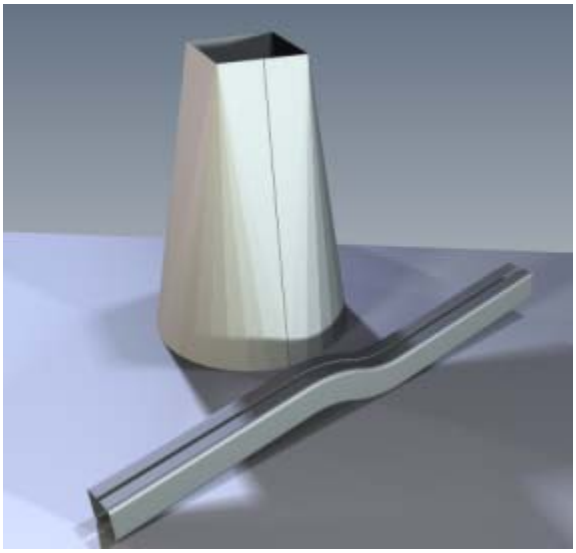
Sheet Metal Parts 2

Topics in this section

Explore Sheet Metal Functionality



Category	**Mechanical Design**
Time Required	60 minutes
Tutorial File Used	Start a new sheet metal part (metric) contour_roll-start.ipt sm_part2_model-completed.ipt (finished version)

Create a lofted flange feature, and then rip and flatten it. Work with the flat pattern to explore many sheet metal features.

Objectives

- Lofted flange features
- Rip features
- Bend order sequence
- Cosmetic centerline features
- Contour Roll features
- Unfold features, with features added to the flattened model
- Added refold features

Prerequisites

- Complete the tutorial **Sheet Metal Parts.**
- Know how to set the active project and navigate the model space with the various view tools.
- See the Help topic “Getting Started” for further information.
- Ensure that **Autoproject edges for sketch creation and edit** on the **Sketch** tab of the Application Options dialog box is not checked.

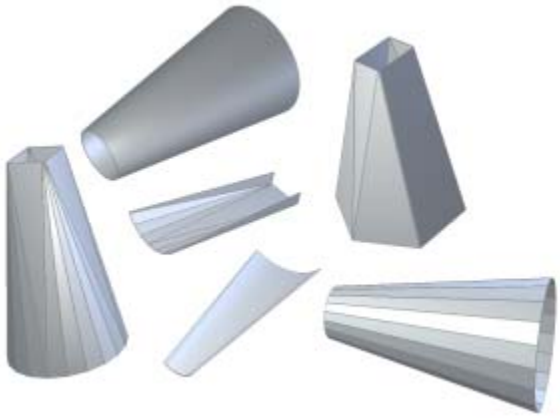
Navigation Tips

- Use **Show** in the upper-left corner to display the table of contents for this tutorial with navigation links to each page.

- Use **Forward** in the upper-right corner to advance to the next page.

Next

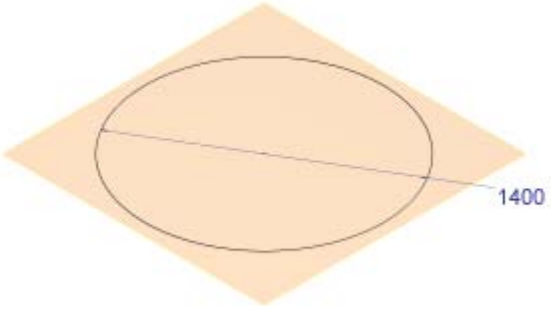
Lofted Flange - Select Profile Sketches



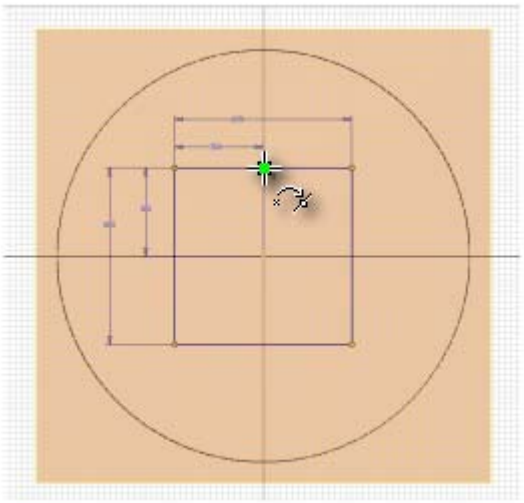
The Lofted Flange feature provides a way to create transitional sections in your model. They may be the only (or primary) feature in a model or they may be part of a more complex design.

The Lofted Flange feature requires the selection of two profile sketches. The profiles can be open or closed (or one of each) and can be on parallel or non-parallel sketch planes. The resulting feature can be optionally targeted at either a press brake or die-form manufacturing process.

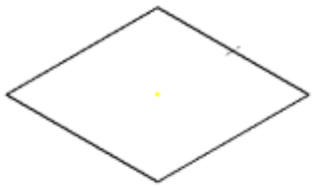
1. To begin this exercise, begin a new sheet metal part using the **Sheet Metal (mm).ipt** template.
2. In the open sketch, create a **1400-mm** circle centered on 0,0.

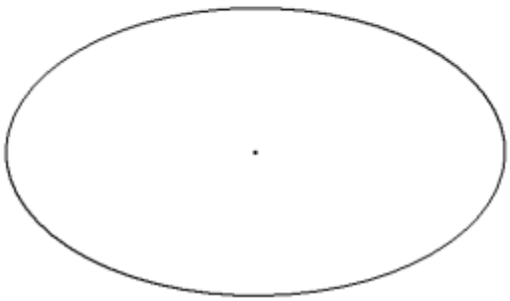


3. On the ribbon, click Sketch tab ➤ Exit panel ➤ Finish Sketch.
4. Create a Work Plane offset (up) from the XY Origin Plane by **2000 mm**.
5. On this new offset Work Plane, create a 2D sketch.
6. In the sketch create a **600 mm x 600-mm** square. Apply **300-mm** dimensions so that the square is centered on 0,0.
7. Place a point on the mid-point of one side of the square.
   Note This point is not required for the Lofted Flange; however, you will use it later to create a Rip feature to flatten the Lofted Flange.



8. Click Sketch tab ➤ Exit panel ➤ Finish Sketch.
9. Hide the work plane that you created and the XY origin plane (if displayed). Hide the dimensions on both Sketch1 and Sketch2. Your model should appear as shown in the following image from the default Home View (**F6**).





Next, you create the Lofted Flange.

Previous | Next

Lofted Flange - Create the Flange

1. Before you create the Lofted Flange, on the ribbon click Sheet Metal tab ➤ Setup panel ➤ Sheet Metal Defaults.
2. In the Sheet Metal Defaults dialog box, clear the **Use Thickness from Rule** option. Enter a value of 4 mm in the **Thickness** value entry field.
3. Click **OK** to accept the new material thickness and close the dialog box.
4. Click Sheet Metal tab ➤ Create panel ➤ Lofted Flange.
5. In the graphics window, click to select the sketched square as **Profile 1**.
6. Click to select the sketched circle as **Profile 2**. A preview of the resulting Lofted Flange displays using the default settings.
7. Since **Press Brake** is the selected **Output** option, the preview shows a Lofted Flange that can be created using straight bends. This results in a faceted approximation of the circle. There are three methods that you can use to adjust the resulting facets. In this exercise, you increase the default value for the **Chord Value**. Highlight the value of **0.5 mm**, and enter a new value of 4 mm. Notice that your preview adjusts to show fewer facets.

