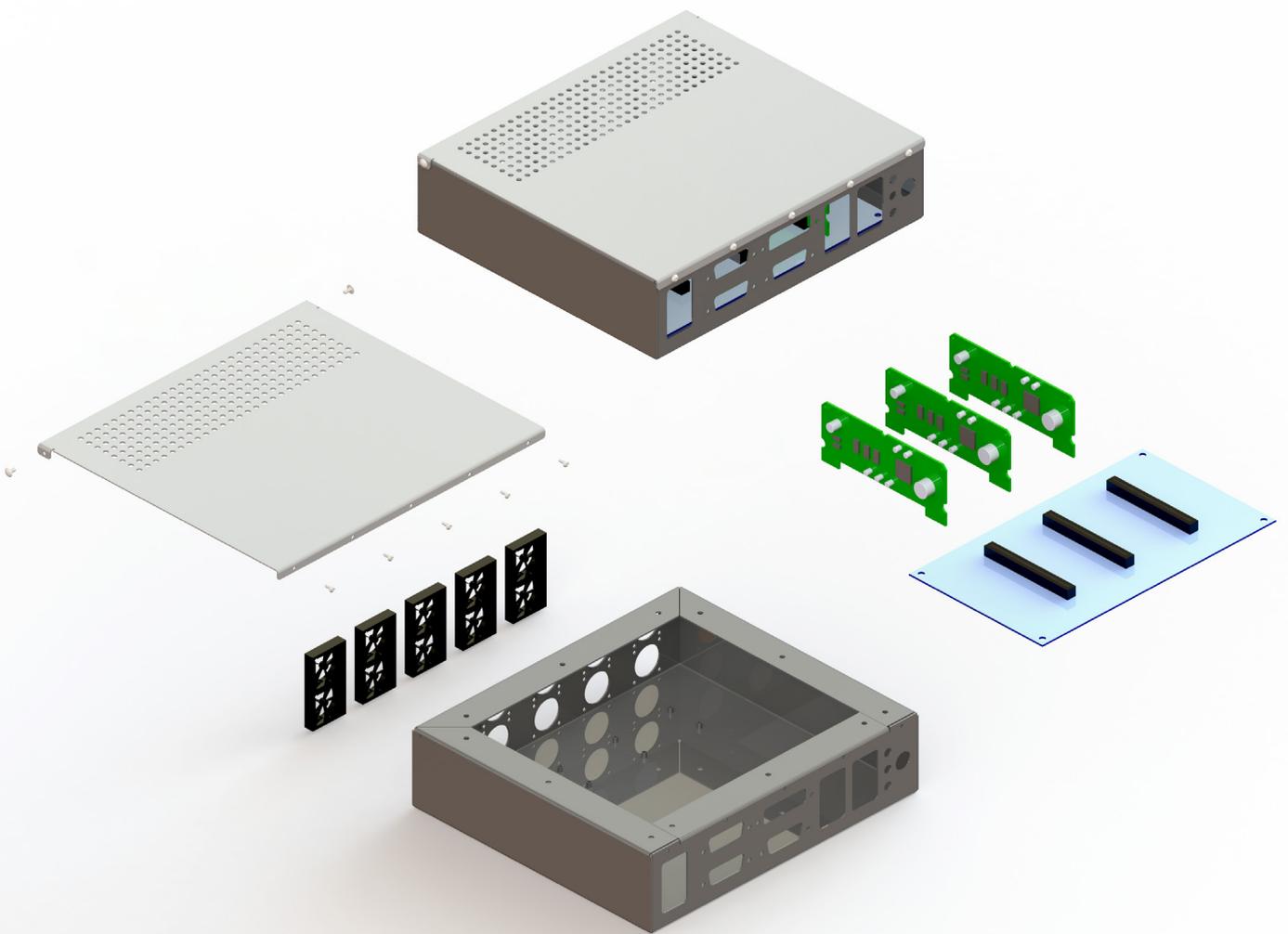


Mastering Sheet Metal and Weldments with **SOLIDWORKS® 2026**

From Design to Manufacturing and Fabrication



Lani Tran, CSWE

Visit the following websites to learn more about this book:



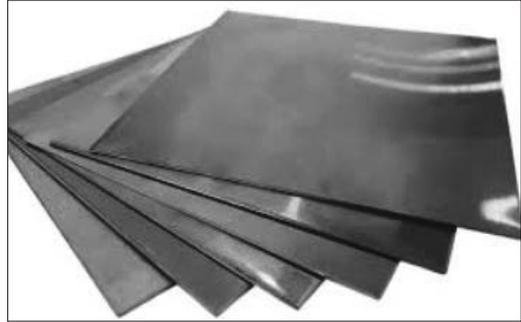
[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

