Specify **Column Span** as 0.6.

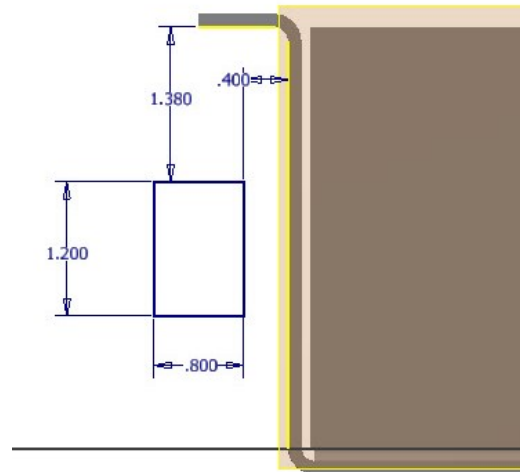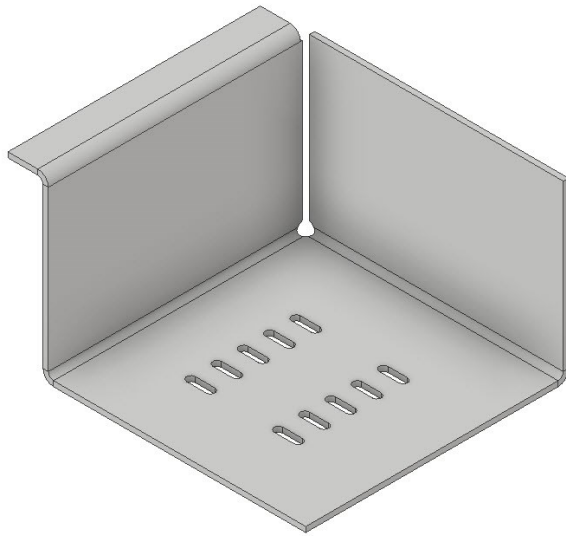8. Click the **Direction 2** button on the dialog.

9. Select the edge on the base feature, as shown below.

10. Click the **Flip** button in the **Direction 2** section to make sure the arrow is pointed toward right.
11. Select **Spacing** from the drop-down located in the **Direction 2** group.
12. Specify **Column Count** as **2**.
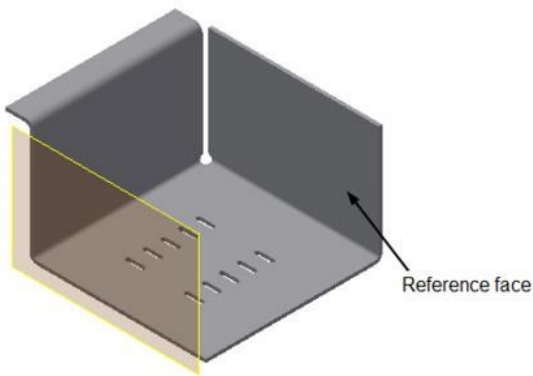13. Specify **Column Span** as 2.

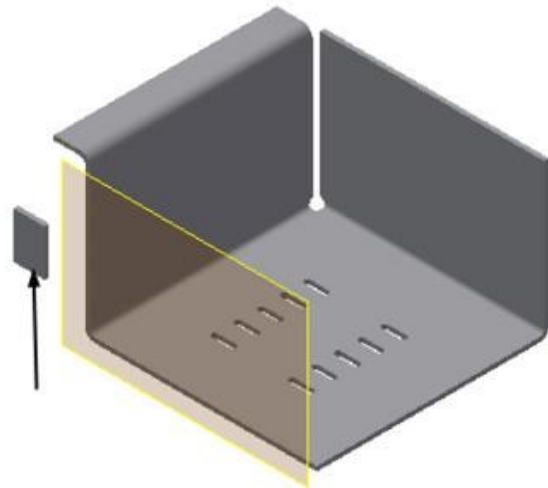14. Click **OK** to create the pattern.