   Note If you have die-forming fabrication available, you can optionally select **Die Form** as the **Output** option. Doing so results in a smooth, conical transition from the circular profile to the square profile.
8. Another optional selection determines if the material thickness is on one side or the other of the sketched profile. To see this better, zoom in to the point that you created on the square profile sketch. By default, the material is offset to the outside of the selected profile. In this exercise, you want the dimensioned size of the profile to represent the outside of the resulting part. The material must be offset to the inside of the profile. Click the middle **Flip Side**.

   Notice the material thickness now previews to the inside of the profile.
9. Click **OK** in the Lofted Flange dialog box to accept the edits you have made, create the Lofted Flange, and close the dialog box.

Because you selected two closed profiles to create this Lofted Flange, the model will not currently create a flat pattern.

Next, you add a Rip feature to allow the model to flatten.

Previous | Next

Rip

Like its physical counterpart, a folded sheet metal model that forms a continuous tube-like shape cannot be flattened. The Rip feature provides an easy way to create a cut in a face of the model that will allow the flat pattern to be produced.

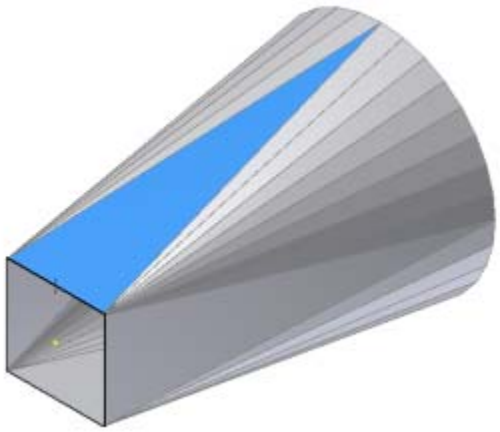
To create a Rip feature, you select a face of the model and (optionally) either one or two points that lie on the selected face. If you select an outside face, any points selected must be on an edge of the outside face. Optionally, you might select an entire face to be removed.

Tip In this exercise, the point used to locate the Rip feature was added to one of the Lofted Flange profile sketches. Another technique is to create a 2D sketch on a flat face of a Lofted Flange targeted for Press Brake output. Then place a point at a strategic vertex or edge midpoint.

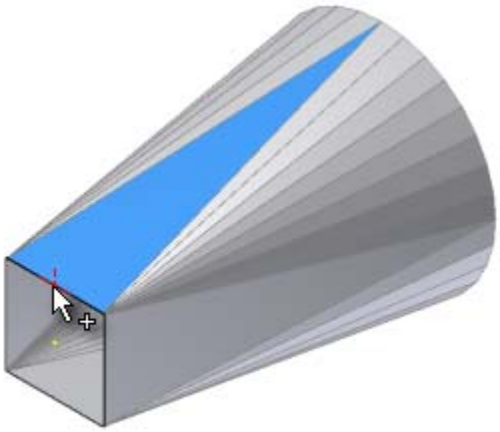
1. In the Model browser, click the **+** to the left of **Lofted Flange1**. Right-click **Sketch2**, and select **Visibility** in the context menu to make Sketch2 visible.

2. Adjust your view of the model so that the edge of the sketched square which contains the point is visible on top.

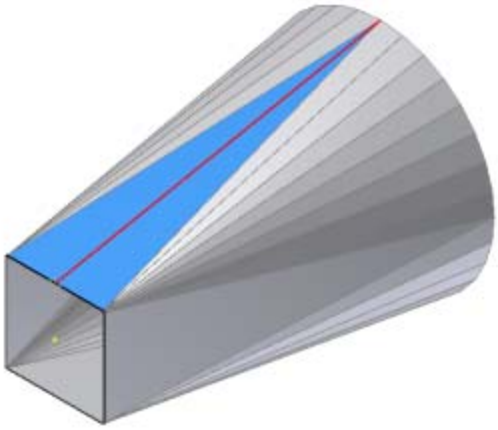
3. On the ribbon, click Sheet Metal tab ➤ Modify panel ➤ Rip.
4. Select the face to be ripped (which contains the sketch point along the edge).



5. Select the point you previously created that defines the location of the single-point Rip.



With the point selected, the Rip feature previews.



Note The Rip can cut a bend face adjacent to the selected face; however, a rip cannot cut across a bend face and through a second face. In this example, if the point was located anywhere other than the midpoint of the edge, the rip could not be created.

6. Click **OK** in the Rip dialog box to create the Rip feature and close the dialog box.
7. In the Model browser, right-mouse select **Sketch2**, and click **Visibility** in the pop-up context menu to switch off **Sketch2** visibility.

Previous | Next

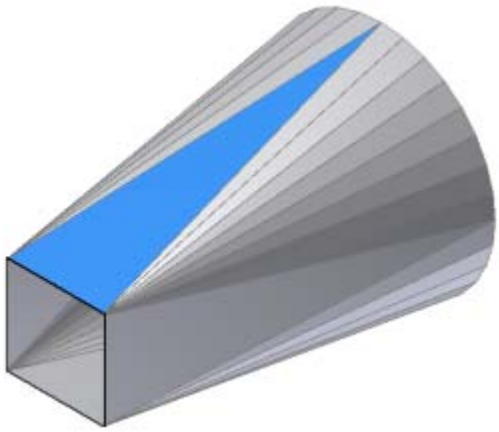
Rip (continued)

The previous Rip feature exercise directed you to create a sketch point to serve as the rip point. While the creation of sketch points are required for certain rip workflows, there are many instances where sketch points are not necessary. Inventor also accepts work points, midpoints on edges, or endpoints on face vertices as valid rip point selections.

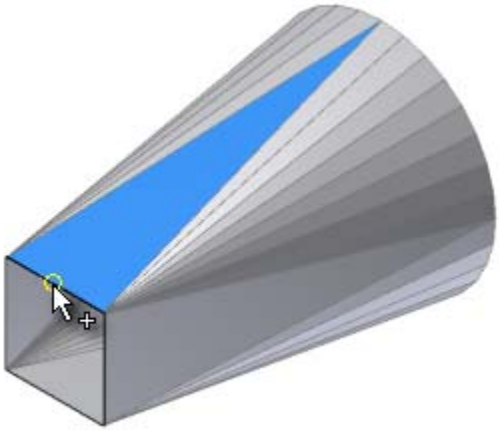
Now, you will delete the rip feature and try creating it once again. But this time, you will use the midpoint on the top edge of the 600mm x 600mm square as the rip point.