Lesson 1: Introduction to Sheet Metal Parts

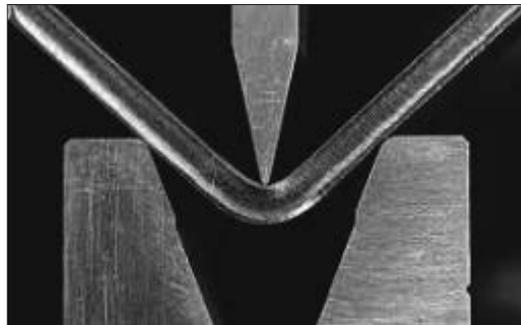
Sheet metal components are metal parts formed from flat sheets by cutting, bending, and shaping. They are widely used in various industries due to their versatility and ease of fabrication. These components can be made from various materials like steel, aluminum, stainless steel, copper, and brass, and can range in thickness from thin foils to thicker plates.



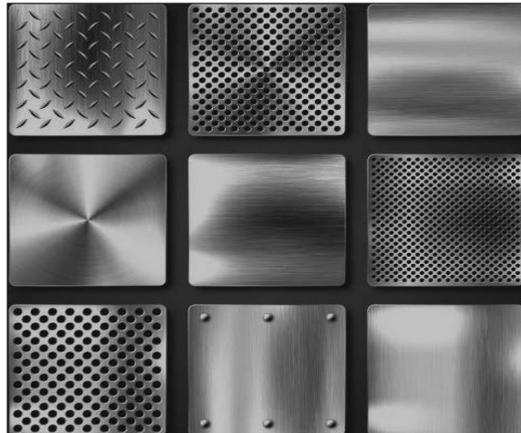
Depending on the complexity of the design and production volume, various methods can be used from laser cutting and welding to CNC punching and stamping. After the metal sheets are cut and formed into the required shapes, they are assembled and welded to create the final products.



The cheapest type of sheet metal is usually low-carbon or mild steel due to its affordability and widespread availability. It is a cost-effective choice for general-purpose applications.



Sheet metal finishes are applied surface treatments that enhance the appearance, durability, and functionality of metal parts. These finishes can range from simple coatings like powder coating and galvanizing to more complex processes like electropolishing and electroless plating. The choice of finish depends on the specific needs of the application, such as corrosion resistance, aesthetics, and cost.



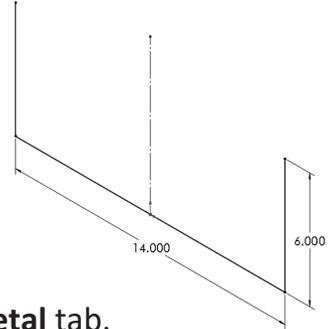
1. Opening a part document:

Browse to the Training Folder and open a part document named: **Sheet Metal_Part1**.



2. Creating a Base Flange:

The 1st feature in a sheet metal part is the parent feature; it usually gets created with the **Base Flange** command.

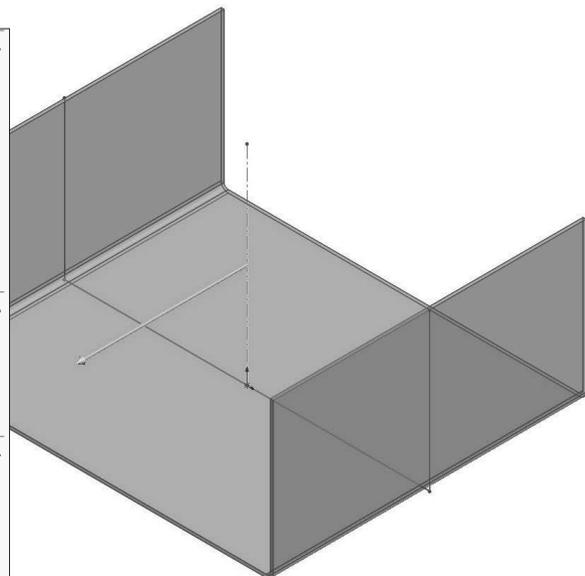
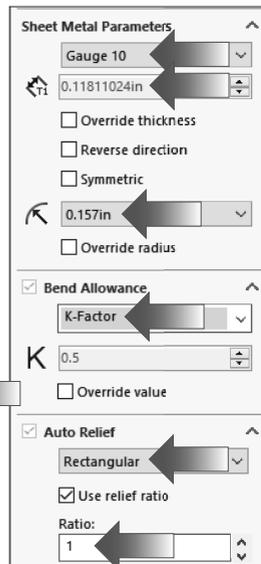
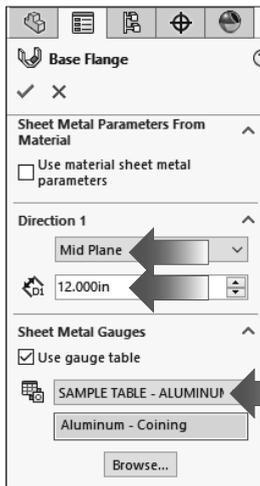


Select **Sketch1** and click **Base Flange** on the **Sheet Metal** tab.

