

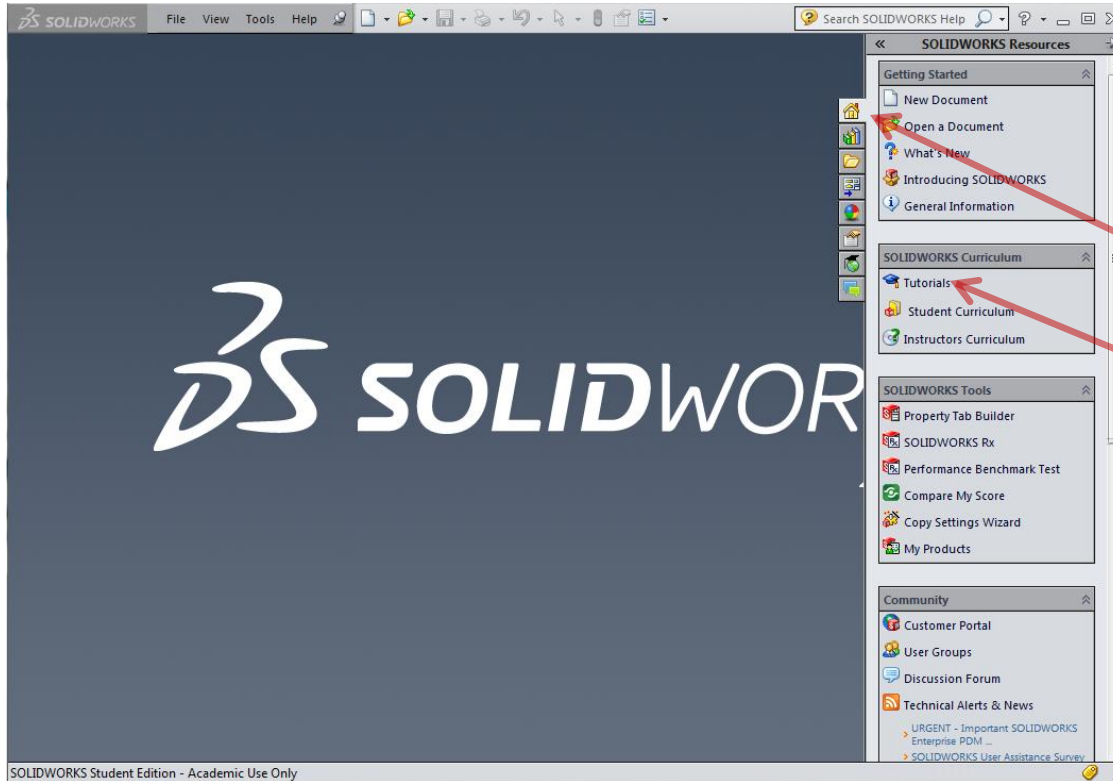
# SolidWorks Part 4

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<http://saliterman.umn.edu/>

# Starting The Tutorials



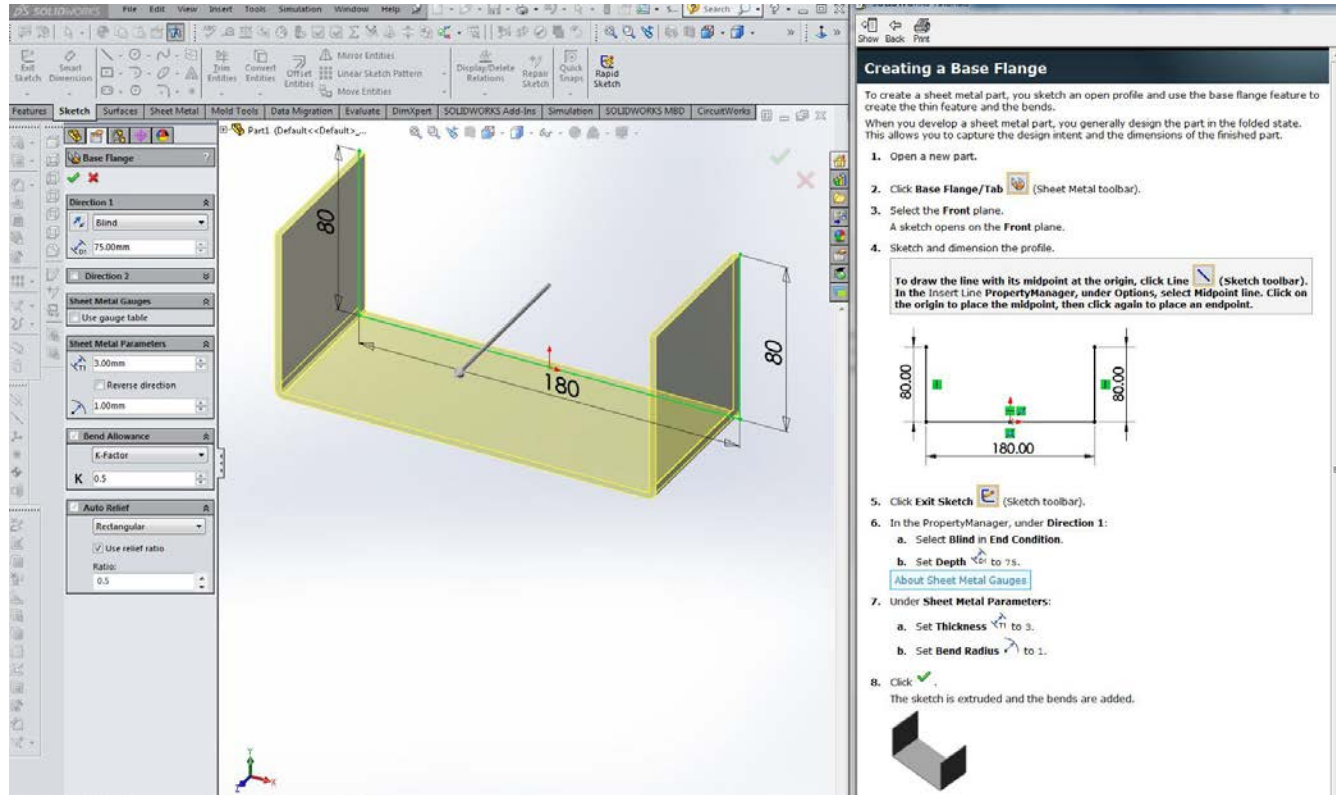
- Launch SolidWorks
- Select Resources to open the Task Pane.
- Select Tutorials

# Select Basic Techniques



Sheet Metal

# Creating the Base-Flange



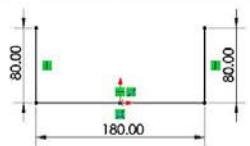
**Creating a Base Flange**

To create a sheet metal part, you sketch an open profile and use the base flange feature to create the thin feature and the bends.


When you develop a sheet metal part, you generally design the part in the folded state. This allows you to capture the design intent and the dimensions of the finished part.

1. Open a new part.
2. Click **Base Flange/Tab** (Sheet Metal toolbar).
3. Select the **Front** plane.  
A sketch opens on the **Front** plane.
4. Sketch and dimension the profile.

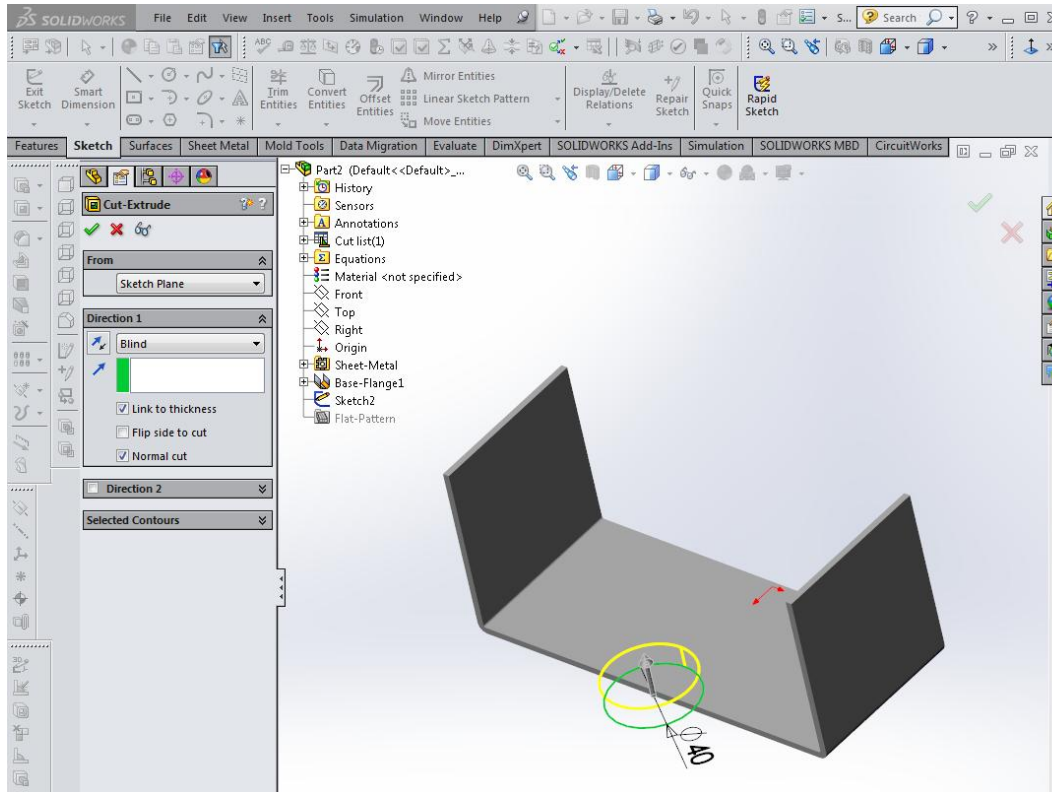
To draw the line with its midpoint at the origin, click **Line** (Sketch toolbar). In the **Insert Line PropertyManager**, under **Options**, select **Midpoint** line. Click on the **origin** to place the midpoint, then click again to place an endpoint.



5. Click **Exit Sketch** (Sketch toolbar).
6. In the **PropertyManager**, under **Direction 1**:
  - a. Select **Blind** in **End Condition**.
  - b. Set **Depth** to 75.[About Sheet Metal Gauges](#)
7. Under **Sheet Metal Parameters**:
  - a. Set **Thickness** to 3.
  - b. Set **Bend Radius** to 1.
8. Click **✓**.  
The sketch is extruded and the bends are added.