1. First, make sure that the **Sketch2** visibility is turned off as previously directed.
2. Next, right-click the **Rip** node in the Model browser and select **Delete** from the pop-up context menu. Your model should appear as shown.

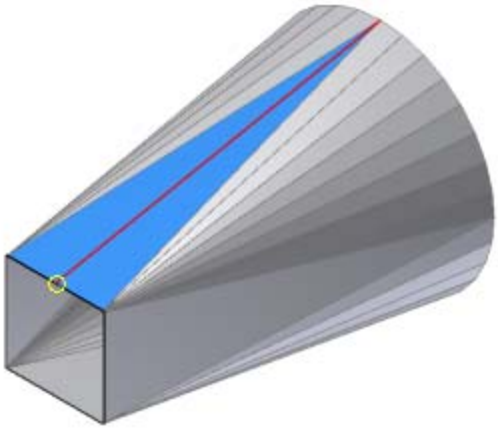
3. On the ribbon, click Sheet Metal tab ➤ Modify panel ➤ Rip.
4. As you did before, select the identical top face to rip.



5. Next, move your cursor to the midpoint of the top edge. When the midpoint appears, click to select.



6. With the midpoint selected, the Rip feature previews.



7. Click **OK** in the Rip dialog box to create the Rip feature and close the dialog box.

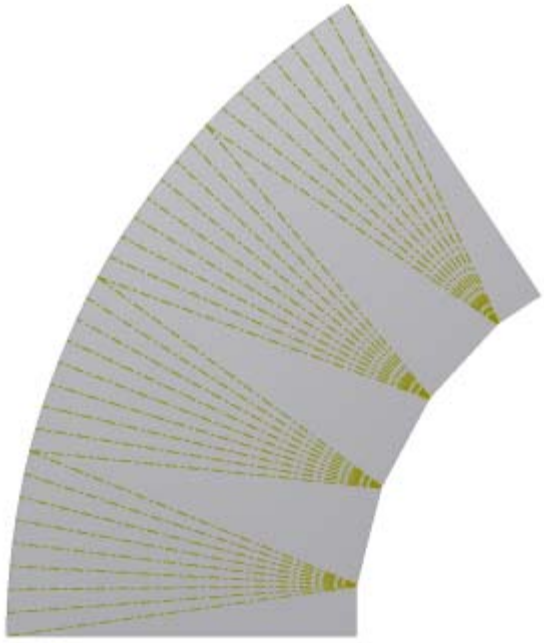
Next, you will create a flat pattern of the ripped Lofted Flange.

Previous | Next

Flatten the Ripped Lofted Flange

Now that the Lofted Flange has had a Rip feature applied it is no longer a continuous closed shape. It is now possible to create a flat pattern suitable for manufacture.

1. On the ribbon, click Sheet Metal tab ▶ Flat Pattern panel ▶ Create Flat Pattern.



The flat pattern displays the bend centerlines and the bend extent lines which indicate the bend zones required to flatten the lofted flange.

Note You can easily add a Rip feature to a lofted flange created from two closed profiles and generate a valid flat pattern. Your manufacturing shop may prefer to fabricate this type of part as two pieces.

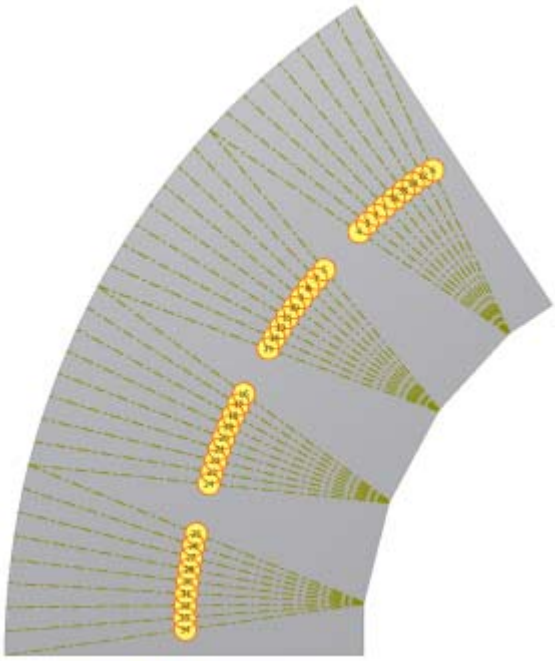
Using this flat pattern, you next explore **Bend Order Annotation**.

Previous | Next

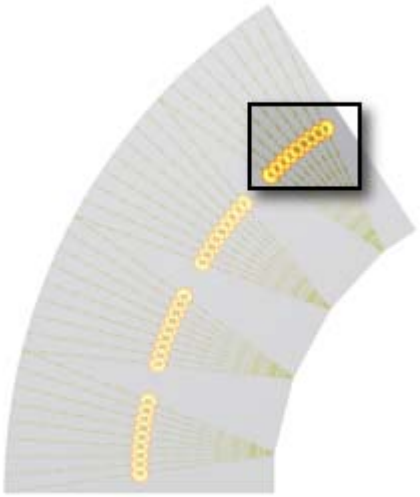
Bend Order Annotation

To accommodate efficient manufacturing, bends must often be created on the shop floor in a specific sequence. The manufacturing sequence has little in common with the design sequence. Using the flat pattern you currently have open, you can explore modifications to the bend order sequence.

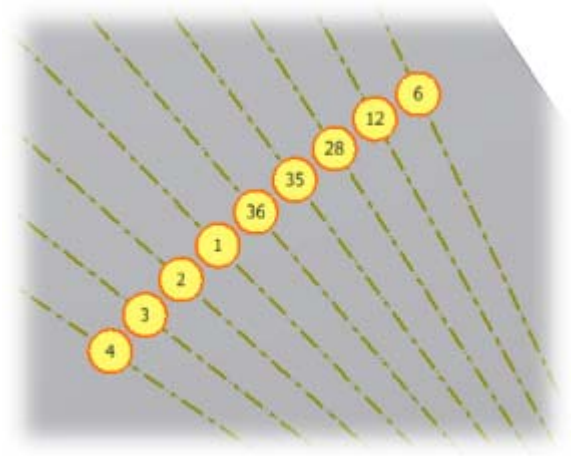
1. On the ribbon, click Flat Pattern tab ➤ Manage panel ➤ Bend Order Annotation.



Notice that a series of numbers appear within circles with a yellow background. As you proceed, focus on these numbered symbols in the upper portion of the flat pattern:



As you can see, these numbers do not currently have a logical sequence:



Next, you explore creating a directed sequence of bend overrides.

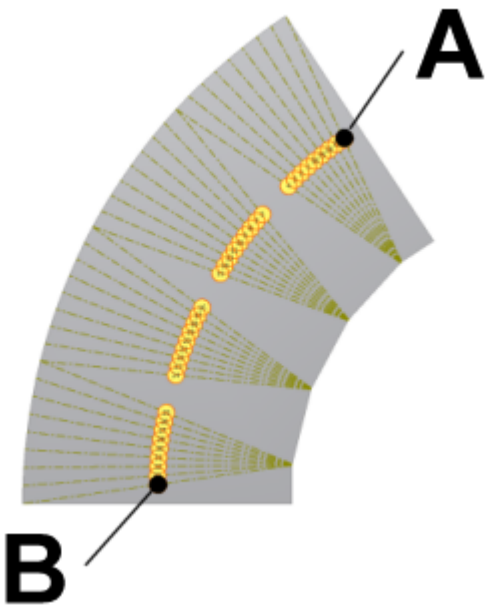
Note Do not be concerned if the order of the numbered symbols on your flat pattern differ from those shown in the images.