For Direction 1, select **Mid Plane** and enter: **12.00in** for depth.

Enable the **Use Gauge Table** checkbox and select:

- * Sample Table - **Aluminum**
- * Gauge **10** (.11811024 in.)
- * Bend Radius: **.157in.**
- * K-Factor: **0.5**
- * Auto Relief: **Rectangular**
- * Ratio: **1**



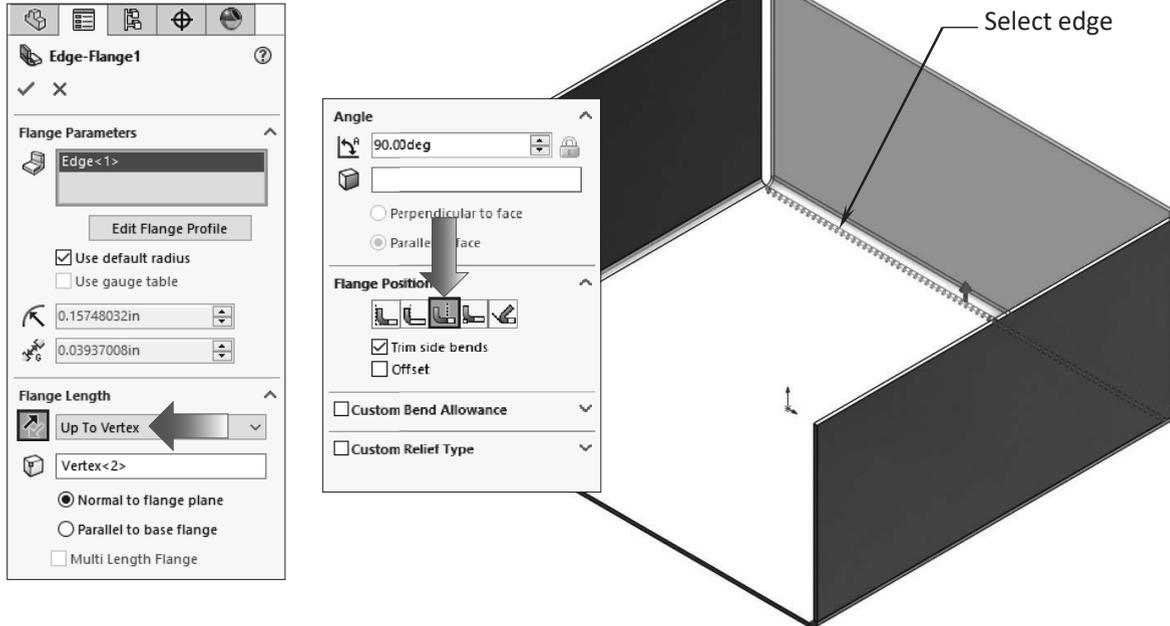
Click **OK**.

3. Making an Edge Flange:

The **Edge Flange** tool combines a bend and sheet metal tab in a single operation. Switch to the **Sheet Metal** tab and click **Edge Flange**.

For Flange Parameters, select the **upper edge** on the right side of the model.

For Flange Length, select the **upper vertex** as noted.



Enable the **Use Default Radius** checkbox.

Keep the Bend Angle at **90deg**.

For Flange Position, select:
Bend Outside.

Click **OK**.

The gap between the flanges will be extended and closed-off in the next step.