# Making the Extruded-Cut



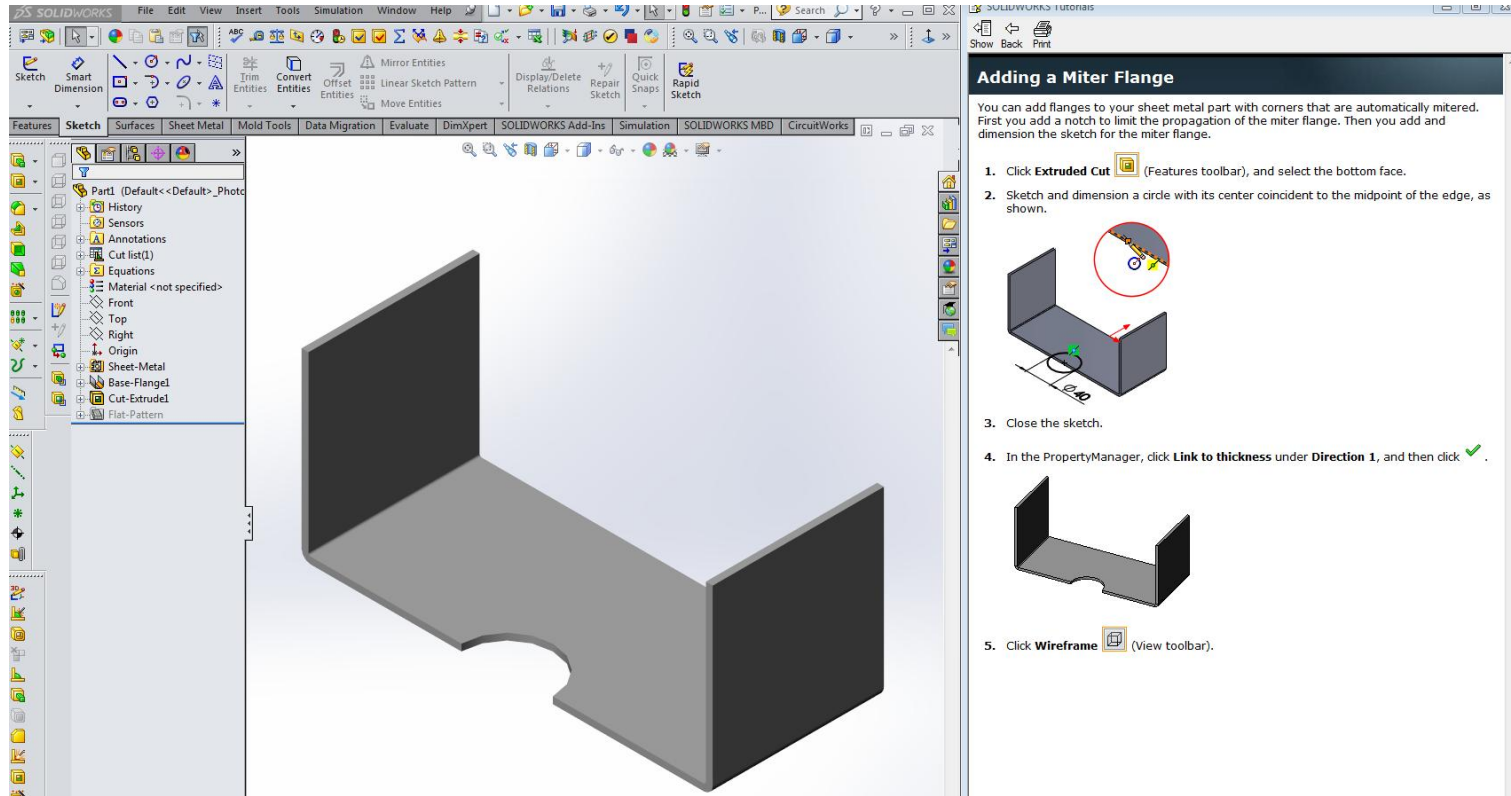
## SOLIDWORKS Tutorials

### Adding a Miter Flange

You can add flanges to your sheet metal part with corners that are automatically mitered. First you add a notch to limit the propagation of the miter flange. Then you add and dimension the sketch for the miter flange.


1. Click **Extruded Cut** (Features toolbar), and select the bottom face.
2. Sketch and dimension a circle with its center coincident to the midpoint of the edge, as shown.
3. Close the sketch.
4. In the PropertyManager, click **Link to thickness** under **Direction 1**, and then click
5. Click **Wireframe** (View toolbar).

# The Extruded Cut



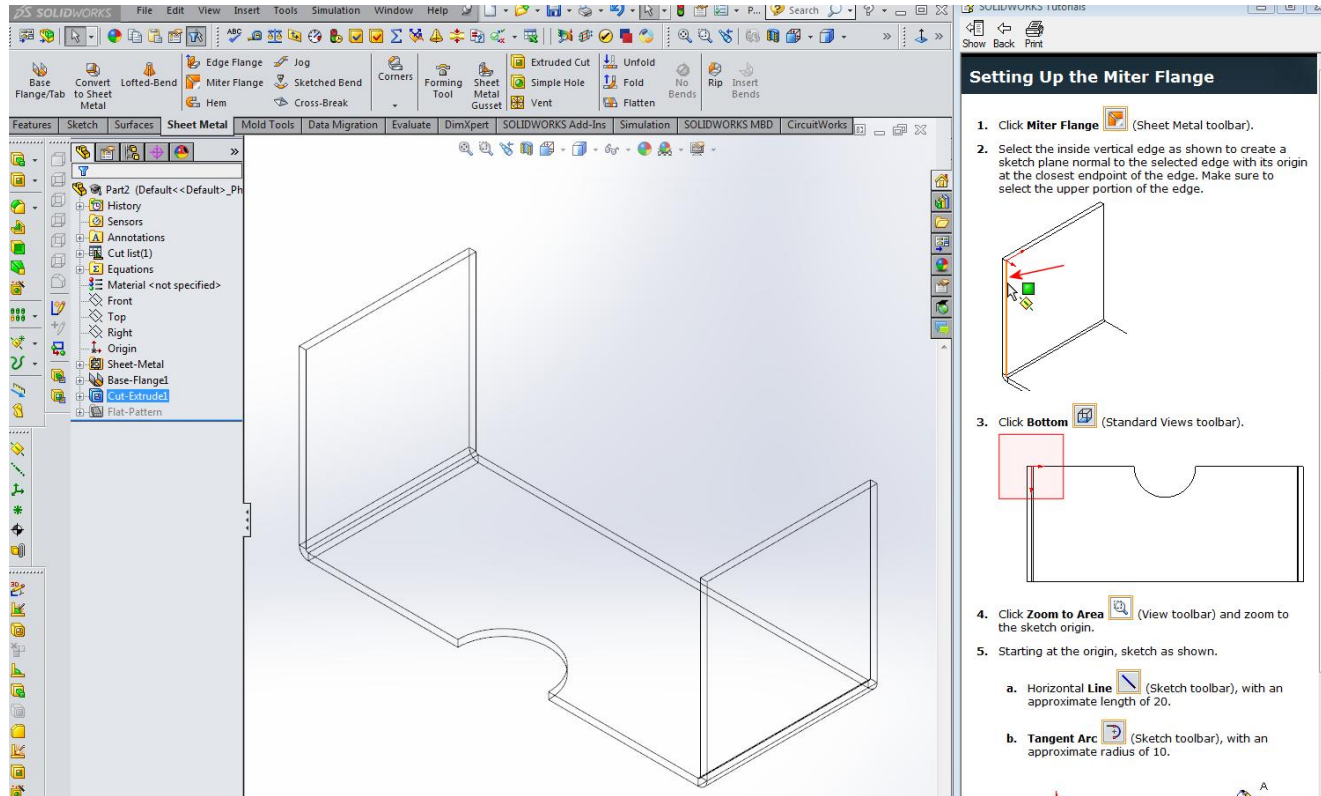
**Adding a Miter Flange**

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5. Click **Wireframe** (View toolbar).



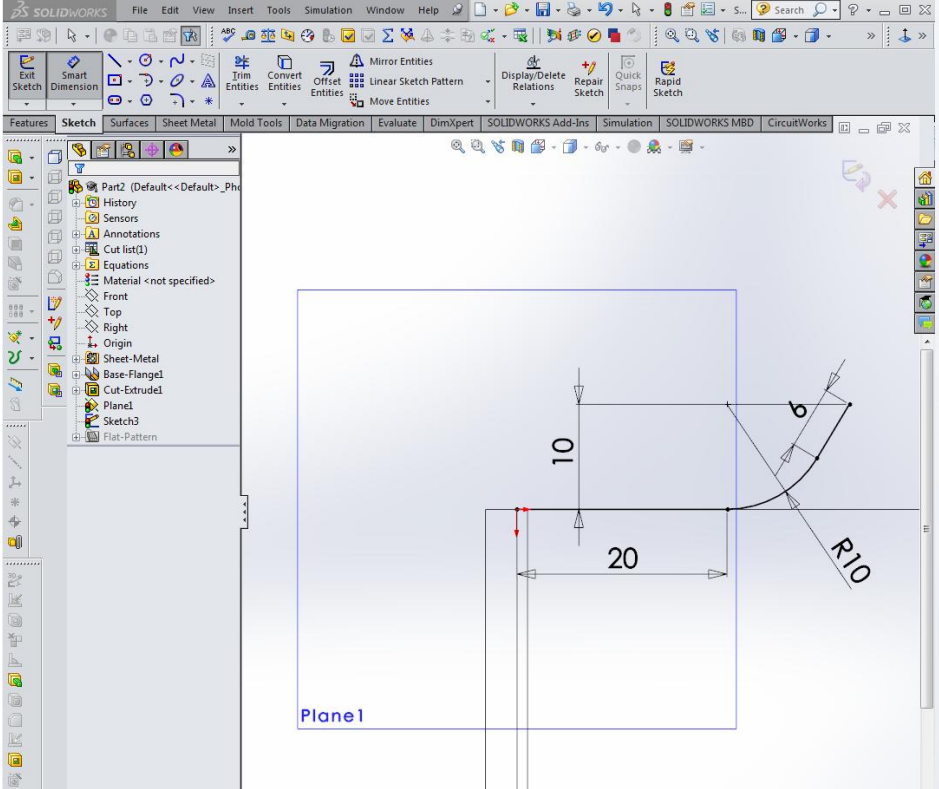
# Wire-Frame View



The image shows a SolidWorks interface with a wire-frame model of a miter flange. The software's ribbon is set to the 'Sheet Metal' tab, and the 'Miter Flange' tool is selected. The left-hand 'Feature Tree' shows the model's structure, including 'Part2 (Default << Default >> Ph...', 'History', 'Sensors', 'Annotations', 'Cut list(1)', 'Equations', 'Material <not specified>', 'Front', 'Top', 'Right', 'Origin', 'Sheet-Metal', 'Base-Flange1', 'Cut-Extruded1', and 'Flat-Pattern'. The main 3D view displays the wire-frame of the miter flange, which has a semi-circular cutout on one side. To the right, a tutorial window titled 'Setting Up the Miter Flange' provides a five-step guide:

1. Click **Miter Flange** (Sheet Metal toolbar).
2. Select the inside vertical edge as shown to create a sketch plane normal to the selected edge with its origin at the closest endpoint of the edge. Make sure to select the upper portion of the edge.
3. Click **Bottom** (Standard Views toolbar).
4. Click **Zoom to Area** (View toolbar) and zoom to the sketch origin.
5. Starting at the origin, sketch as shown.
  - a. **Horizontal Line** (Sketch toolbar), with an approximate length of 20.
  - b. **Tangent Arc** (Sketch toolbar), with an approximate radius of 10.

# Making a Path for the Miter

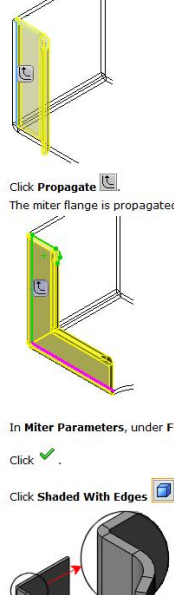


SOLIDWORKS Tutorials

## Completing the Miter Flange

Now you create the miter flange.

1. Click **Isometric** (Standard Views toolbar).
2. Click **Exit Sketch** (Sketch toolbar).
3. Click **Propagate**.  
The miter flange is propagated to the tangent edges, stopping at the notch.
4. In **Miter Parameters**, under **Flange position**, click **Material Outside**.
5. Click **✓**.
6. Click **Shaded With Edges** (View toolbar).





# Miter

**SOLIDWORKS** File Edit View Insert Tools Simulation Window Help

Part2 (Default <<Default>\_...