Previous | Next

Directed Reorder

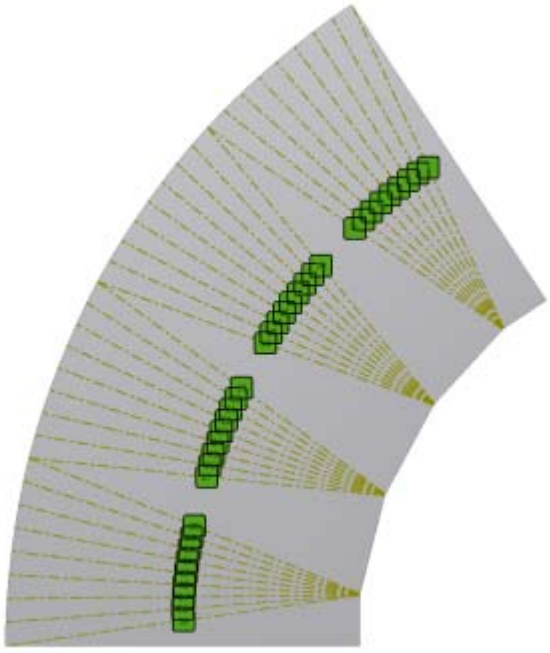
1. Right-click in the graphics window, and select **Directed Reorder** from the pop-up context menu.

   The Directed Reorder method of applying bend sequence overrides requires that you select a beginning bend and an ending bend. The system applies a new bend order sequence between the selected bends.

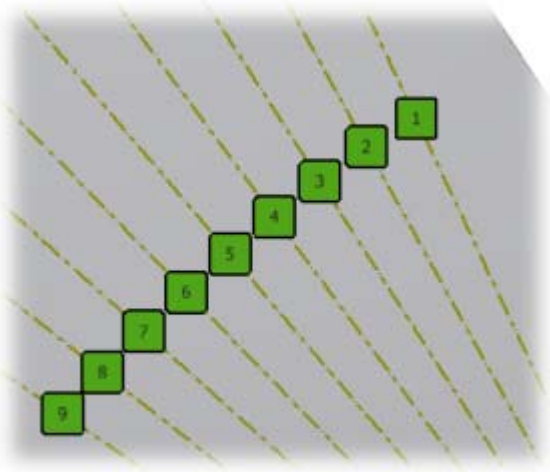


2. Click the upper-most bend symbol (labeled **A** in the previous image) to select the starting position of the override sequence.
3. Click the lower-most bend symbol (labeled **B**) to select the ending position. Notice that the

symbols all change from yellow circles to green squares.



Also notice that the numbering sequence has been changed. The bend you selected as the starting position is now numbered **1**. The remaining bends are numbered in sequence to the bend that you selected as the ending position.



Next, you explore creating a sequential reorder of bend identification overrides.

Previous | Next

Sequential Reorder

If you were happy with the sequence you obtained, you can right-click and select **Done**, then right-click again and select **Finish Bend Order**. Alternatively, pressing **Esc** twice is equivalent to selecting **Done** and **Finish Bend Order** from the pop-up context menu.

Since this is an exercise, you will not keep the directed sequence that you created.

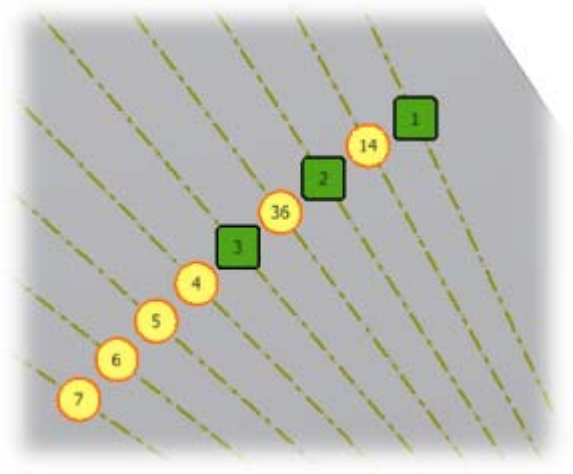
1. Right-click, and select **Remove All Overrides** from the pop-up context menu. Notice that the

green squares revert to yellow circles and the numbering sequence returns to the initial sequence generated by Autodesk Inventor.

2. Right-click again, and select **Sequential Reorder** from the pop-up context menu.

   Use sequential reordering to pick bends manually in the order that you need them to be manufactured.

3. Beginning again with the upper-most bend, click every other bend. As you click a bend, notice that the yellow circle again changes to a green square. The numbers change to correspond to the selected sequence. Click two or three more bends until you get the feel of this technique.



   As explained previously, when you are happy with the reordered sequence, press **Esc** twice, or use **Done** followed by **Finish Bend Order** from the context menu.

4. Since this is an exercise, right-click again and select **Remove All Overrides** from the context menu followed, by **Finish Bend Order** to exit the command.

Note In a third method of editing the bend order, you select a single, individual bend and change the bend order identification.

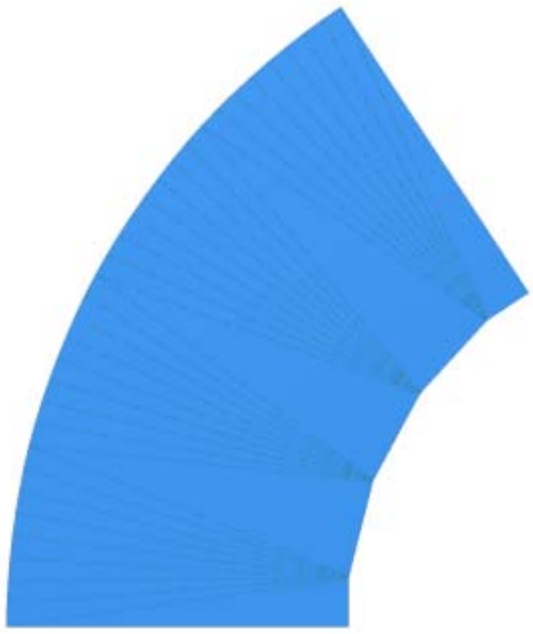
Next, you explore converting lines sketched on your flat pattern into cosmetic centerlines.