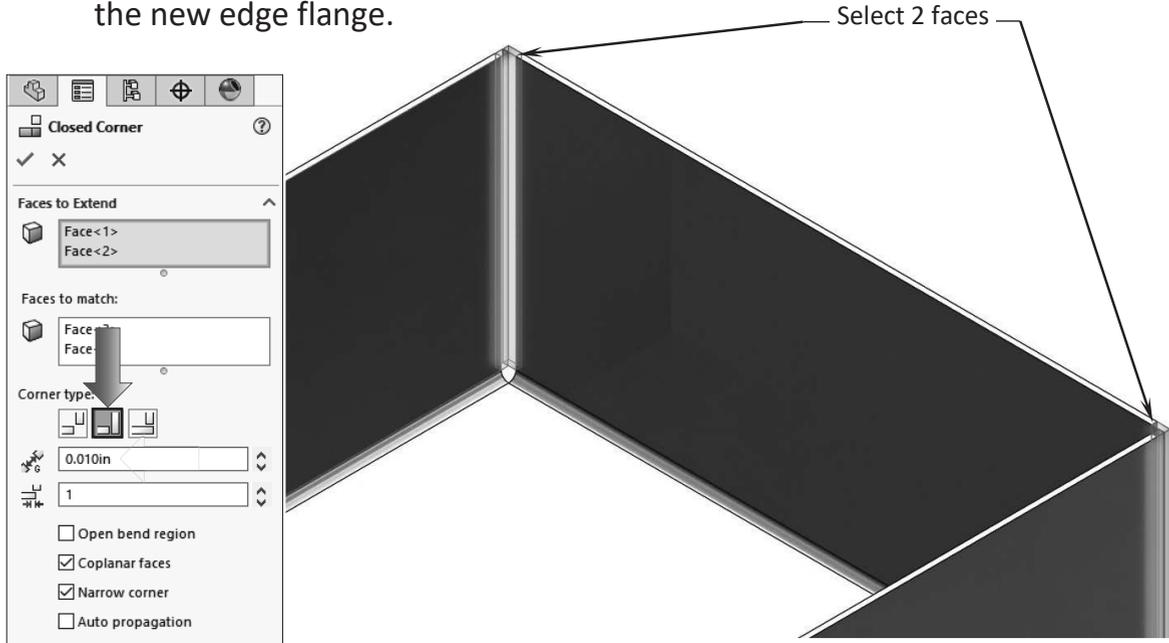


4. Closing the corners:

The **Closed Corner** options are used to close the gaps between flange corners and control if and how they overlap.

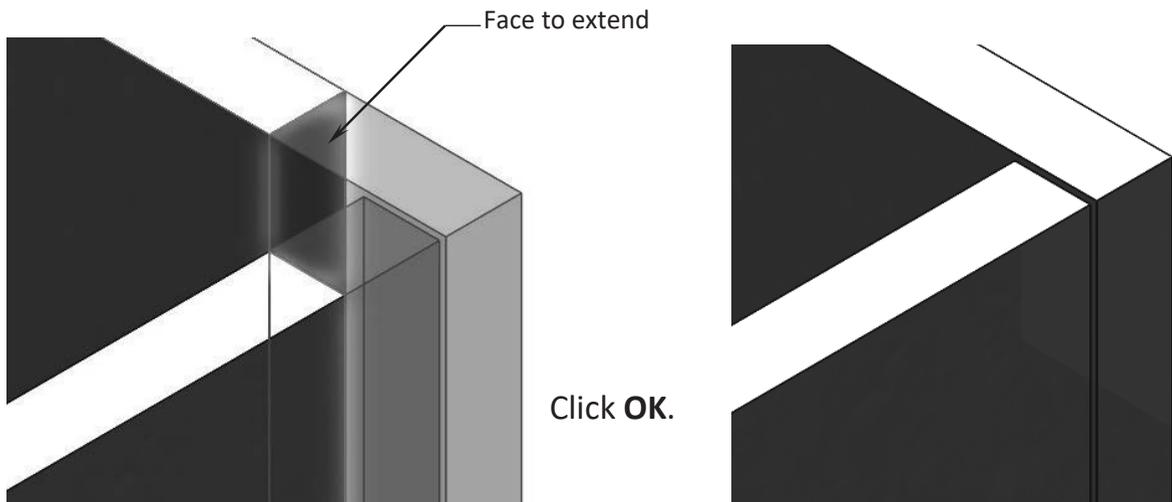
Select: **Corners, Closed Corners** on the **Sheet Metal** tab.

For Faces to Extend, select the the **thin faces** on the left and right sides of the new edge flange.



The Faces to Match are selected automatically.

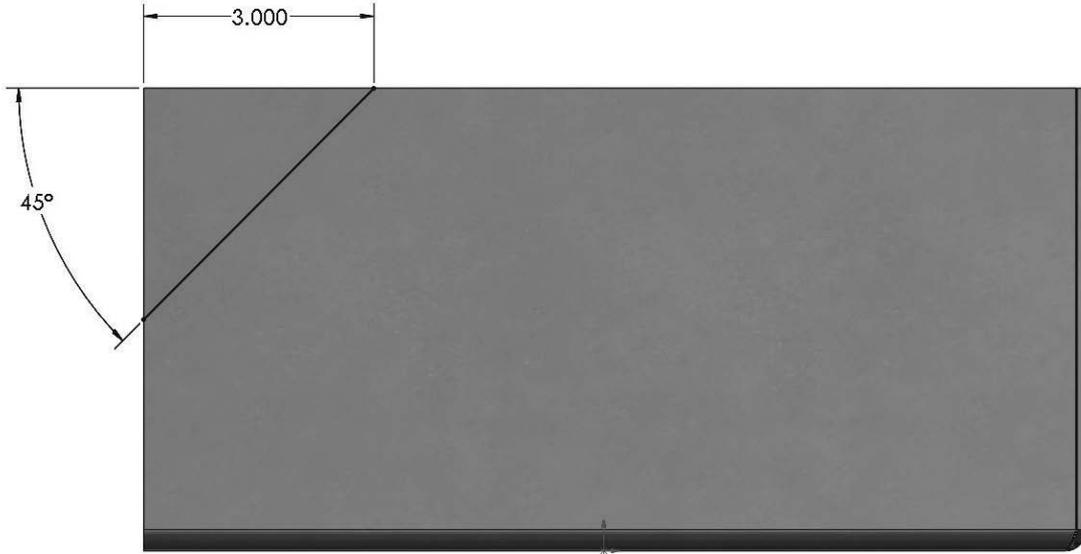
For Corner Type, select: **Overlap**. Keep the gap at the default **.010in**.