3. Click **Finish Sketch**.
4. Click **Sheet Metal > Create > Face** on the ribbon and create a face feature.

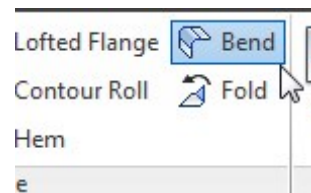## Creating the Bend Feature

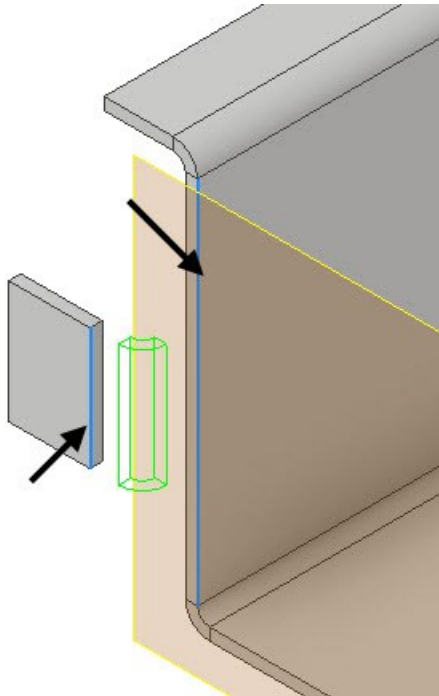1. Create a plane parallel to the front face of the flange feature. The offset distance is 6.3.
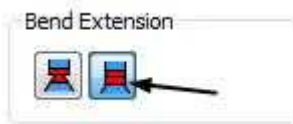

2. Create a sketch on the new work plane.

5. Click **Sheet Metal > Create > Bend** on the ribbon. The **Bend** dialog appears.
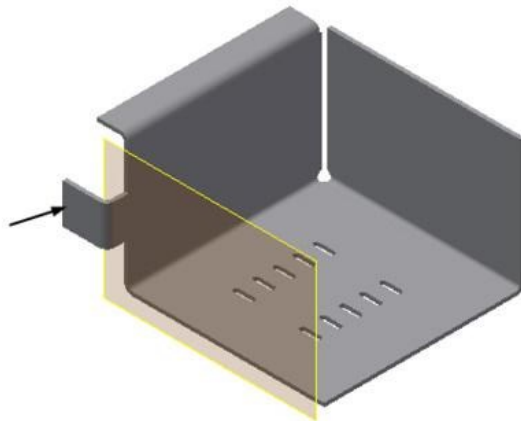

6. Select the edges from the model, as shown below.

446

7. Make sure the **Bend Extension** is set to perpendicular.

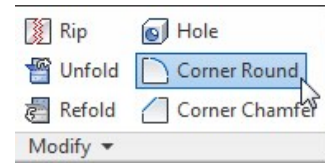8. Click **OK** to create the bend feature.

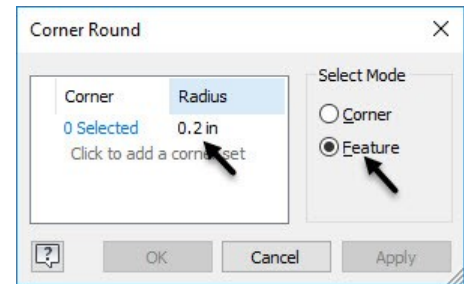9. Hide the work plane (Right-click on it and uncheck **Visibilty**).