Previous | Next

Cosmetic Centerlines - Create Sketched Lines

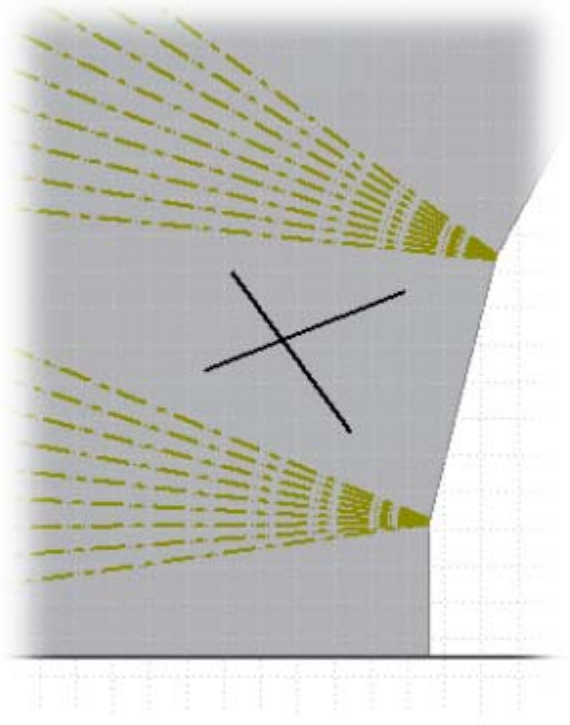
Cosmetic centerlines are straight lines sketched on a flat pattern that are converted to bend lines with bend extent lines. They represent bends that do not exist in your folded model, possibly stiffening creases or a bend line you want to place on a die-formed portion of your flat pattern. Cosmetic centerlines carry bend attributes that can be recovered in drawings. They can be sequenced using the Bend Order Annotation techniques explored in the previous exercise.

1. Click to select the face of the flat pattern.



Note If you did not clear the **Autoproject edges for sketch creation and edit** application option, as specified in the Prerequisites at the beginning of this tutorial, the sketch and all subsequent sketches made in this tutorial will have unneeded projected geometry.

2. On the ribbon, click Flat Pattern tab ➤ Sketch panel ➤ Create 2D Sketch.
3. Click **OK** in the dialog box that displays the message **Edits to the flat pattern are exclusively applied to the flat pattern and will not be reflected on the folded model.**
4. Sketch two straight lines as shown:



Note In this example, the size and position are not critical so dimensions will not be applied. In your designs, you will likely want to apply dimensions to position these lines accurately.

5. Click Sketch tab ➤ Exit panel ➤ Finish Sketch.

Next, you convert the sketched lines to cosmetic centerlines.

Previous | Next

Cosmetic Centerlines - Convert Sketched Lines

1. On the ribbon, click Flat Pattern tab ➤ Create panel ➤ Cosmetic Centerline.

   The Cosmetic Centerlines dialog box displays, and bend direction information appears on the existing bends in the flat pattern. The **Sketched Bend Lines** selection cursor is active.

2. Select the two straight lines that you added to the sketch.

   Notice that the bend direction for these two lines differs from the bend direction of the other bends. This may or may not suit your purposes. To minimize material handling during the creation of this example part, you change the bend direction.

3. Click **Specifies Bend Up or Bend Down** in the Cosmetic Centerlines dialog box to change the bend direction attribute.

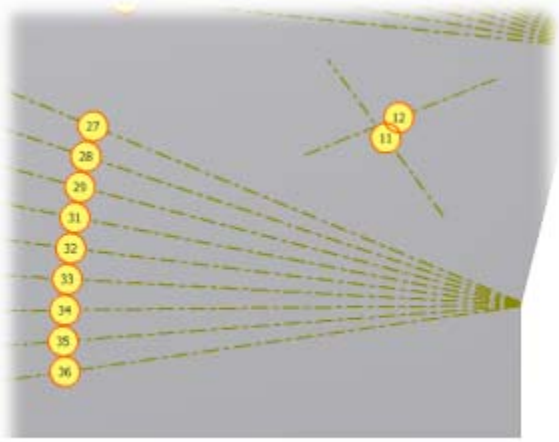
   

   Notice that the displayed bend attributes of the sketched lines now match the displayed bend attributes of the other bend lines on the flat pattern.

4. Change the **Bend Angle** value to 3 degrees in the Cosmetic Centerlines dialog box.

5. Click **OK** in the Cosmetic Centerlines dialog box to create the cosmetic centerlines using the specified attributes and close the dialog box.

Notice that the sketched lines now display using the **Bend Centerline** linetype. Using the techniques learned in the Bend Order Annotation exercise, click Flat Pattern tab ➤ Manage panel ➤ Bend Order Annotation. Notice that the cosmetic centerlines now participate in the bend order sequence.



These cosmetic centerlines can now be included in bend tables and bend notes you create in your drawings.

6. Click the **Esc** key to exit Bend Order Annotation.
7. On the ribbon, click Flat Pattern tab ➤ Folded Part panel ➤ Go to Folded Part to return to the folded model state.
8. You can **Save** your exercise file; however, the file is not used in further exercises.
9. Close the file that you have been using for these exercises.

Next, you will add Contour Roll features to a supplied sheet metal part.

Previous | Next

Contour Roll

Creating digital prototypes of roll formed sheet metal parts in Autodesk Inventor requires using a Contour Roll feature. The **Contour Roll** command creates a feature like a Contour Flange that uses a sketched profile as well as a sketched axis of revolution. The profile and axis geometry must exist within the same sketch.

In this exercise, you open a file that contains a straight Contour Flange feature. You add two Contour Roll features and a final Contour Flange. You then create a flat pattern of the resulting folded model.

1. Set your project to **tutorial_files**.
2. Open **Sheet Metal Parts 2** ➤ **contour_roll-start.ipt**.

3. On the ribbon, click Sheet Metal tab ▶ Sketch panel ▶ Create 2D Sketch.
4. When prompted to select a plane or sketch, select the top face of the Contour Flange feature as shown:

Note All sketch illustrations in this tutorial show the grid displayed. If you recently completed either the **Parts 1** or **Parts 2** tutorials, you have undisplayed the sketch grid by changing the Application Options. This tutorial does not require the use of the sketch grid and may be completed with the grid displayed or undisplayed.

Next, you project edges into the sketch.