5. Making a cut:

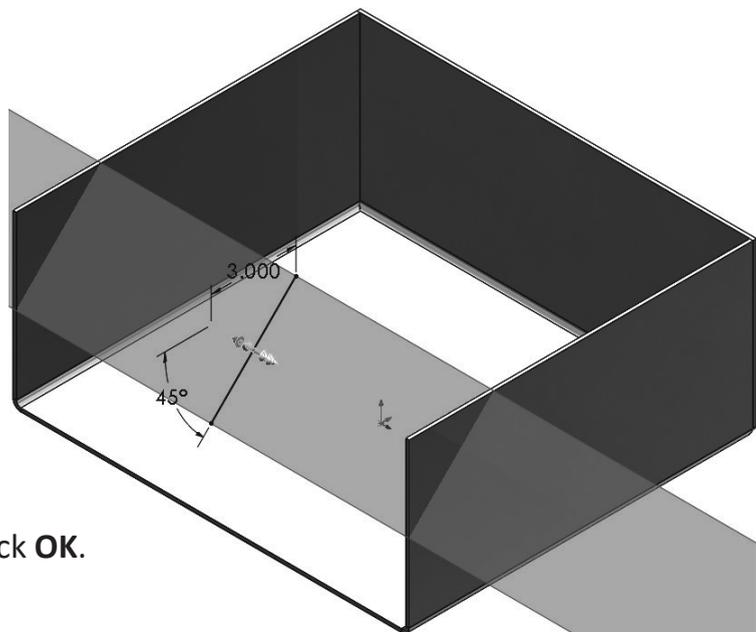
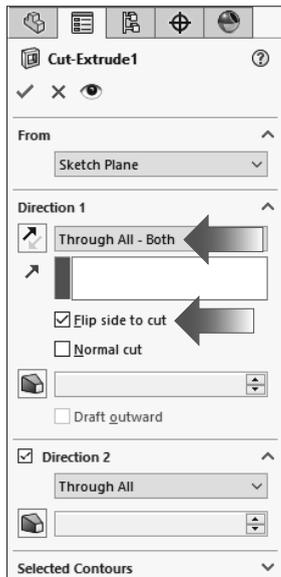
Select the Right plane and open a **new sketch**.

Sketch a **line** and add the dimensions shown in the image to fully define the sketch.



Switch to the **Sheet Metal** tab and click **Extruded Cut**.

Select **Through All Both** and enable the **Flip Side to Cut** checkbox.

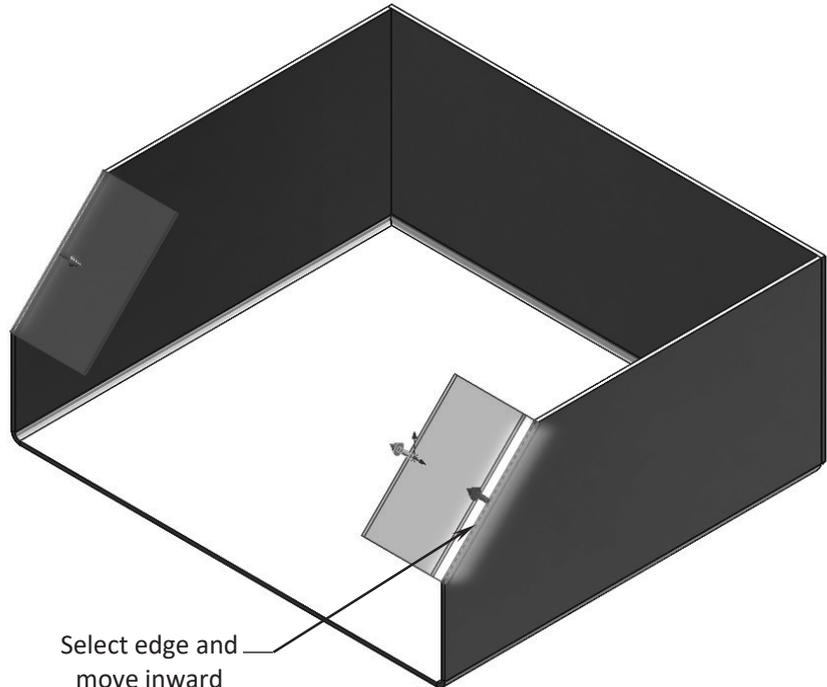
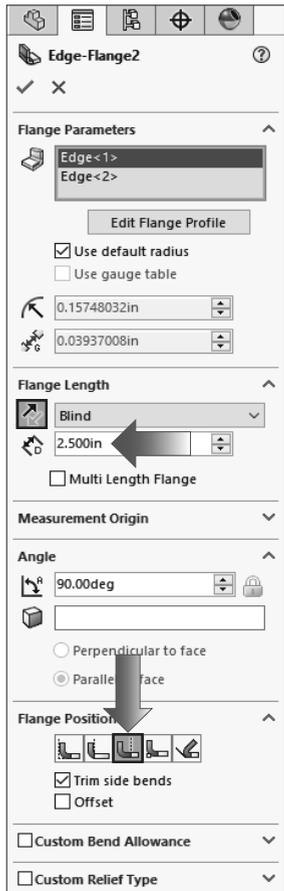