Applying a corner round

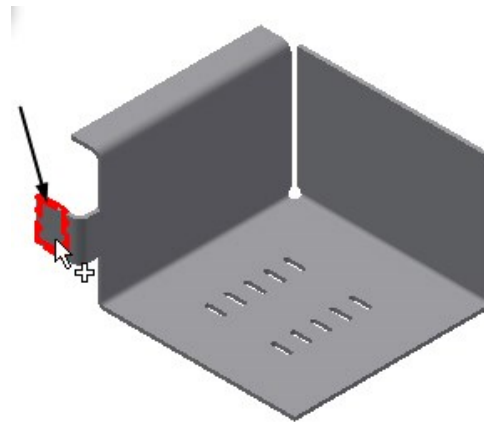1. To apply a corner round, click **Sheet Metal >**

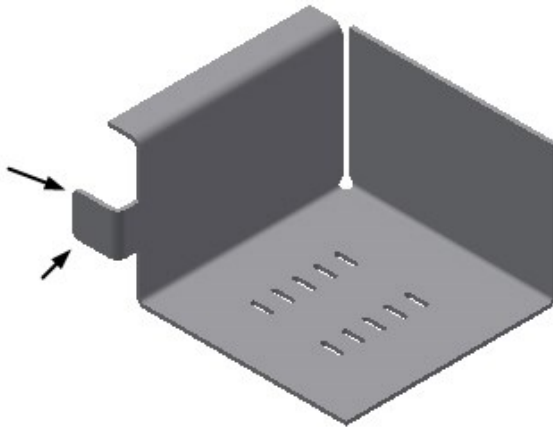**Modify > Corner Round** on the ribbon; the **Corner Round** dialog appears.

2. Set the **Radius** value to 0.2.
3. Set the **Select Mode** to **Feature**.
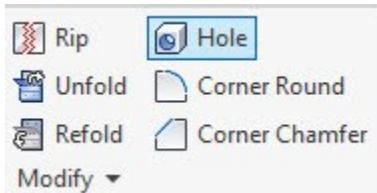

4. Select the face feature from the model.

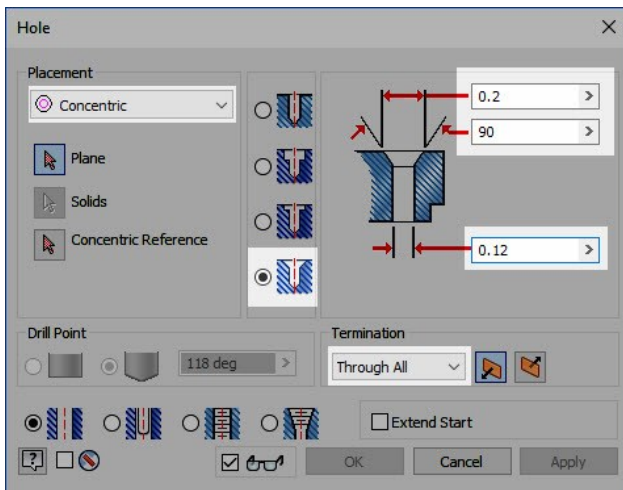
5. Click **OK** to apply the rounds.
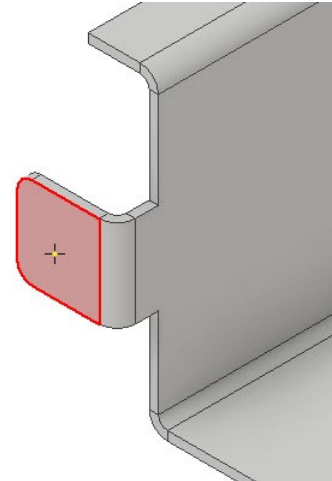
Creating Countersink holes

1. Click **Sheet Metal > Modify > Hole** on the ribbon; the **Hole** dialog appears.
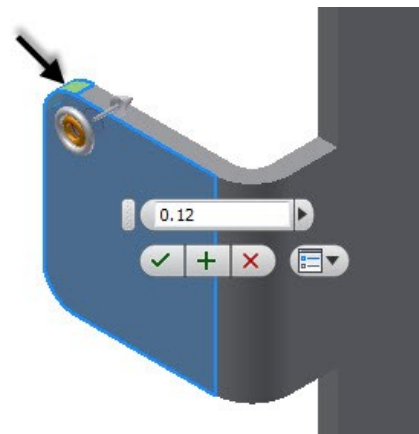

2. Set the **Placement** method to **Concentric**.
3. Set the hole type to **Countersink**.
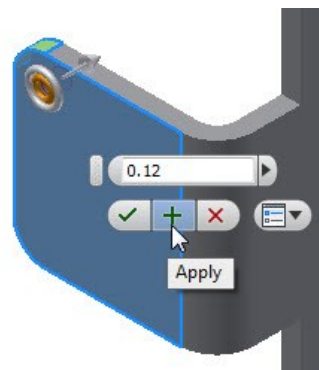4. Set the other parameters on the dialog, as shown below.