**Miter Flange**

**Miter Parameters**

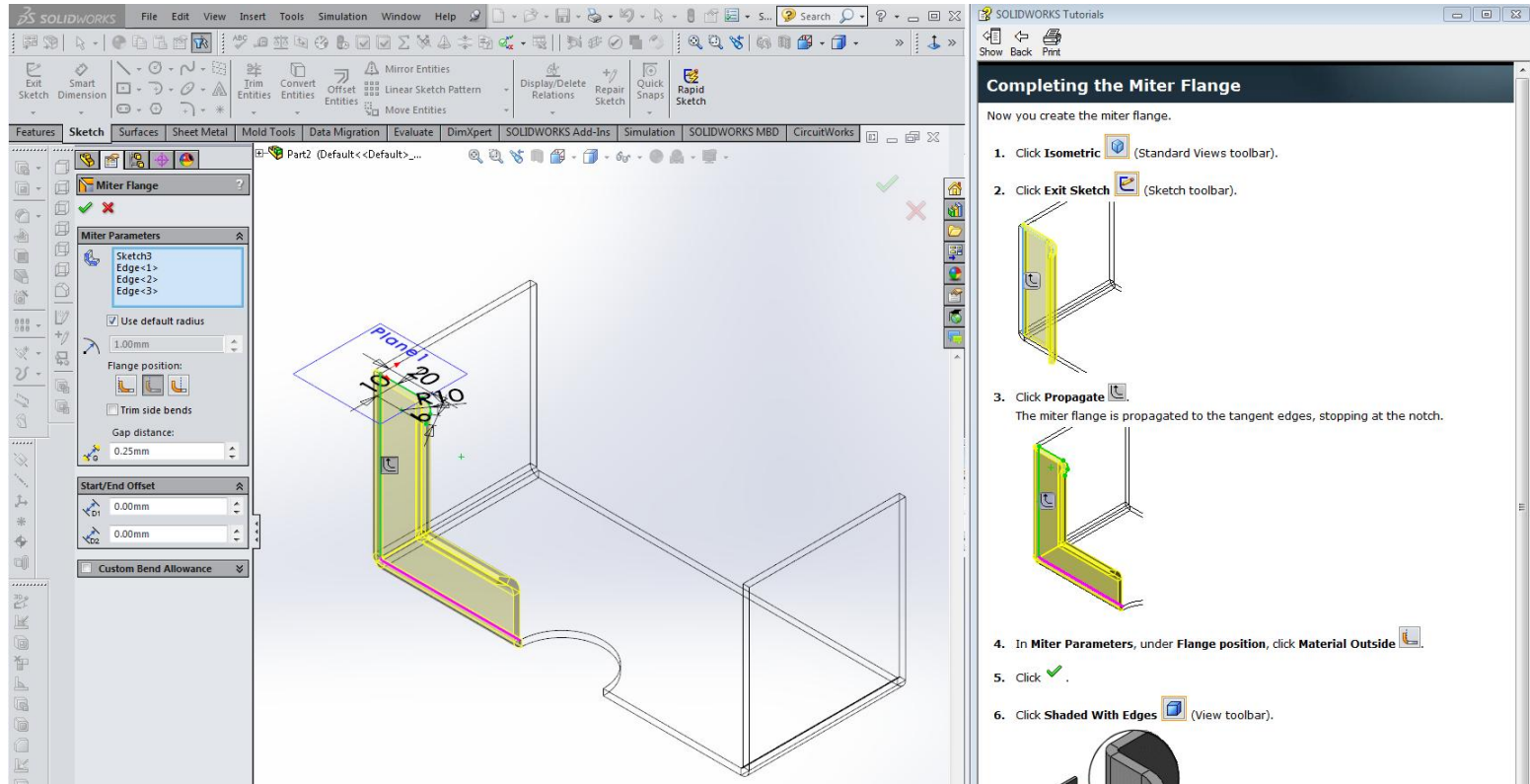
- Sketch3 Edge-1>
- Use default radius
- 1.00mm
- Flange position:
  - Trim side bends
- Gap distance:
  - 0.25mm
- Start/End Offset
  - 0.00mm
  - 0.00mm
- Custom Bend Allowance

**Completing the Miter Flange**

Now you create the miter flange.

1. Click **Isometric** (Standard Views toolbar).
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3. Click **Propagate**.  
The miter flange is propagated to the tangent edges, stopping at the notch.
4. In **Miter Parameters**, under **Flange position**, click **Material Outside**.
5. Click .
6. Click **Shaded With Edges** (View toolbar).

# Propagation of Miter



**SOLIDWORKS** File Edit View Insert Tools Simulation Window Help

Sketch Surfaces Sheet Metal Mold Tools Data Migration Evaluate DimXpert SOLIDWORKS Add-Ins Simulation SOLIDWORKS MBD CircuitWorks

Part2 (Default <<Default>\_...

**Miter Flange**

**Miter Parameters**

Sketch3  
Edge <1>  
Edge <2>  
Edge <3>

Use default radius  
1.00mm

Flange position:  
 Trim side bends

Gap distance:  
0.25mm

Start/End Offset  
0.00mm  
0.00mm

Custom Bend Allowance

**SOLIDWORKS Tutorials**

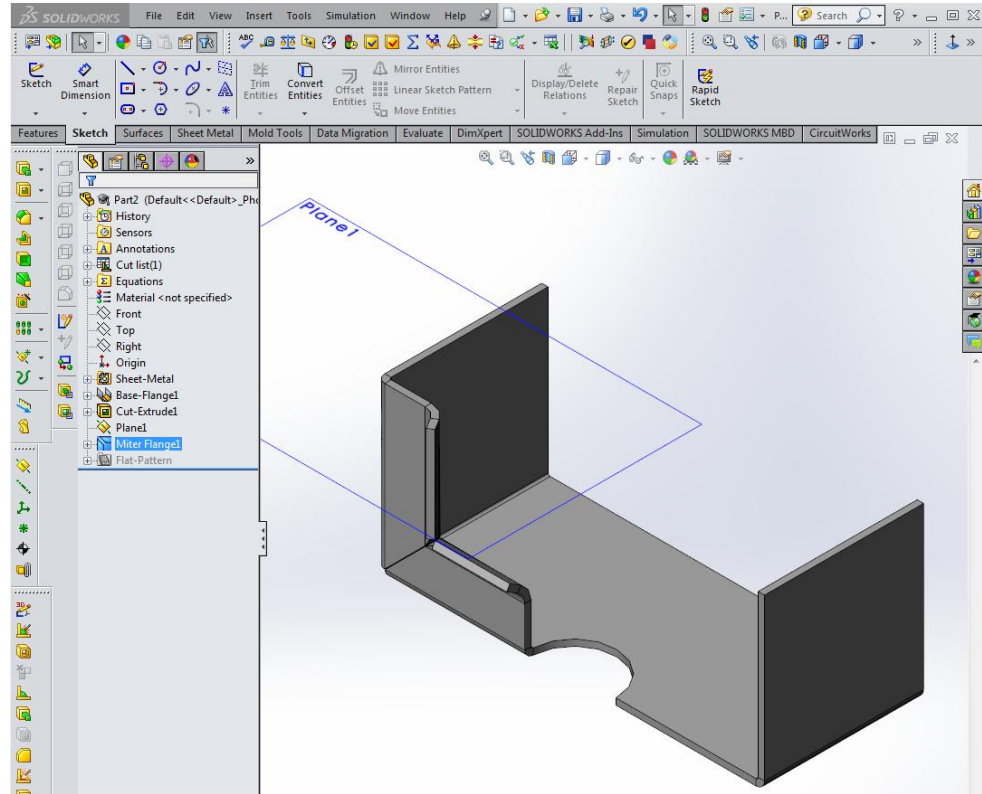
Show Back Print

### Completing the Miter Flange

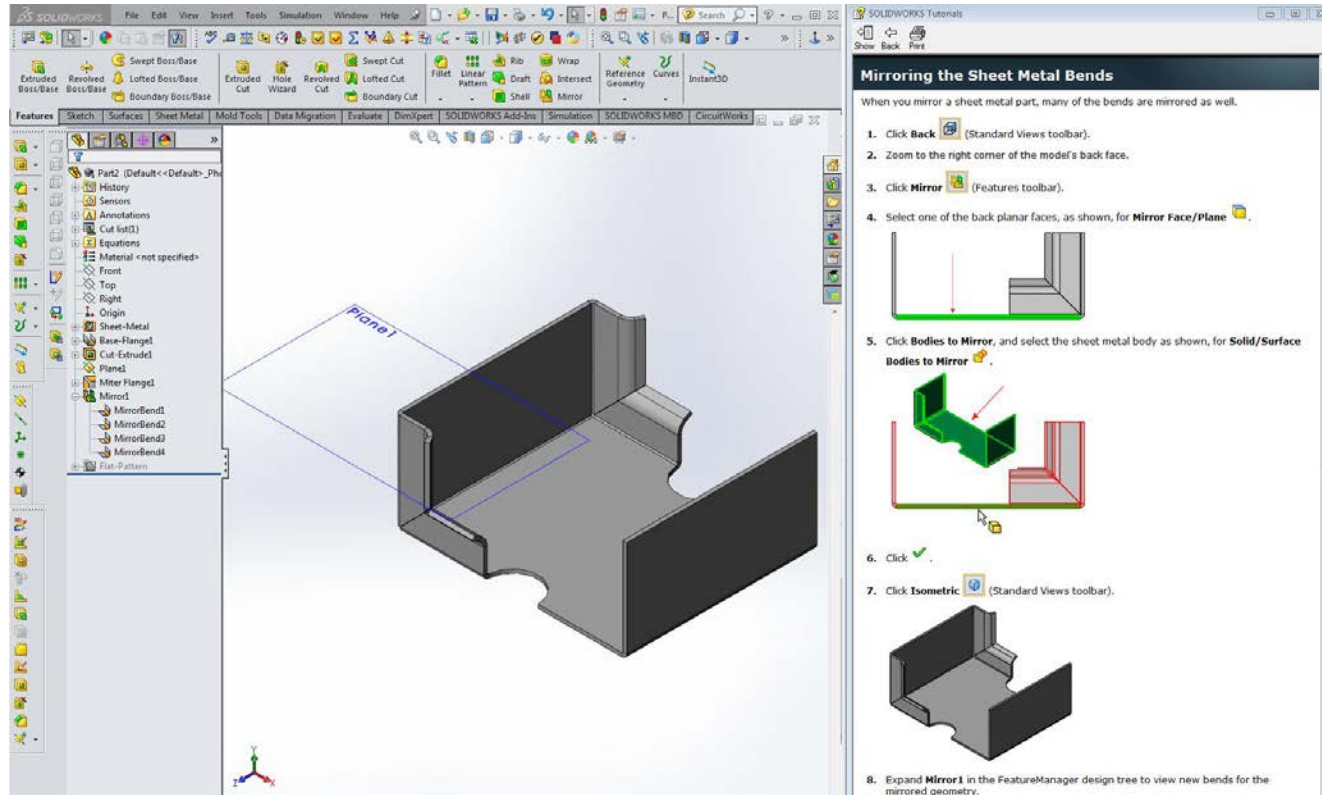
Now you create the miter flange.

1. Click **Isometric** (Standard Views toolbar).
2. Click **Exit Sketch** (Sketch toolbar).
3. Click **Propagate**  
The miter flange is propagated to the tangent edges, stopping at the notch.
4. In **Miter Parameters**, under **Flange position**, click **Material Outside**.
5. Click .
6. Click **Shaded With Edges** (View toolbar).