Click **OK**.

6. Adding the angled flanges:

Click **Edge Flange** on the **Sheet Metal** tab.

For Flange Parameters, select the edge as indicated below, move inward and click the mouse to lock the direction.



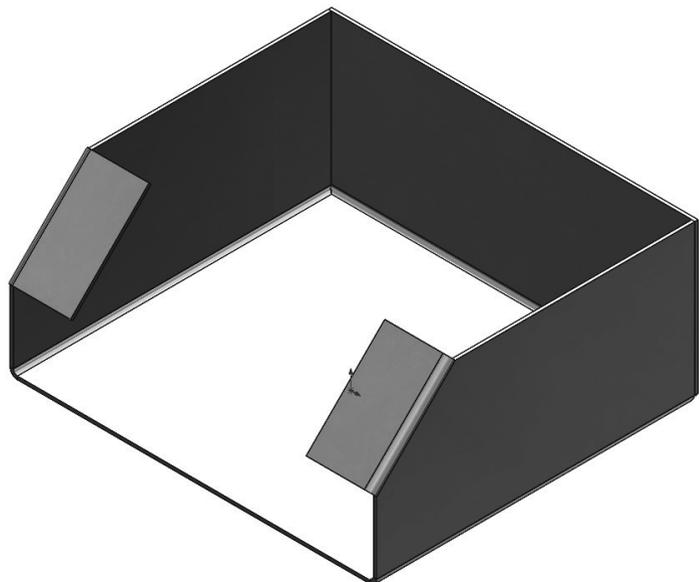
Next, select the edge on the opposite side.

For Depth, enter **2.500in.**

Keep the Angle at **90deg.**

For Flange Position, select: **Bend Outside.**

Click **OK.**

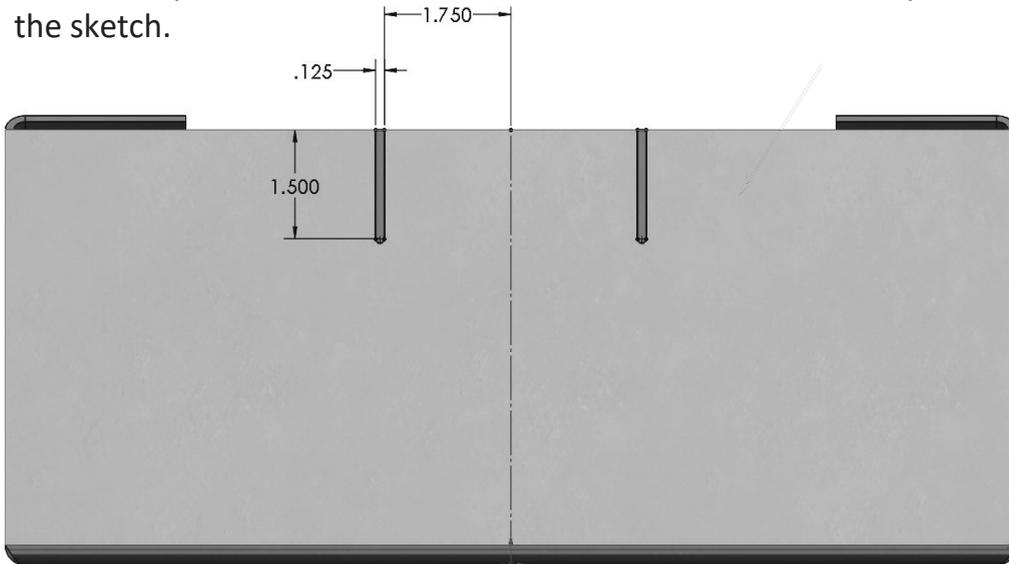


7. Making a manual relief cut:

Relief cuts are generally used to prevent splitting, binding, or cracking when bending sheet metal parts.

Select the face in the back of the model and open a **new sketch**.