5. Click on the face of the flange, as shown below.

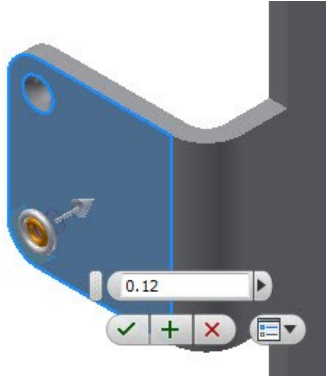6. Select the corner round as the concentric reference.

7. Click **Apply**.

8. Again, click on the flange face and select the other corner round as the concentric reference.
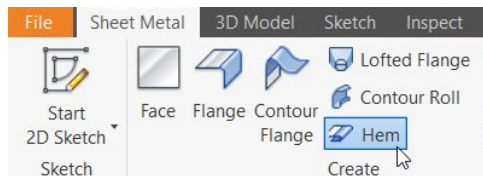
9. Click **OK** to create the countersink.

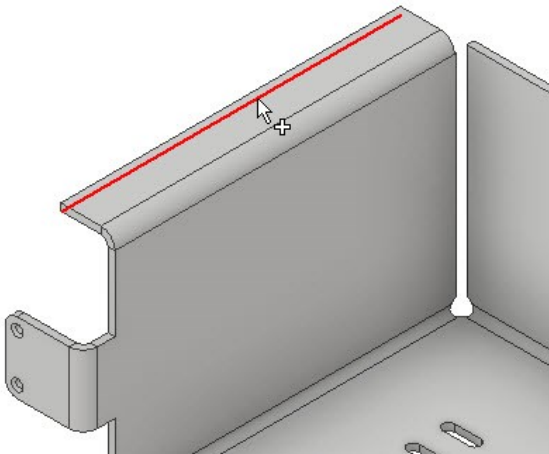## Creating Hem features

1. To create the hem feature, click **Sheet Metal > Create > Hem** on the ribbon; the **Hem** dialog appears.
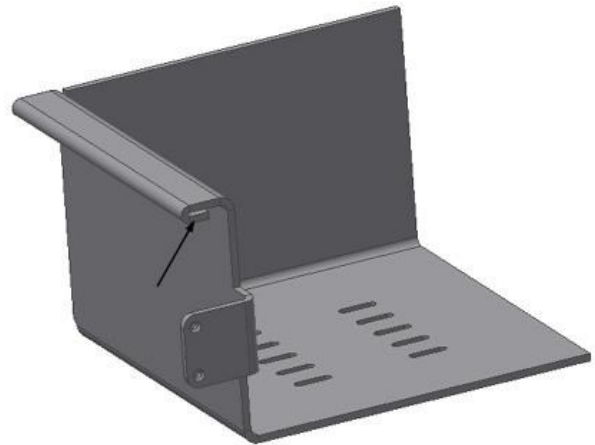

2. Set the **Type** to **Single**.

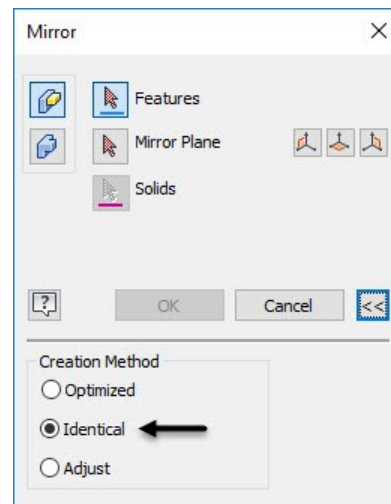3. Select the edge of the contour flange, as shown below.