# Miter Shown with Edges



# Mirrored Geometry

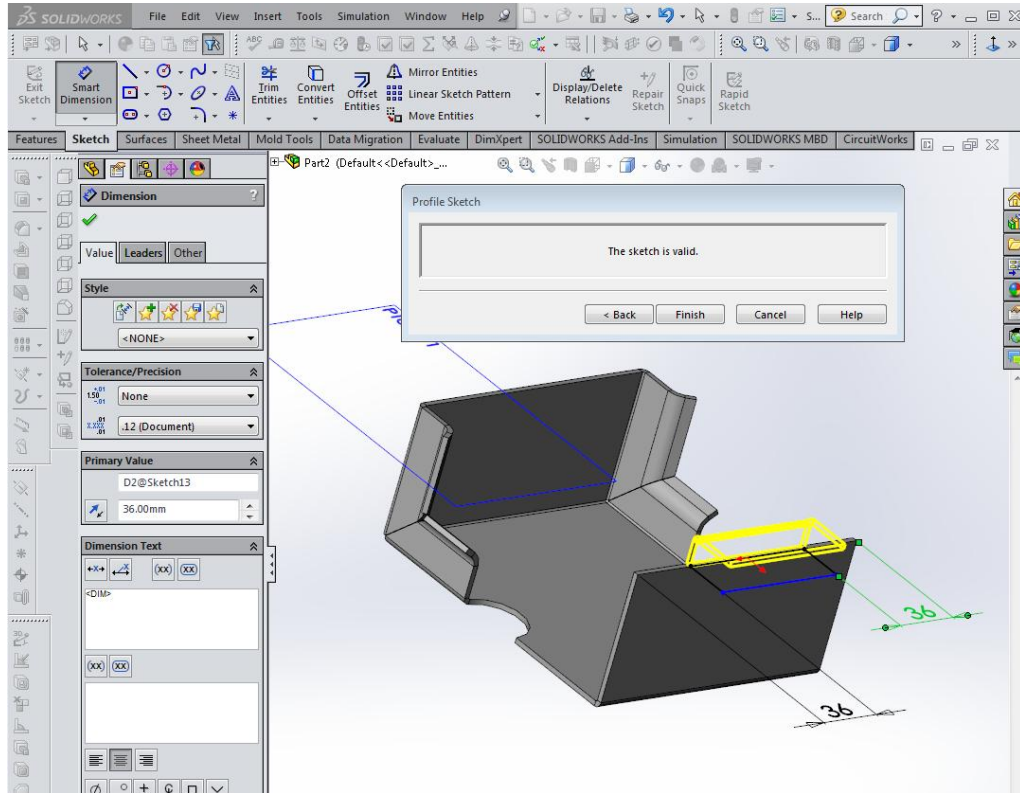


The image displays a SolidWorks software interface. The main window shows a 3D model of a sheet metal part with a mirrored section. A blue dashed box labeled "Plane1" indicates the mirror plane. The FeatureManager design tree on the left lists features including "Mirror1", "MirrorBend1", "MirrorBend2", "MirrorBend3", "MirrorBend4", and "Flat-Pattern".

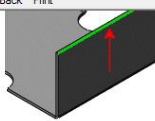
On the right, a tutorial window titled "Mirroring the Sheet Metal Bends" provides a step-by-step guide:

1. Click **Back** (Standard Views toolbar).
2. Zoom to the right corner of the model's back face.
3. Click **Mirror** (Features toolbar).
4. Select one of the back planar faces, as shown, for **Mirror Face/Plane**.
5. Click **Bodies to Mirror**, and select the sheet metal body as shown, for **Solid/Surface Bodies to Mirror**.
6. Click **✓**.
7. Click **Isometric** (Standard Views toolbar).
8. Expand **Mirror1** in the FeatureManager design tree to view new bends for the mirrored geometry.

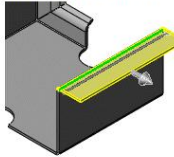
# Creating Edge Flange



SOLIDWORKS Tutorials



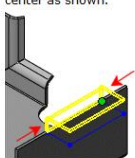
3. Drag the edge flange as shown, and click.



4. In the PropertyManager:

- a. Under **Flange Length**, set **Length** to 30.
- b. Under **Flange Position** select **Material Outside**.
- c. Under **Flange Position** select **Offset**.
- d. Under **Flange Position** set **Offset Distance** to 15.
- e. Under **Flange Parameters**, click **Edit Flange Profile** to display the **Profile Sketch** dialog box.

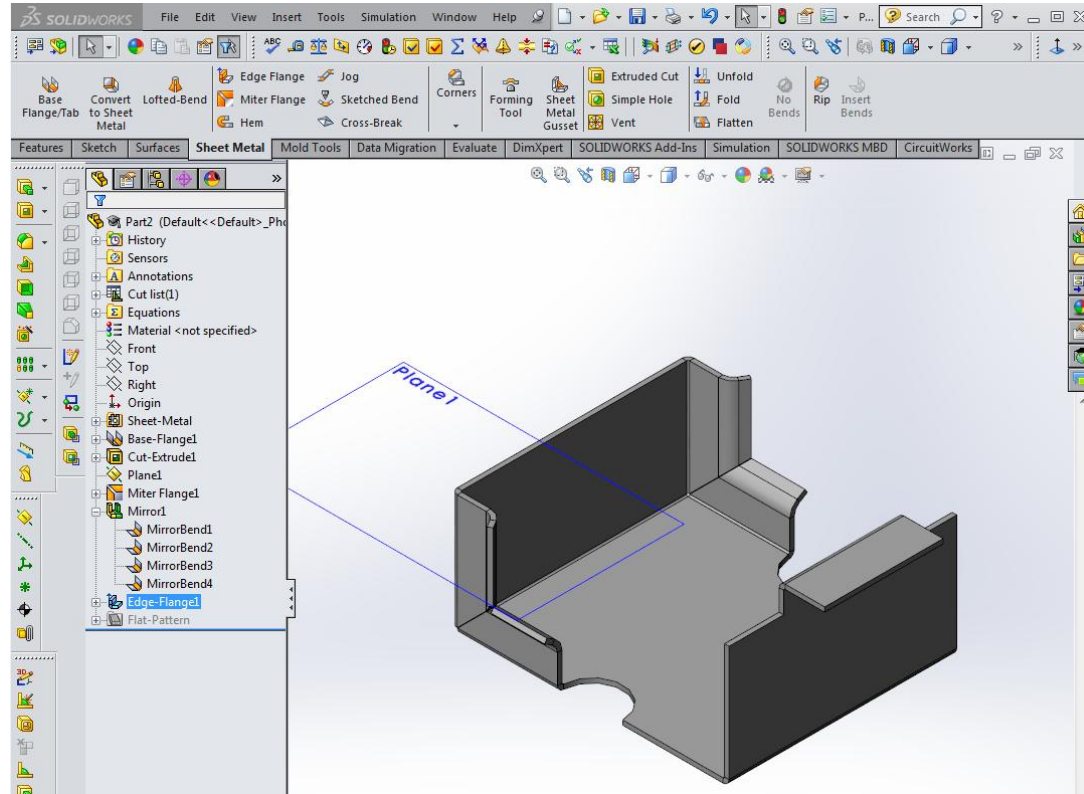
5. Select the end points along the base flange (inside edge) and drag them towards the center as shown.



Repositioning the end points removes the relation between the width of the base flange and the width of the edge flange.