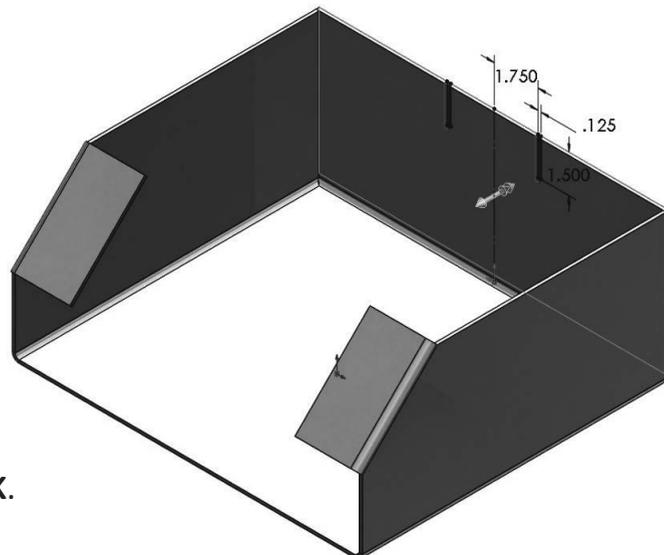
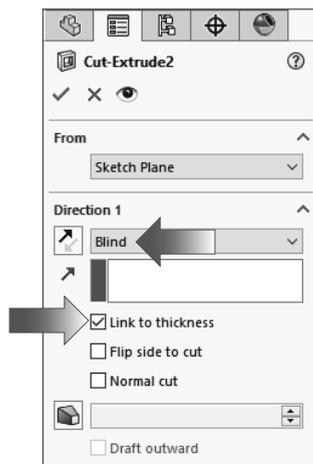
Sketch the profile and add the dimensions and relations to fully define the sketch.



Switch to the **Sheet Metal** tab and click **Extruded Cut**.

For Direction 1, use the default **Blind** option.

Enable the **Link To Thickness** checkbox.



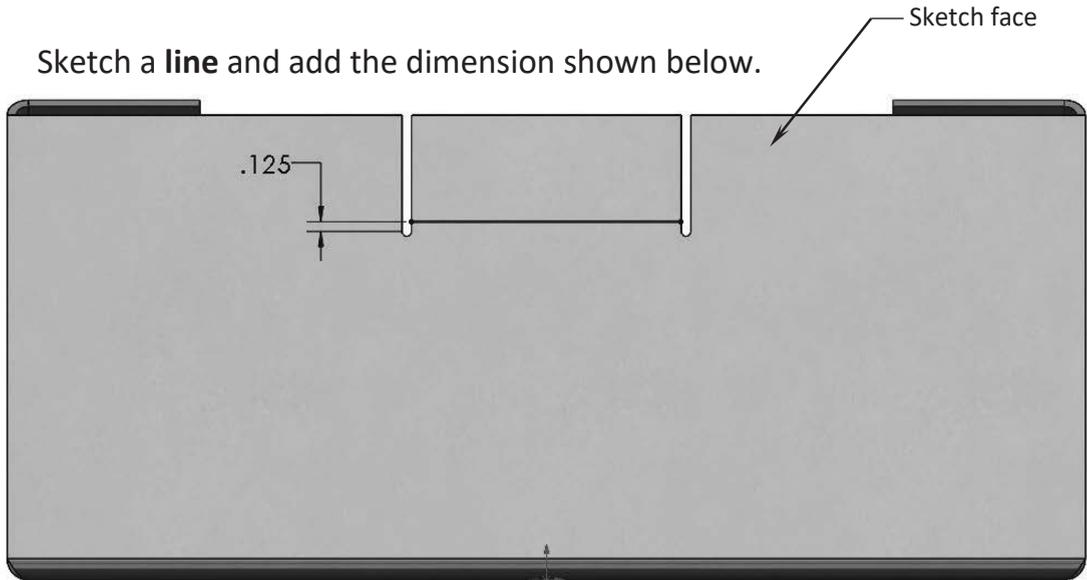
Click **OK**.

8. Creating a Sketched Bend feature:

The **Sketched Bend** tool uses sketch lines to bend sheet metal parts. Only lines are allowed in the sketch.

Select the face in the back of the model and open a **new sketch**.