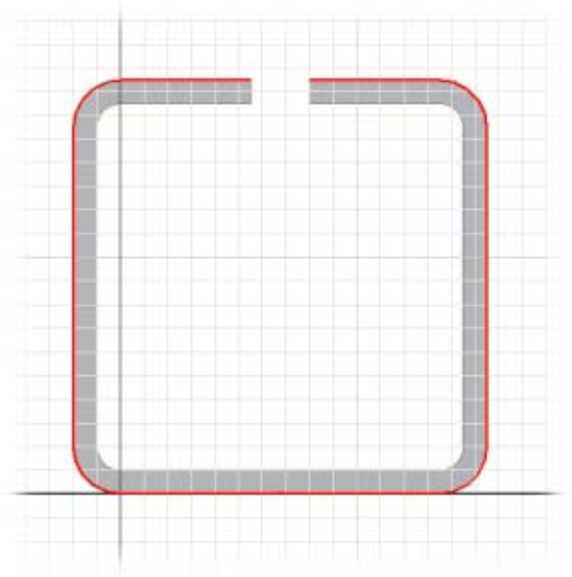
Previous | Next

Project Contour Roll Profile Geometry

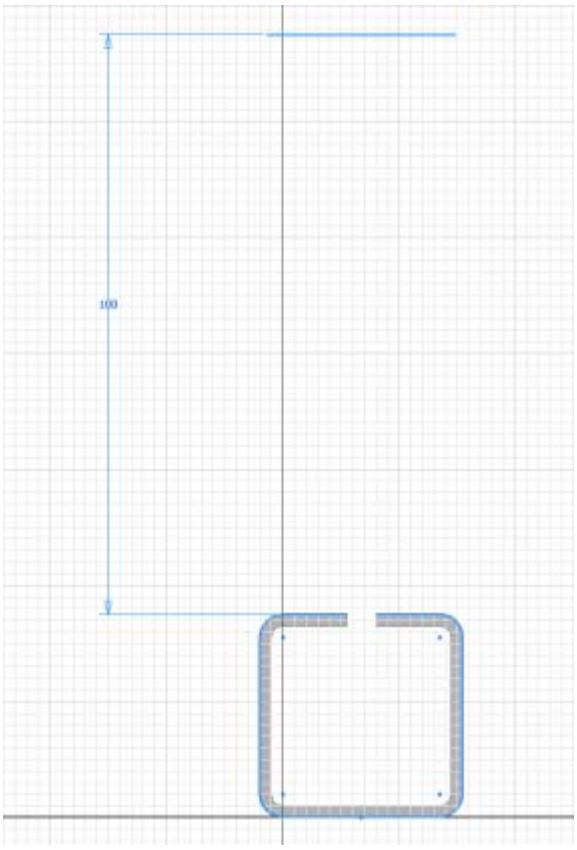
In this exercise, you project the edges of the existing Contour Flange and add a straight line as the axis of revolution. However, you can use any open profile consisting of lines, arcs, splines, and elliptical arcs to create a Contour Roll feature.

Note The Contour Roll feature will transform sharp sketch corners into bends in the finished part using the bend radius value. This behavior is like the Contour Flange feature and is not apparent in the following exercise.

1. Orient your sketch using the **View Cube** or **View Face** so that you are looking at the sketch plane.
2. On the ribbon, click Sketch tab ➤ Draw panel ➤ Project Geometry.
3. In the graphics window, click to select the lines and arcs that define the outside edge of the Contour Flange feature as shown:



   Note Be sure to select individual lines and arcs rather than the face loop of the detail faces.
4. Click Sketch tab ➤ Draw panel ➤ Line.
5. Create a line to represent your axis of revolution as shown:



Note The length of this line is not important; however, the line should be parallel to the short, horizontal line segments that you projected into your sketch. You can either imply the parallel constraint as you draw the line or add a parallel constraint after the line has been drawn.

6. Click Sketch tab ➤ Constrain panel ➤ Dimension, and place a **100-mm** dimension between the line you created and the short, horizontal line segment that you projected into your sketch.
7. Click Sketch tab ➤ Exit panel ➤ Finish Sketch.
8. Right-click, and select **Home View** from the context menu (or press **F6**) to reorient your view.

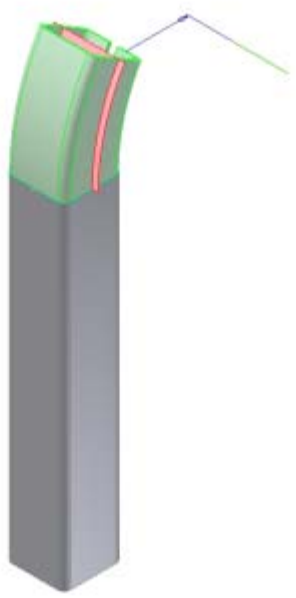
Previous | Next

Create a Contour Roll

1. On the ribbon, click Sheet Metal tab ➤ Create panel ➤ Contour Roll. The Contour Roll dialog box displays, you are prompted to select an open profile.
2. In the graphics window, click the edge geometry that you projected into your sketch. The selected geometry highlights, and the **Axis** selection becomes active.
3. In the graphics window, click the straight line that you created parallel to the projected edge and then offset with a 100-mm dimension.

   A 90 degree Contour Roll section previews; however, what is previewed is not what you want for this exercise.
4. Clear the default **Rolled Angle** value of **90 deg** degrees and enter 30 deg into the value field.

   Your preview should now appear as follows:



5. Click **OK** to create the 30-degree Contour Roll segment and close the dialog box.

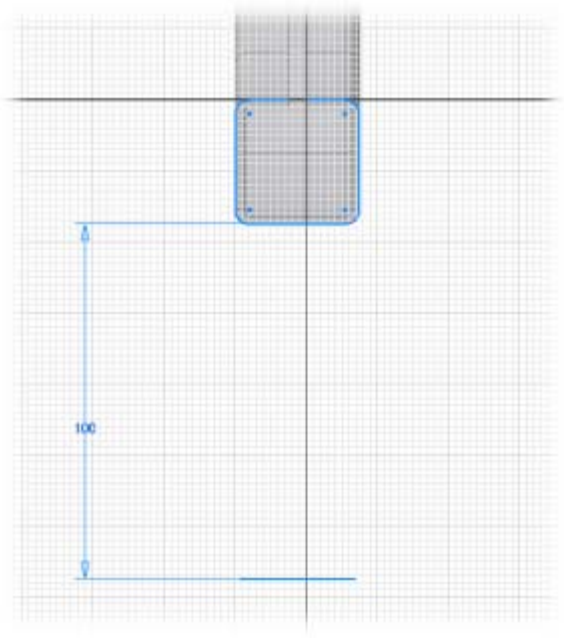
Next, you repeat these steps with a few minor differences to create a similar Contour Roll that sweeps 30 degrees in the opposite direction.

Previous | Next

Create a Second Contour Roll

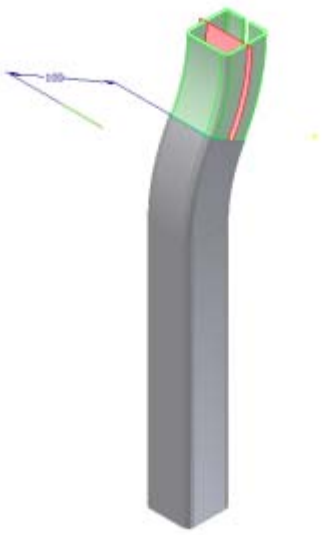
1. Repeat the steps you used to create the Contour Roll feature, using the end face of the edge of the first Contour Roll as the new sketch plane:

2. As you did previously, project the outside edges of the sheet metal material, and then add a straight line segment. This straight line segment should be on the side of (and parallel to) the long horizontal projection and offset by 100 mm. For the previous contour roll, the axis of revolution was on the side of the two short horizontal segments. By putting the axis on the opposite side of the profile, the revolution will curve in the opposite direction when you create this contour roll.



3. With your sketch completed, create the Contour Roll feature.

   Notice that the **Rolled Angle** value is 30 degrees, the last value you used in this command. It is the value you will use for this second contour roll. Your preview should appear as shown in the following image.