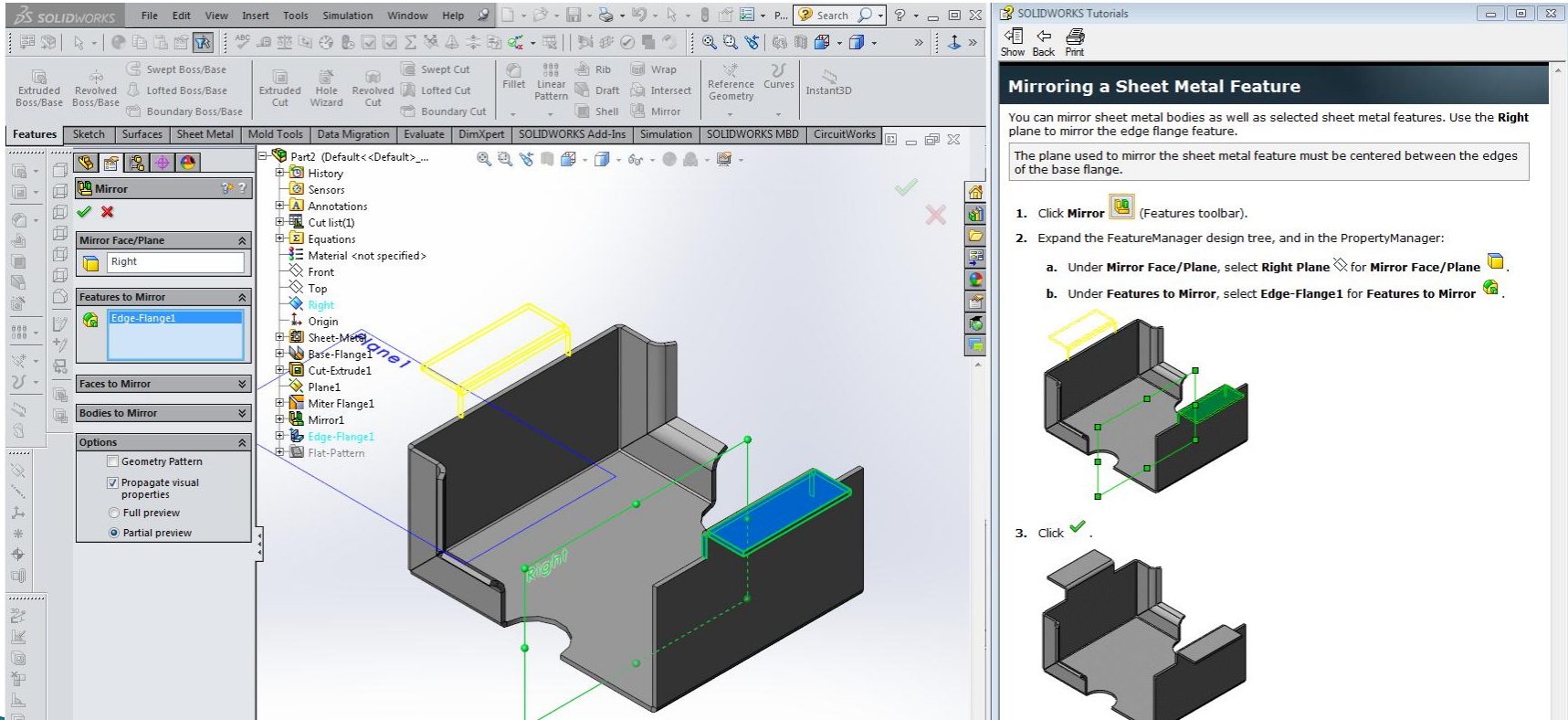


# Finished Edge Flange





# Mirroring a Sheet Metal Feature



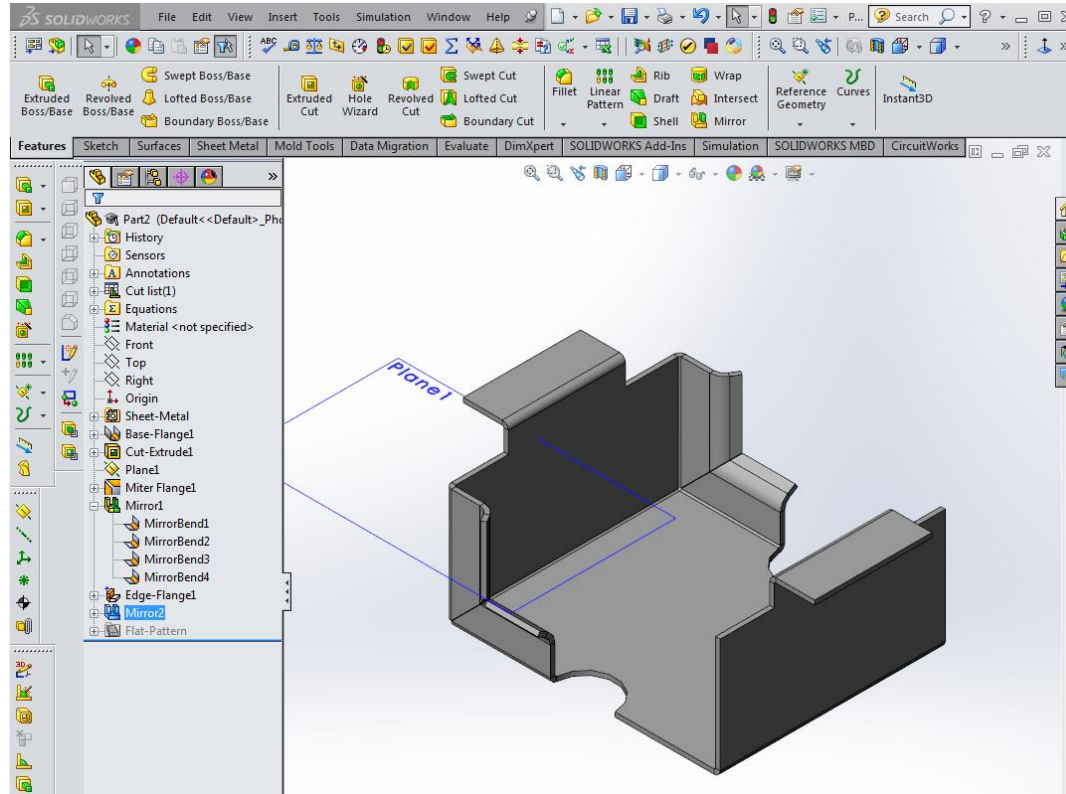
**Mirroring a Sheet Metal Feature**

You can mirror sheet metal bodies as well as selected sheet metal features. Use the **Right** plane to mirror the edge flange feature.

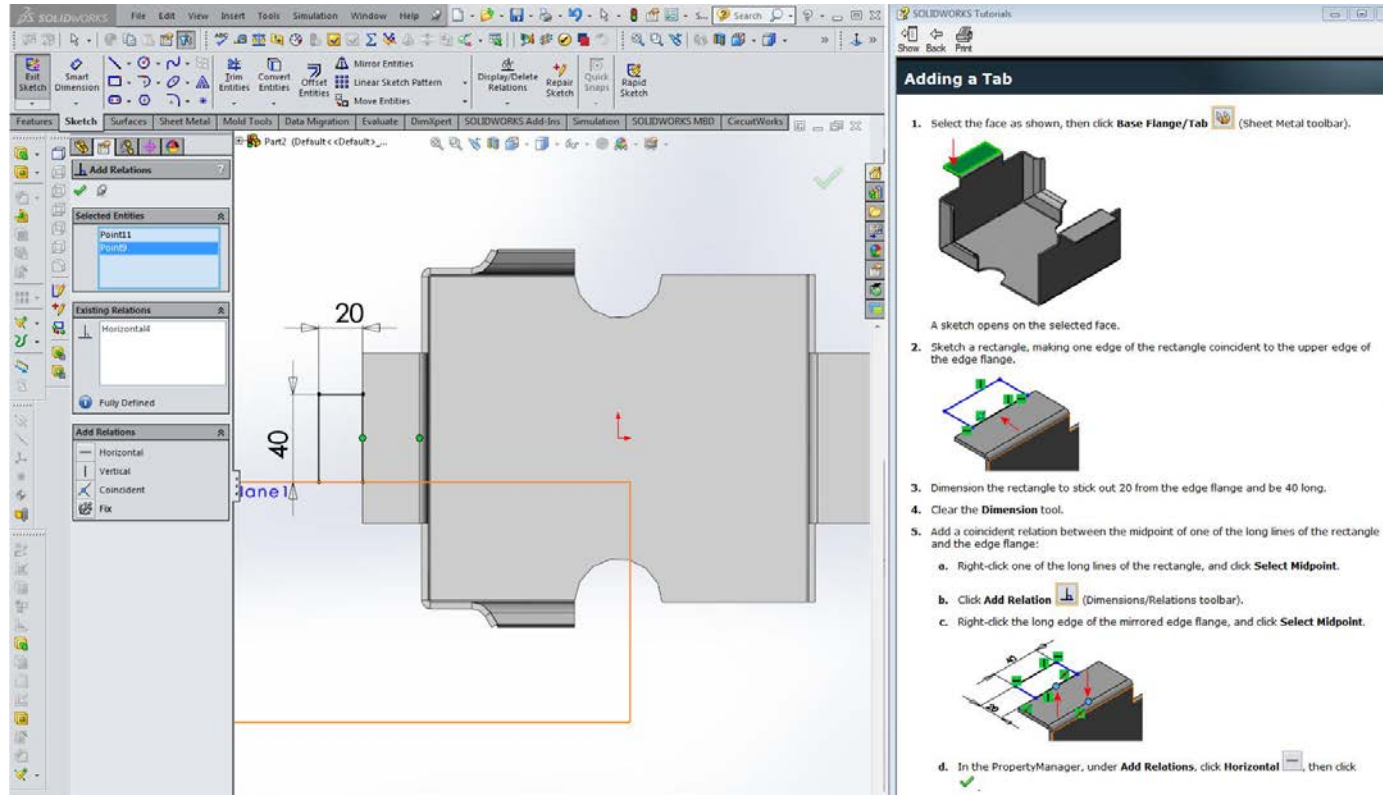
The plane used to mirror the sheet metal feature must be centered between the edges of the base flange.

1. Click **Mirror** (Features toolbar).
2. Expand the FeatureManager design tree, and in the PropertyManager:
  - a. Under **Mirror Face/Plane**, select **Right Plane** for **Mirror Face/Plane**.
  - b. Under **Features to Mirror**, select **Edge-Flange1** for **Features to Mirror**.
3. Click **✓**.

# Mirrored Feature



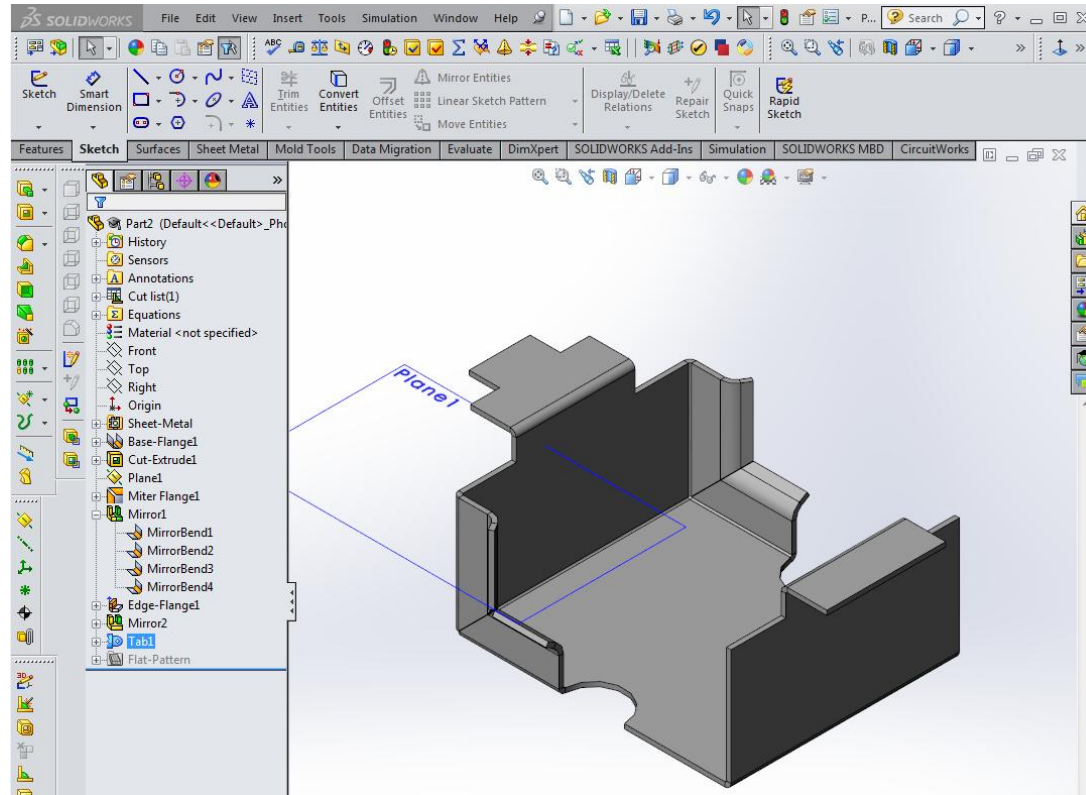
# Adding a Tab



**Adding a Tab**

1. Select the face as shown, then click **Base Flange/Tab** (Sheet Metal toolbar).
2. Sketch a rectangle, making one edge of the rectangle coincident to the upper edge of the edge flange.
3. Dimension the rectangle to stick out 20 from the edge flange and be 40 long.
4. Clear the **Dimension** tool.
5. Add a coincident relation between the midpoint of one of the long lines of the rectangle and the edge flange:
  - a. Right-click one of the long lines of the rectangle, and click **Select Midpoint**.
  - b. Click **Add Relation** (Dimensions/Relations toolbar).
  - c. Right-click the long edge of the mirrored edge flange, and click **Select Midpoint**.
  - d. In the PropertyManager, under **Add Relations**, click **Horizontal**, then click **OK**.