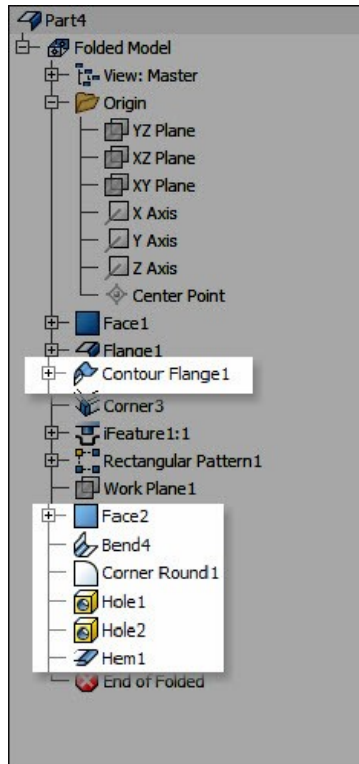4. Leave the default settings of the dialog and click **OK** to create the hem.

Mirroring the Features

1. Click **Mirror** on the **Pattern** panel; the **Mirror** dialog appears.
2. Click **>>** at the bottom of the dialog and make sure the **Creation Method** is set to **Identical**.
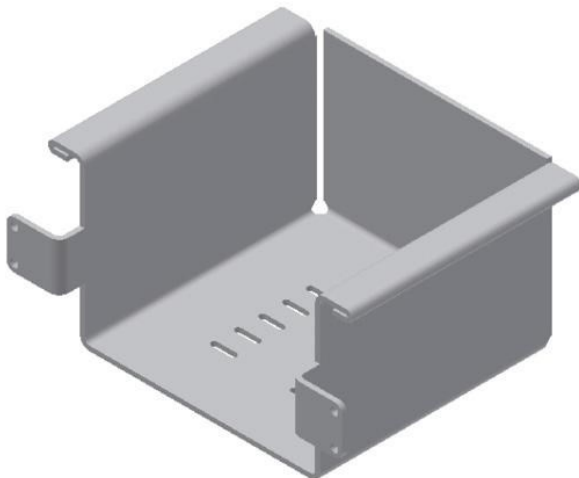

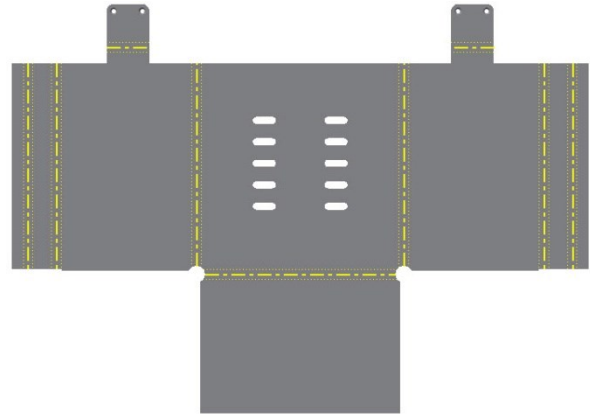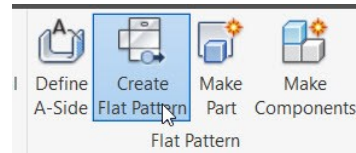3. Select the features from the Browser window, as shown below.