4. Click **OK** to create the second 30-degree contour roll segment and close the dialog box.

As a final modeling step, you will use the same sketch and project edges technique to create a sketch to use for a second contour flange.

Previous | Next

Add another Contour Flange

1. Repeat the steps you used to create the Contour Roll feature, using the end face of the edge of

the second Contour Roll as the new sketch plane.

2. As you did previously, project the outside edges of the sheet metal material.
3. With your sketch completed, create the Contour Flange feature.

   Expand the dialog box using **More** (**>>** in the lower right), and use the **Width Extents** type of **Distance** and a value of 200 mm.
4. Click **OK** to create the second 200-mm Contour Flange segment and close the dialog box.

   Your completed model should appear similar to the following image. Adjust your view of the model as needed.

Next, you create a flat pattern.

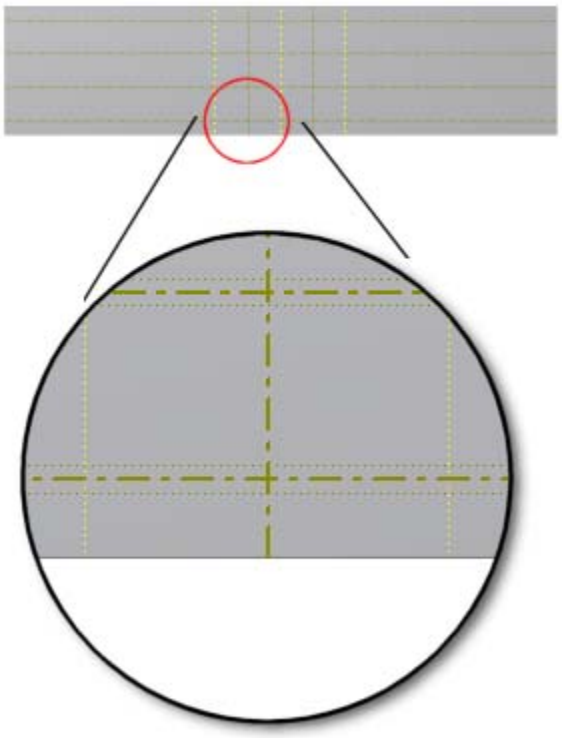
Previous | Next

Flatten the Rolled Tube

Because the Contour Flanges and Contour Roll features have formed an open tube, you can create a flat pattern without adding any additional features.

1. On the ribbon, click Sheet Metal tab ➤ Flat Pattern panel ➤ Create Flat Pattern.

The flat pattern is created. The **Flat Pattern** tab displays as the active ribbon tab.



The flat pattern displays the bend centerlines and bend extents for the four 90-degree bends that form the square tube, It also displays as the two roll centerlines for the two 30-degree rolls created by the Contour Roll features. Bend centerlines, bend extents, roll centerlines, and roll extents are all exported to separate layers when a flat pattern is exported to DWG or DXF formats to facilitate flexibility in CNC manufacturing.

In the final portion of this tutorial, you explore the use of the Unfold and Refold feature. You unroll and unfold the model that you created, adding some features and then refolding and rerolling the model.

Previous | Next

Unfold and Refold Feature Pair

There are some features that are easier to create when the model is flat. Use the Unfold feature to unfold (or unroll) all or some of the bends (or rolls) within your model. With the model unfolded, you can then add features and use Refold features to return the model to the folded state.

In this portion of the tutorial, you unroll the two Contour Roll features and unfold two of the four bends that form the square tube. You add a hole which you will pattern down the length of the part. To complete the exercise, you add two Refold features to refold and reroll the model. The completed model will appear as shown in the following image.

1. On the ribbon, click Flat Pattern tab ➤ Folded Part panel ➤ Go to Folded Part to return to the folded model.
   Note Alternatively, you can also double-click the **Folded Model** node in the Model browser to return to the folded model state.
2. Right-click, and select **Home View** from the pop-up context menu (or press **F6**).
3. Click Sheet Metal tab ➤ Modify panel ➤ Unfold. The Unfold dialog box displays, and two stationary reference planes appear at either end of the Contour Roll features.

4. In this exercise, we first unroll the Contour Roll features. Click in the graphics window to select the lower stationary reference plane.

Once you select a stationary reference, the rolls that can be unrolled relative to that reference are highlighted.

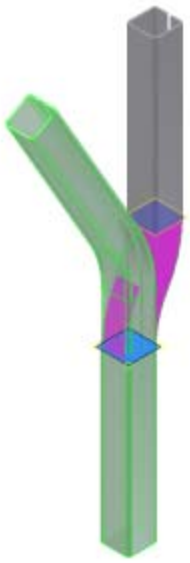
Next, you continue with additional Unfold selection steps.

Previous | Next

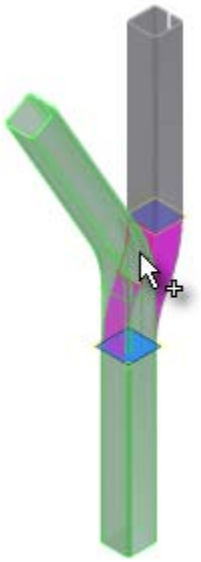
Continue Unfold Selection

1. Click to select the lower curved face as shown in the following image.

As you select faces, the preview shows the model state that results by unrolling the selection.



2. In this exercise, we straighten the tube completely. Click the upper curved face (not the preview graphic) to select the second rolled face.



Once the second rolled face is selected, the model again previews the unroll results.

3. Click **Apply** in the Unfold dialog box to straighten the model as shown in the preview and to reset the dialog box for the next round of unfolds.

Next, you unfold two of the 90 degree bends which form the square tube.

Previous | Next

Partially Unfold the Tube

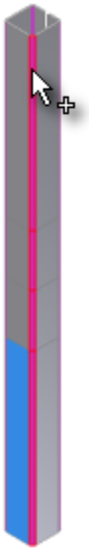
Use the Unfold feature to pick the bends that you want to flatten. It is not necessary to flatten the model completely. It is possible to add the linear hole pattern. For this exercise, with the model unrolled in its current state, you will first unfold an additional two 90-degree bends.

1. As with the unrolled model that you created, you first identify a stationary face. Click the face shown in the following image.

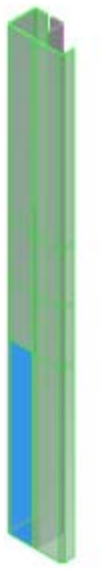
As soon as you select the face shown, the bends that can be unfolded relative to that face

highlight:

2. Click to select the bend which forms the 90-degree corner closest to you:



Once the bend is selected, the part previews in the unfolded state.

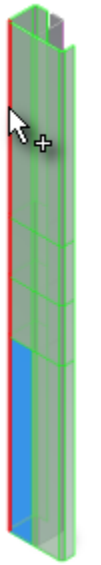


Next, you finish creating the Unfold feature.

Previous | Next

Complete the Unfold Feature

1. Click to select the second 90-degree bend as shown in the following image.



Following the select, the unfolded bend is previewed:

2. Click **OK** in the Unfold dialog box to flatten the two bends as shown in the preview and to close the dialog box. Your model should now appear as shown in the following image.