# Finished Tab



# Bending a Tab with Sketched Bend

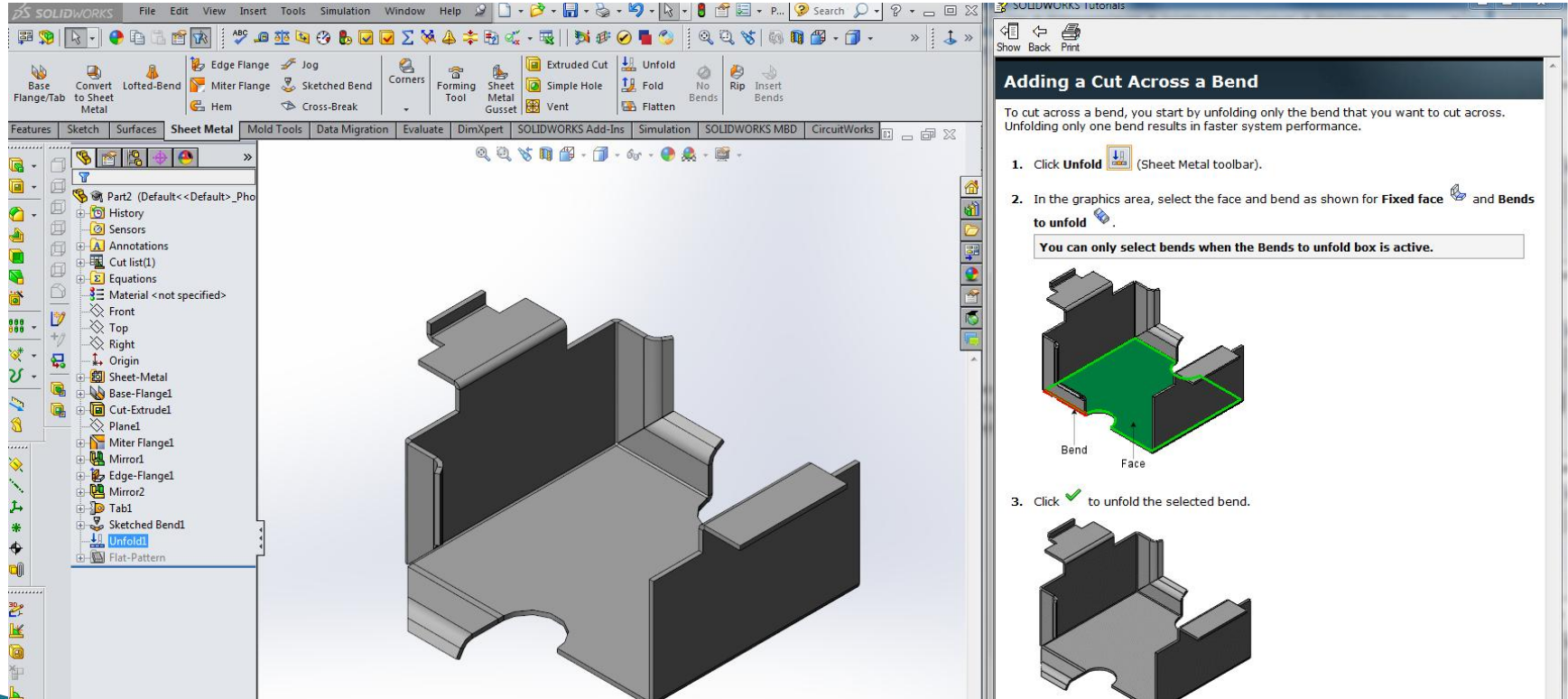
**Bending a Tab**

Now you specify how to bend the tab.

1. Select the top face of the tab, then click **Sketched Bend** (Sheet Metal toolbar).
2. Sketch a horizontal line of any length. Dimension it as shown.  
**The bend line does not need to be the exact length of the face you are bending.**
3. Click **Exit Sketch** (Sketch toolbar).
4. In the PropertyManager under **Bend Parameters**:
  - a. Select the face with the sketch below the bend line for **Fixed Face**.
  - b. Click **Material Outside**.
  - c. Set **Bend Angle** to 90.
  - d. Select **Use default radius**.
5. Click **✓**.



# Unfolding a Bend



**Adding a Cut Across a Bend**

To cut across a bend, you start by unfolding only the bend that you want to cut across. Unfolding only one bend results in faster system performance.

1. Click **Unfold** (Sheet Metal toolbar).
2. In the graphics area, select the face and bend as shown for **Fixed face** and **Bends to unfold**.

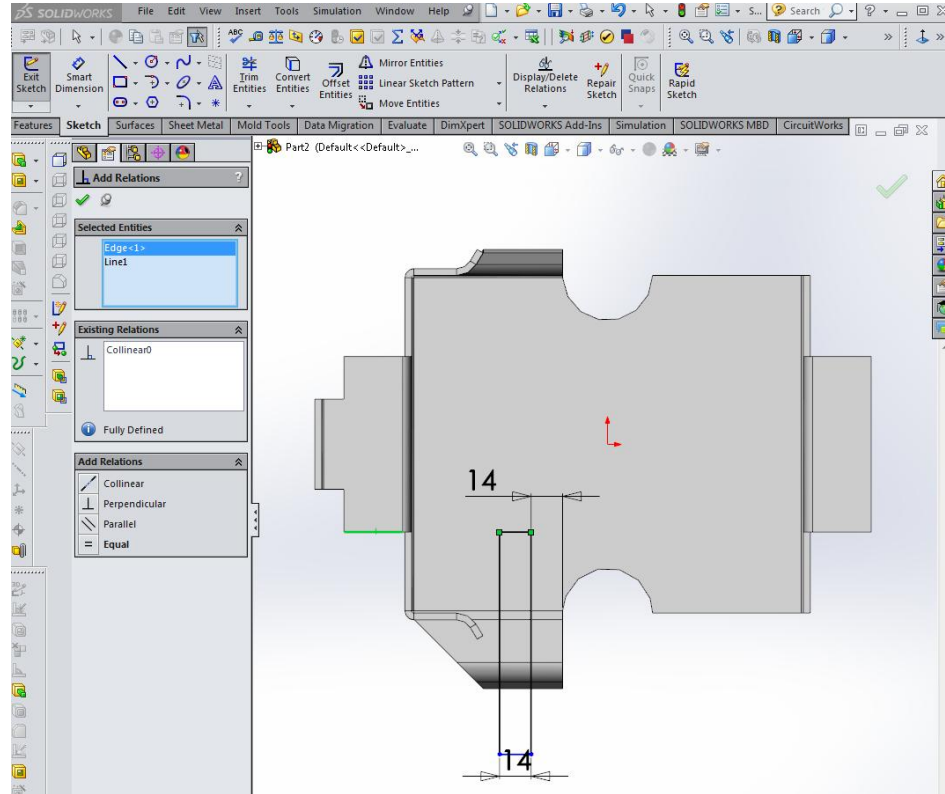
**You can only select bends when the Bends to unfold box is active.**

Bend Face

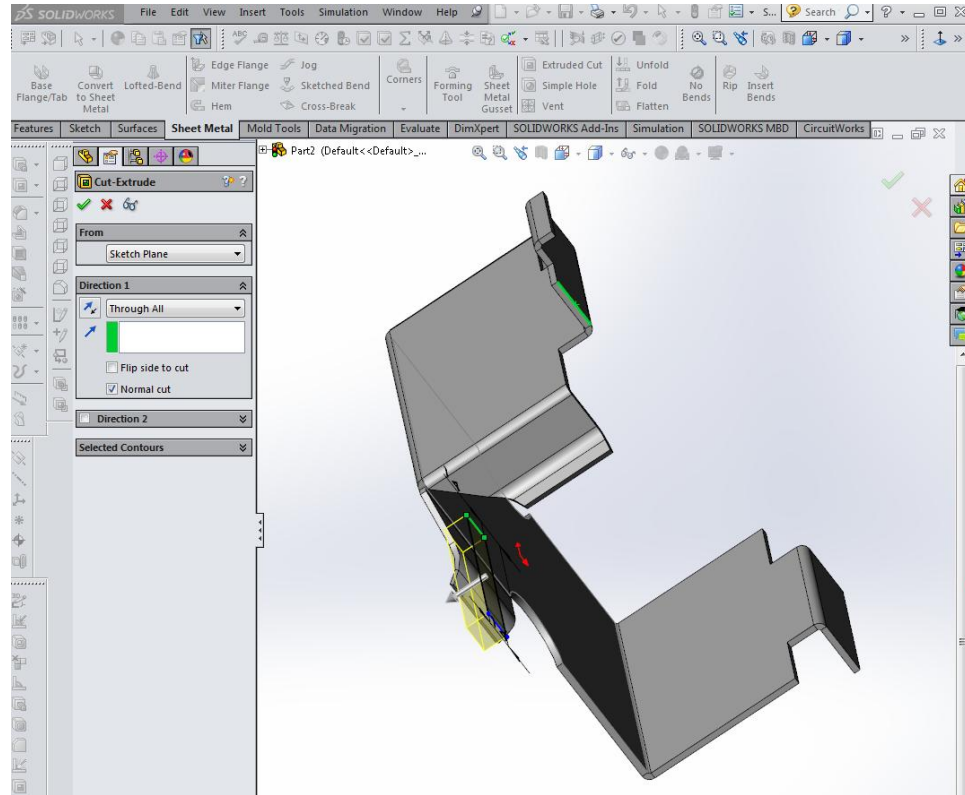
3. Click  to unfold the selected bend.