4. Click the **Origin YZ Plane** button on the dialog
5. Click **OK** to mirror the feature.
6. Create a corner seam between the mirrored counter flange and flange.
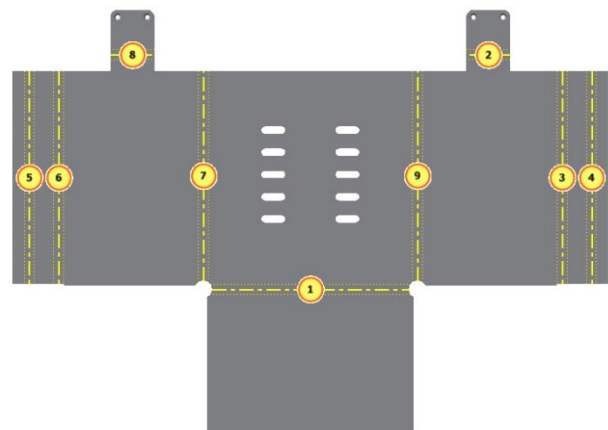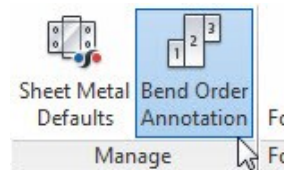

Creating the Flat Pattern

1. To create a flat pattern, click **Sheet Metal > Flat Pattern > Flat Pattern** on the ribbon.


You can set the order in which the bends will be annotation.

2. Click the **Bend Order Annotation** button on the **Manage** panel of the **Flat Pattern** tab. The order in which the bends will be annotated is displayed.

---

3. To change the order of the bend annotation, click on the balloon displayed on the bend. The **Bend Order Edit** dialog appears.

4. Select the **Bend Number** check box and enter a new number in the box.
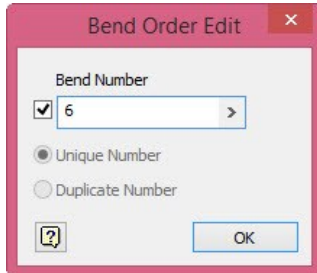

5. Click **OK** to change the order.

6. To switch back to the folded view of the model, click **Go to Folded Part** on the **Folded Part** panel.


7. Save the sheet metal part.

## Creating 2D Drawing of the sheet metal part

1. On the Quick Access toolbar, click the **New** button.

2. On the **Create New File** dialog, double-click on **Standard.idw**.

3. Activate the **Base View** tool.

4. Click Home icon on the ViewCube.

5. Leave the default settings on the Drawing View dialog and click **OK**.

6. Click and drag the drawing view to top right corner of the drawing sheet.
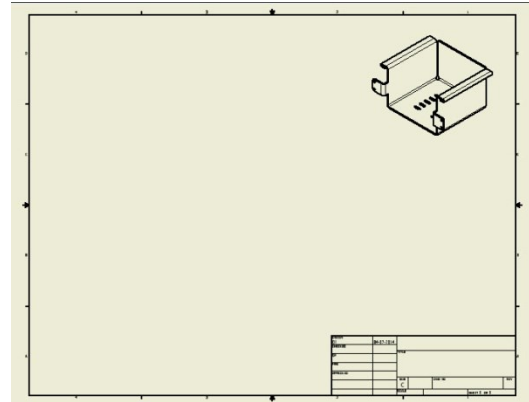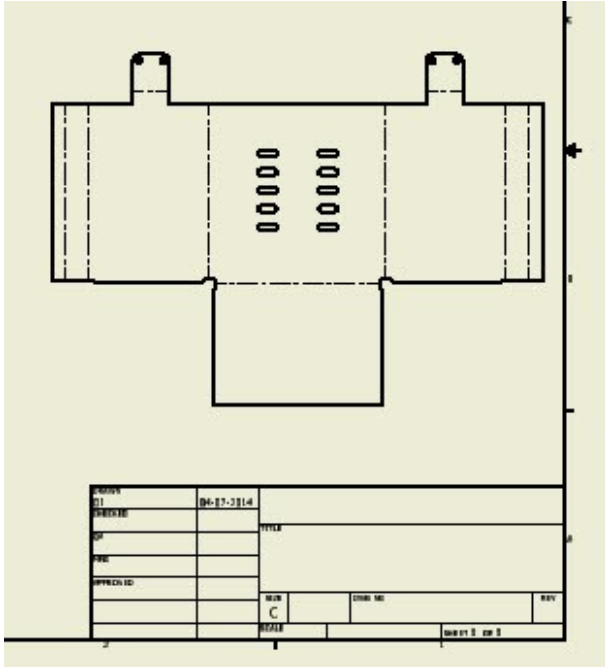