Although these steps are not required to add the holes (that you will add next) they illustrate adding an Unfold feature to flatten straight bends.

Notice that your feature browser now contains two Unfold features: one for the unfolding of the two contour rolls and one for the unfolding of the two straight bends.

Next, you add a hole and pattern the hole so that it crosses the (now flat) faces of the Contour Roll features.

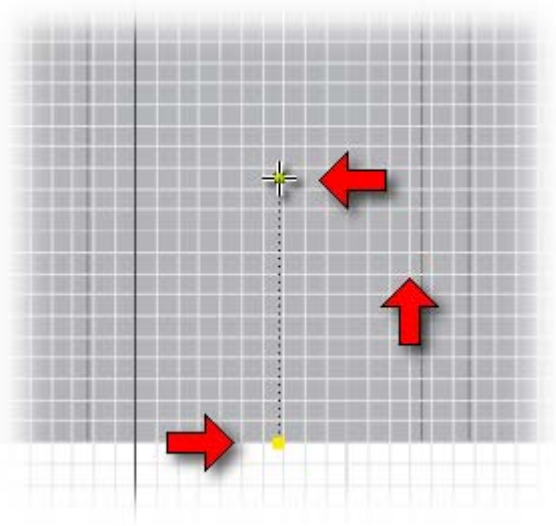
Previous | Next

Add a Hole

1. On the ribbon, click Sheet Metal tab ➤ Sketch panel ➤ Create 2D Sketch, and then select the face shown in the following image.

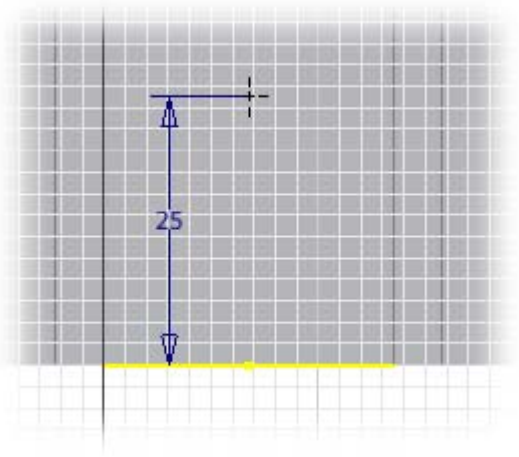
2. Reorient your view, using the **View Cube** or **View Face**, so that you are looking directly at the sketch.

3. Click Sketch tab ➤ Draw panel ➤ Point. Drag over the projected origin point, and then up. You should see the dotted line which indicates that the point you will place is aligned with the origin point.



4. Click to place the point.

5. Click Sketch tab ➤ Constrain panel ➤ Dimension, and place a **25-mm** dimension between the point and the bottom edge of the unfolded part.



6. Click **Finish Sketch** to exit the sketch environment.
7. Click Sheet Metal tab ➤ Modify panel ➤ Hole, and place a **5-mm** diameter hole with a **Through All** termination on the sketched point.

Next, you pattern the hole.

Previous | Next

Pattern the Hole

1. On the ribbon, click Sheet Metal tab ➤ Pattern panel ➤ Rectangular.
2. Select the hole as the feature to pattern.
3. Click the **Direction 1** button and select a vertical edge and direction as shown:

4. Enter a value of 20 for the **total number of holes**, and a value of 25 mm for the **distance between each hole**.
5. Click **OK** to create the pattern of holes that cross the two flattened roll faces and close the Rectangular Pattern dialog box.

Next, you add two Refold features to return the flattened model to the folded and rolled state.

Previous | Next

Add Two Refold Features

The Refold feature is the complement of the Unfold feature. You are not able to place a Refold feature unless there is an Unfold feature in the model. When there are more than one Unfold features in the model, you must refold them in reverse order. You must refold the most recently created Unfold feature first.

Although there are several Refold workflows, in this exercise you use the most common workflow. You right-click to select the Unfold feature and select Refold Feature in the pop-up context menu. This method automatically selects the originally selected stationary face and which ever bends or rolls were originally selected to create the Unfold feature. Other Refold workflows provide more flexibility and allow partial refolding when that makes sense for your design situation.

1. Right-click the **Unfold** feature in the Model browser that is immediately above the **Hole** feature. Select **Refold Feature** in the pop-up context menu.

   Notice that the two 90-degree straight bends refold, and that a Refold feature is added to the list of features in the Model browser.

2. Right-click the first **Unfold** feature (between the **Contour Flange** feature and the second

Unfold feature), and select **Refold Feature**.

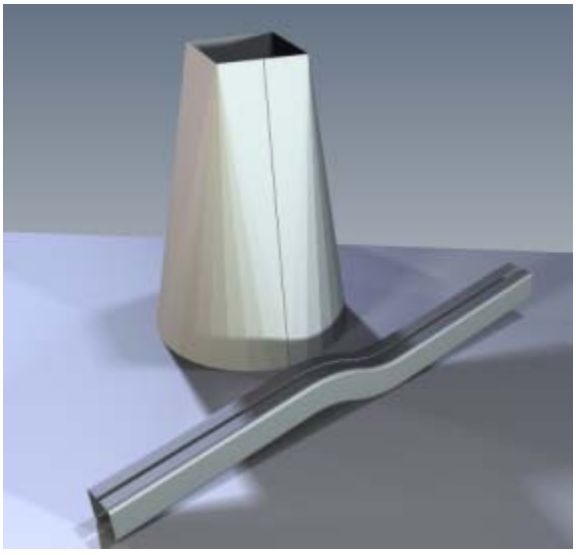
Notice that the two 30-degree Contour Roll features reroll and that a second Refold feature is added to the list of features in the Model browser.

The completed model will appear as shown in the following image.

This completes the exercises of this tutorial.

Previous | Next

Summary



The features you explored in this tutorial represent powerful additions to your sheet metal modeling skills.

- Transitional shapes defined by selecting two profiles for a Lofted Flange feature are common in some sheet metal design situations.
- The ability to define the output of a Lofted Flange targeted at either a Press Brake or Die Form manufacturing process provides flexibility in both design and manufacturing.
- The ease of adding a Rip feature to a Lofted Flange created from two closed profiles provides for

ease of flat pattern creation during the design process.

- Bend Order Annotation on the sheet metal flat pattern allows documentation of the correct fabrication sequence.
- The ability to add cosmetic centerlines provides additional efficiencies by allowing the documentation of bends that have not been created in the folded model state.
- Certain features are easier to create while the model is flat: Unfold and Refold features allow efficient creation of these features while showing them correctly in both the final folded model and the flat pattern.

What Next? As a next step, consider exploring the creation of Lofted Flange features with two open profiles. Or, create a variation of the folded and rolled square tube with cut features that cross over both the 90-degree square corner bend as well as the 30-degree rolled faces. You can also explore the capabilities of the Inventor Studio environment which was used to create several of the photo realistic images that were used in this tutorial.

Previous