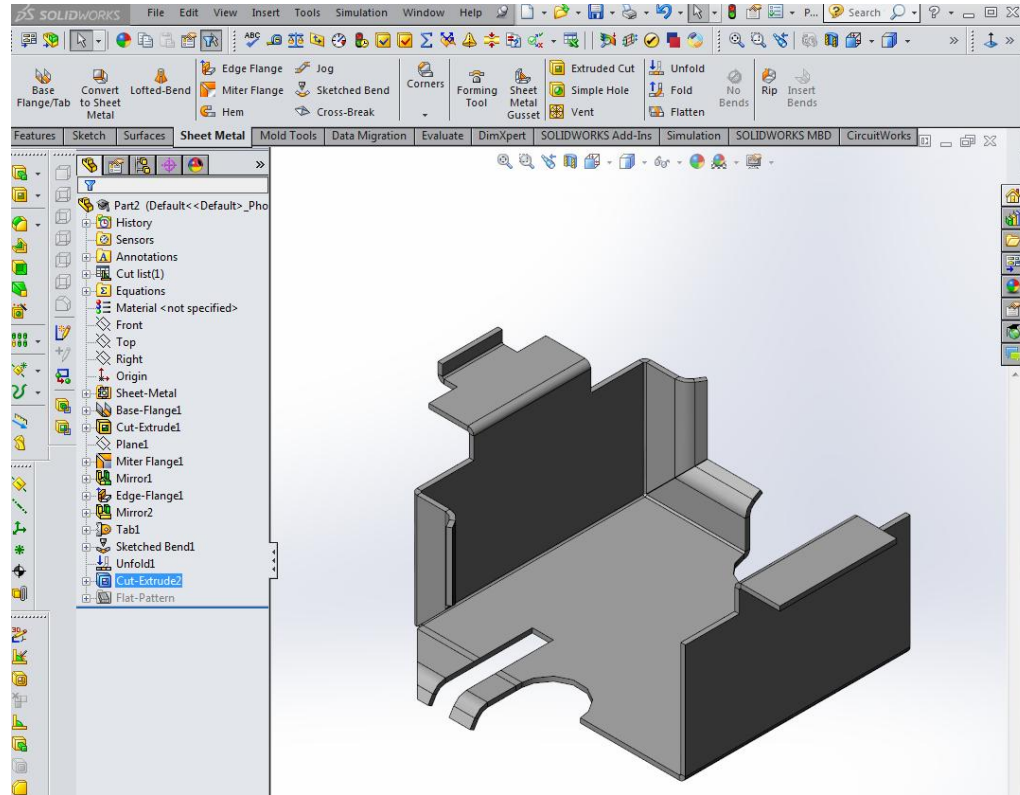
# Adding a Collinear Relationship



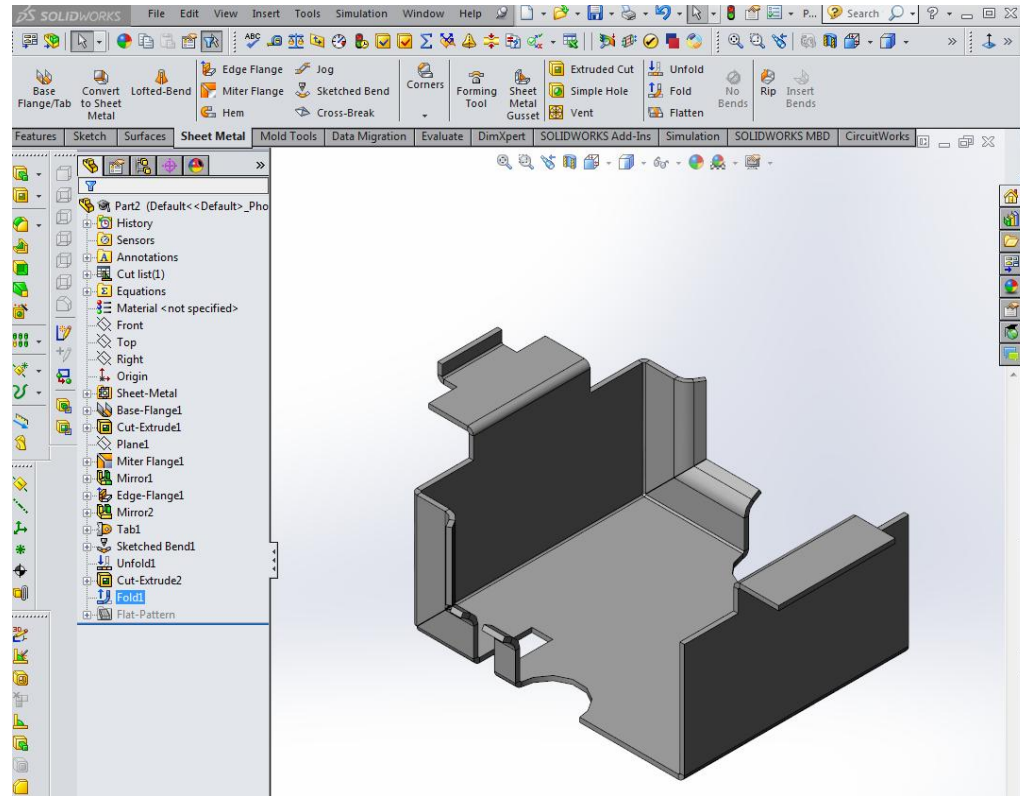
# Extruded Cut Through All



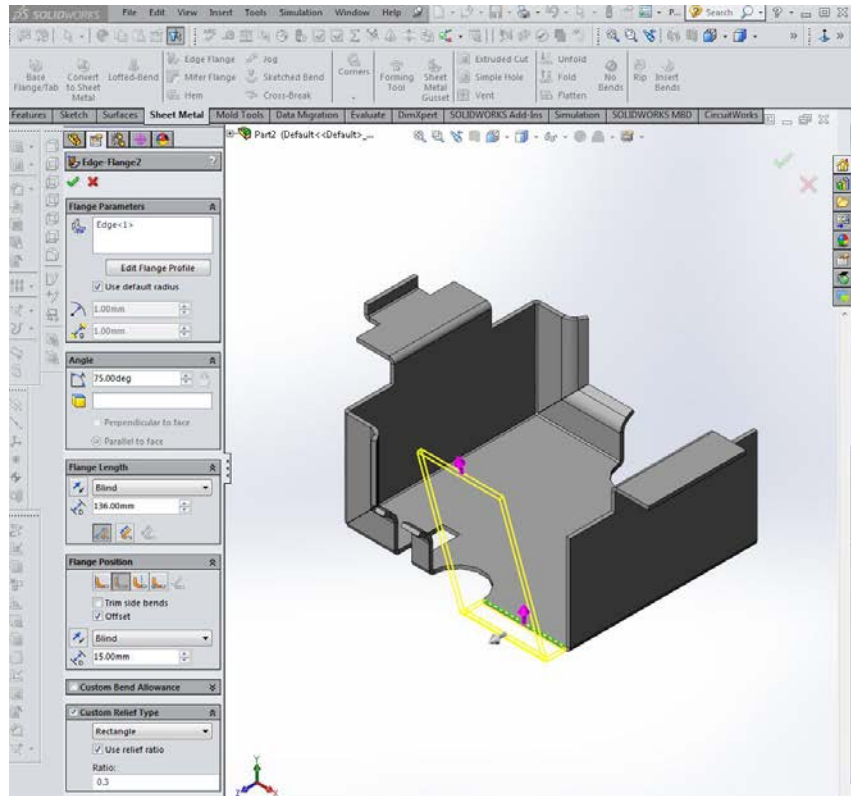
# Finished Cut



# Fold – Collect All Bends

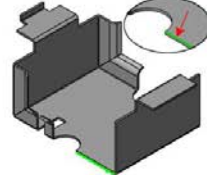



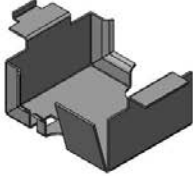
# Creating an Edge Flange



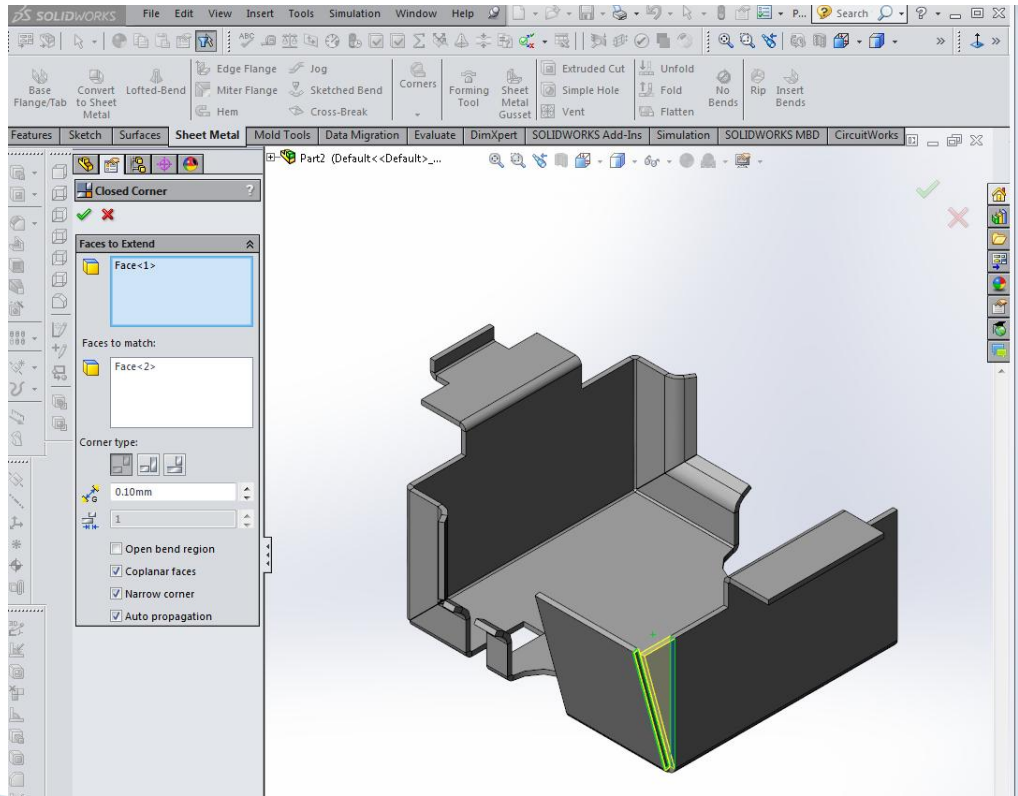
## Creating Closed Corners

Now add a closed corner to one side of the base flange. A closed corner extends a flange so that the gap between two flanges is narrowed. To create a closed corner, first add an edge flange to the base flange.

1. Click **Edge Flange** (Sheet Metal toolbar).
2. Select the edge as shown.  

3. In the PropertyManager:
  - a. Under **Angle**, set **Flange Angle** to 75.
  - b. Under **Flange Length**, set **Length End Condition** to **Blind**.
  - c. Under **Flange Length**, if the preview faces down, click **Reverse Direction**.
  - d. Under **Flange Length**, set **Length** to 85.
  - e. Under **Flange Position** select **Material Outside**.
  - f. Under **Custom Relief Type**, in **Relief Type**, select **Rectangle**.
  - g. Under **Custom Relief Type**, select **Use relief ratio**.
  - h. Under **Custom Relief Type** set **Ratio** to 0.3.
4. Click .



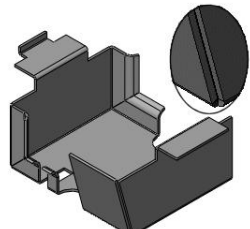
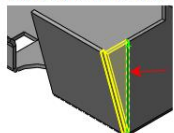
# Closing the Corner with Butt Joint



**Completing the Closed Corners**

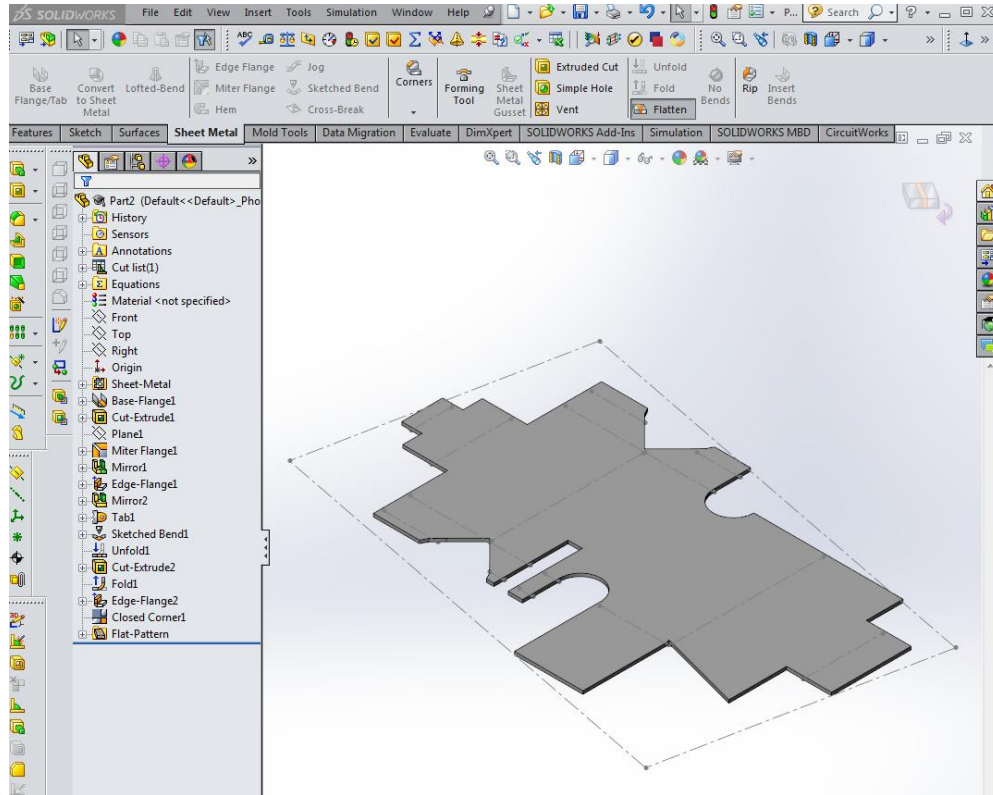
Next, add a closed corner by extending the face that is adjacent to the angled edge flange you just added.

1. Click **Closed Corner** (Sheet Metal toolbar).
2. Select the face of the base flange as shown for **Faces to Extend**.
3. Select **Butt** for **Corner type**.
4. Click .