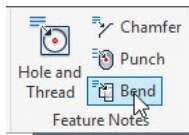7. Likewise, create the front, and top views of the sheet metal part.
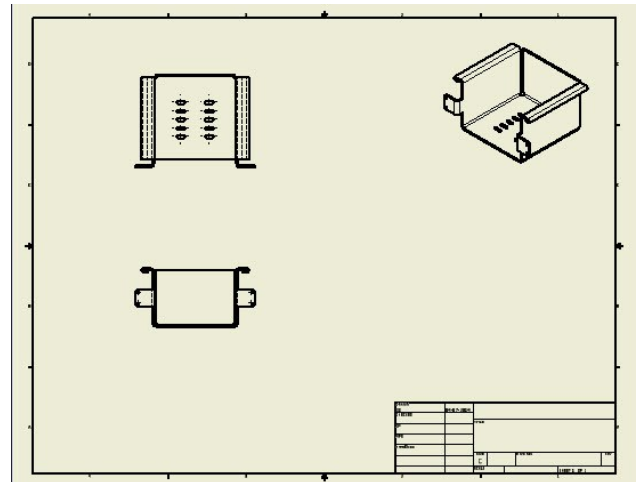
8. Activate the **Base View** tool and select **Sheet Metal View > Flat Pattern** on the **Drawing View** dialog.
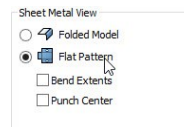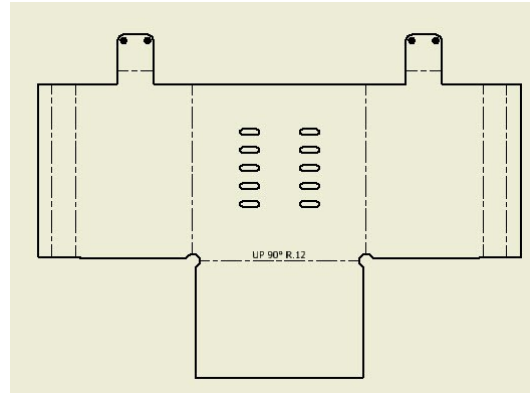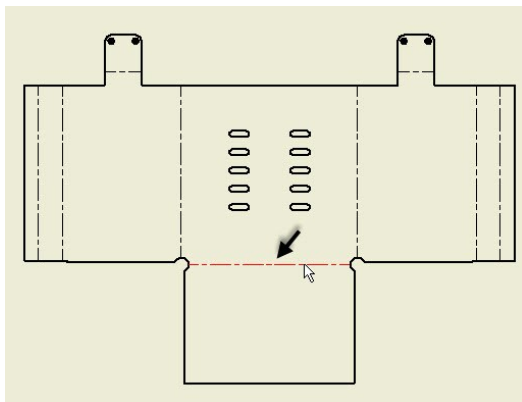

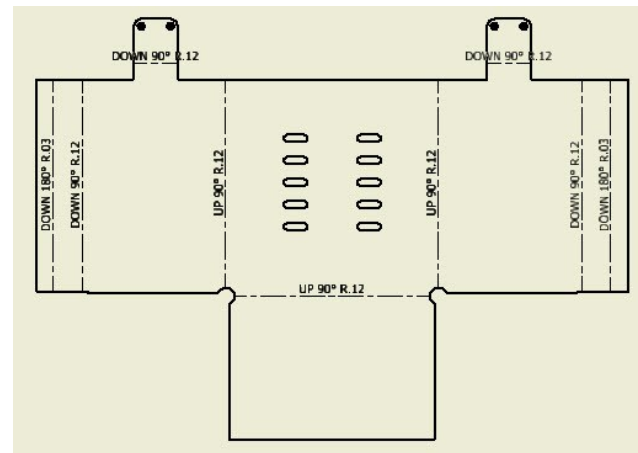9. Place the flat pattern view below the Isometric view.

---

10. To add bend notes to the flat pattern, click
    Annotate > Feature Notes > Bend on the ribbon.
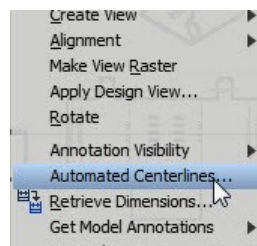

11. Click the horizontal bend line on the flat pattern
    to add the bend note.

12. Likewise, select other bend lines on the flat
    pattern. You can also drag a selection box to
    select all the bend lines from the flat pattern
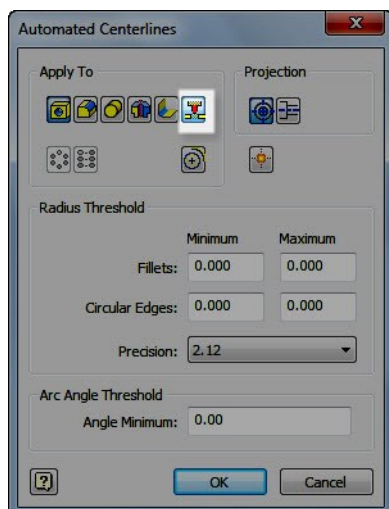   view.

13. To add centerlines to the flat pattern view, click
    the right mouse button on it and select
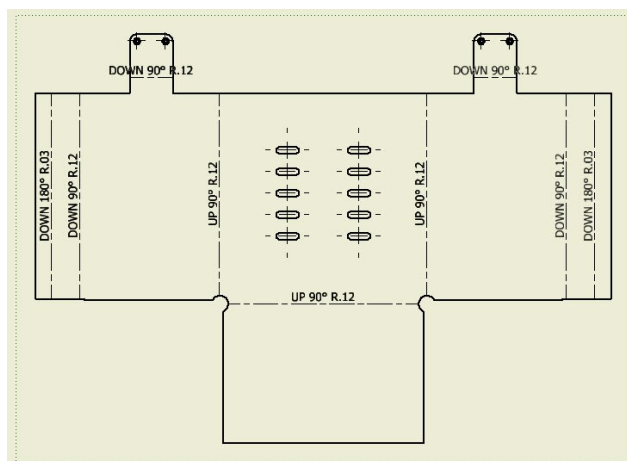   **Automated Centerlines**.

14. On the **Automated Centerlines** dialog, click the
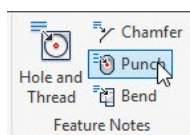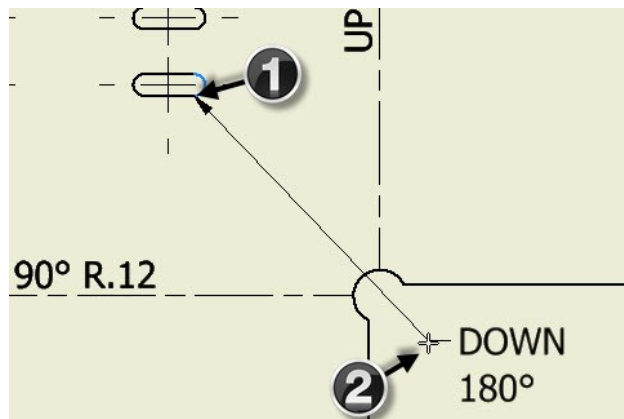   **Punches** button under the **Apply To** section.

15. Click **OK** to add centerlines to the flat pattern view.

16. Likewise, add centerlines to other views on the drawing sheet.
17. To add a punch note, click **Annotate > Feature Notes > Punch** on the ribbon.

18. Zoom into the flat pattern view and click on the arc of the slot.
19. Move the pointer and click to create annotation.

20. Use the **Retrieve Dimension** and **Dimension** tools to add dimensions to drawing.
21. Save and close the drawing and sheet metal part.