# Flatten Sheet

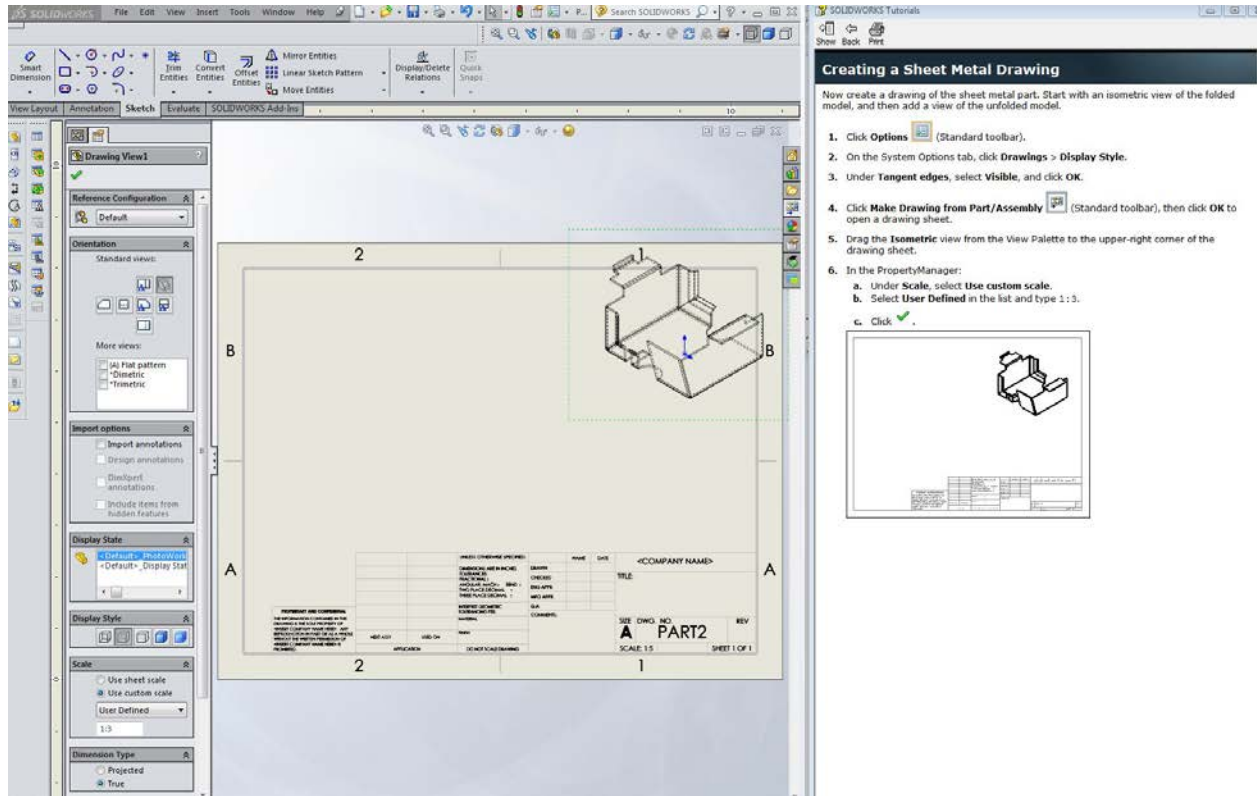


### Flattening and Folding the Part

You can flatten all the bends of a sheet metal part at once.

1. Verify that the rollback bar is rolled to the end, then click **Flatten** (Sheet Metal toolbar).  
This is the same as unsuppressing the Flat-Pattern feature that was created with the Base Flange feature.  
The flattened sheet metal part appears with all of the bend lines shown. You may also see a bounding box around the part. The bounding box is the smallest rectangle in which the flat pattern can fit.
2. To fold the part back up, click **Flatten** again.
3. Save the part.

# Drawing from Part-Assembly, Scale 1-3



**Creating a Sheet Metal Drawing**

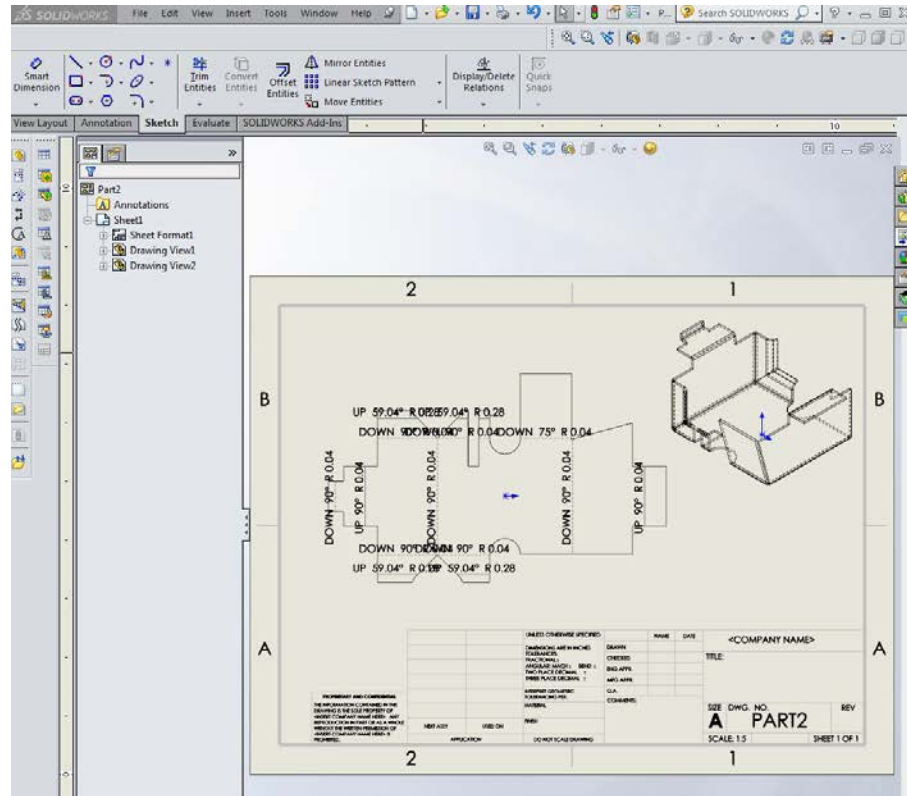
Now create a drawing of the sheet metal part. Start with an isometric view of the folded model, and then add a view of the unfolded model.

1. Click **Options** (Standard toolbar).
2. On the System Options tab, click **Drawings > Display Style**.
3. Under **Tangent edges**, select **Visible**, and click **OK**.
4. Click **Make Drawing from Part/Assembly** (Standard toolbar), then click **OK** to open a drawing sheet.
5. Drag the **Isometric** view from the View Palette to the upper-right corner of the drawing sheet.
6. In the PropertyManager:
  - a. Under **Scale**, select **Use custom scale**.
  - b. Select **User Defined** in the list and type 1:3.
  - c. Click **✓**.

The drawing sheet shows a 3D model of the part with dimensions A, B, and 2. The drawing sheet also includes a title block with the following information:


PROPERTY AND CONTROLS	NAME	DATE	<COMPANY NAME>
DESIGNED BY	USER1		
CHECKED BY			
DATE			
SCALE	1:3		
SIZE	DWG NO.	NO.	REV
A	PART2		
SCALE 1:3			SHEET 1 OF 1

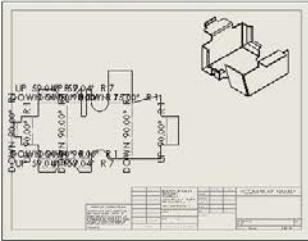
# Model View




### Completing the Sheet Metal Drawing

Next, add a view of the unfolded model. A flat pattern view is automatically added in the PropertyManager when you create a sheet metal part.

1. Click **Model View** (Drawing toolbar).
2. In the PropertyManager:
  - a. Click .
  - b. Under **Orientation**, in **More views**, select **(A) Flat pattern**.
  - c. Under **Scale**, select **Use custom scale**. Select **User Defined** in the list and type 1:3.
  - d. Click to place the view in the drawing sheet.

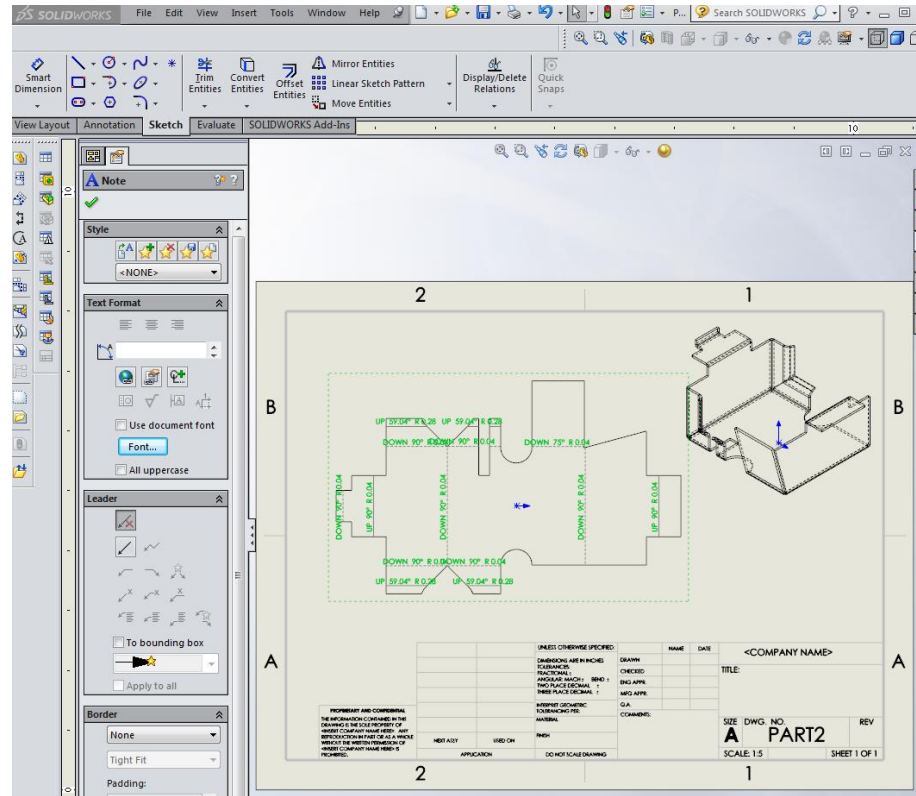


The view orientation of flat patterns in drawing views varies, depending on such factors as the way you extruded the base flange. To rotate the view, select the drawing view, click **Rotate View** (View toolbar), and make your choices in the dialog box. You may also need to use **Flip view** in the PropertyManager.

3. Click .
4. Save the drawing as Cover.slddrw.

Click **Save All** to save both the drawing and the updated model if a message box notifies you that the model referenced in the drawing was modified.

# Adjusting Notes Font



# Adding Leaders and Rotating Notes

**11. Save the drawing.**

**Congratulations!** You have completed this tutorial.

**PROPRIETARY AND CONFIDENTIAL**  
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 HONEYWELL COMPANY NAME HERE. ANY  
 REPRODUCTION OR PART OF A WHOLE  
 WITHOUT THE WRITTEN PERMISSION OF  
 HONEYWELL COMPANY NAME HERE IS  
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UNLESS OTHERWISE SPECIFIED:	DRAWN	NAME	DATE
DIMENSIONS ARE IN INCHES	CHECKED		
TOLERANCES:			
FRACTIONAL:			
ANGULAR MATCH:	ANG APPR		
FACE FINISH:	FACE FINISH		
THREE PLACE DECIMAL:	THREE PLACE DECIMAL		
RESERVED GEOMETRIC TOLERANCING PER ANSYS	Q.A.		
	COMMENTS:		

SIZE A  
 SCALE

# Summary

- ▶ Concepts reviewed:
  - Creating a base-flange.
  - Creating a path for a miter.
  - Mirroring geometry.
  - Creating an edge flange.
  - Mirroring a feature.
  - Adding a tab.
  - Bending with a sketched bend.
  - Unfolding a to make a cut, refolding.
  - Flattening sheet.
  - Making a drawing and modifying the notes with leaders.