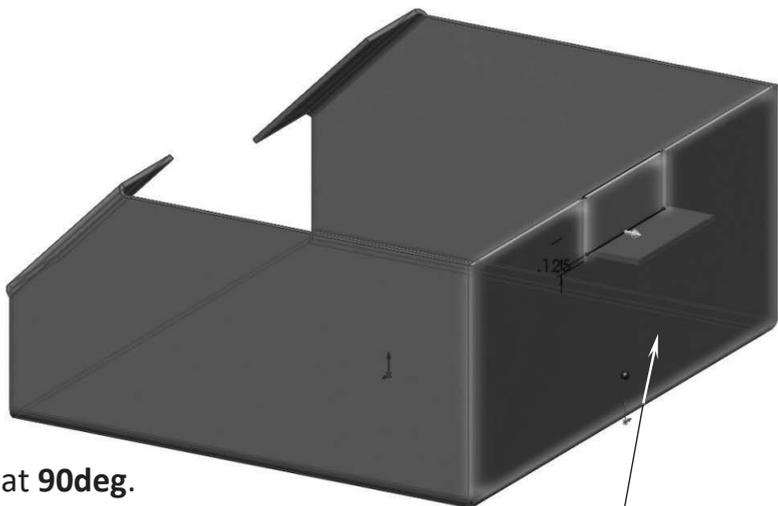
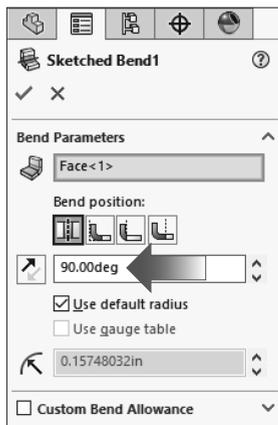
Sketch a **line** and add the dimension shown below.



Switch to the **Sheet Metal** tab and click **Sketched Bend**.

For Fixed Face, select the area below the line.

For Bend Position, use the default **Bend Centerline**.



Keep the Bend Angle at **90deg**.

Click **OK**.

Fixed face

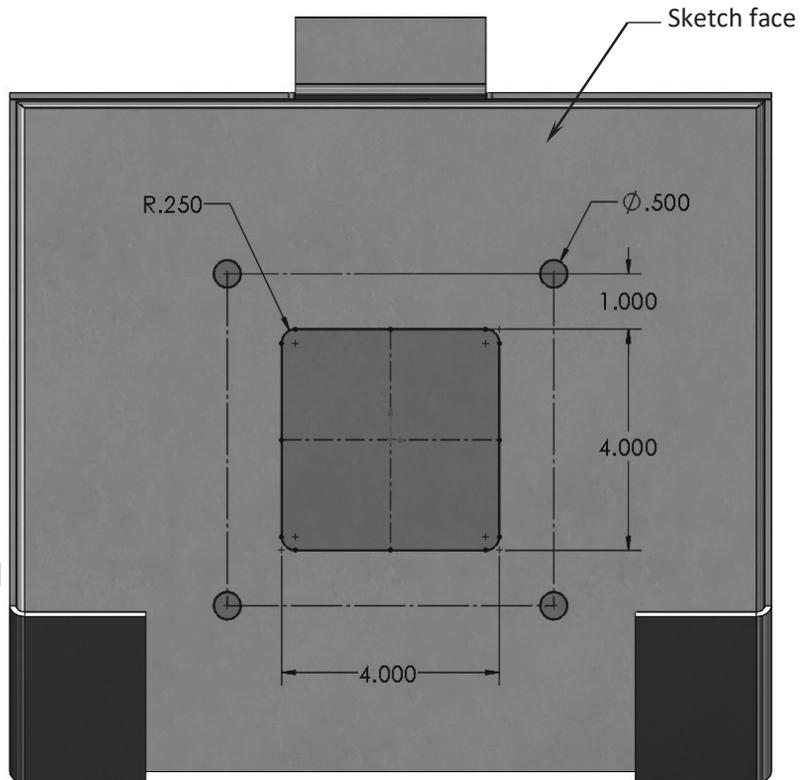
9. Adding the bottom cuts:

Select the face as noted and open a **new sketch**.

Sketch the profile shown in the image on the right.

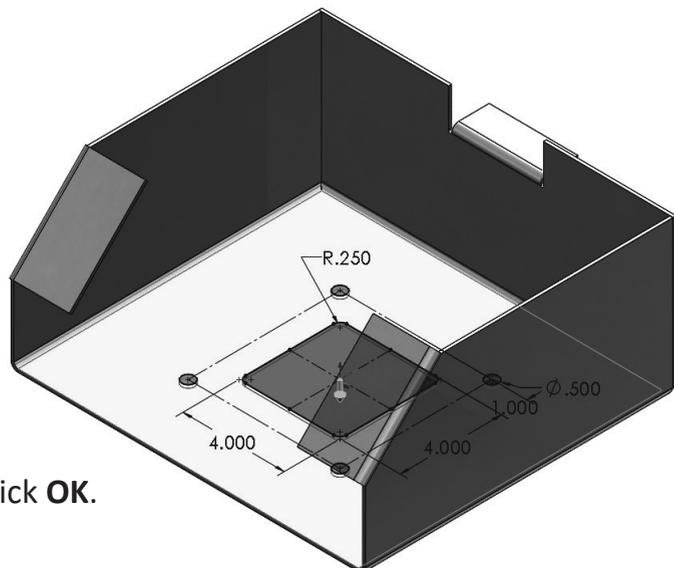
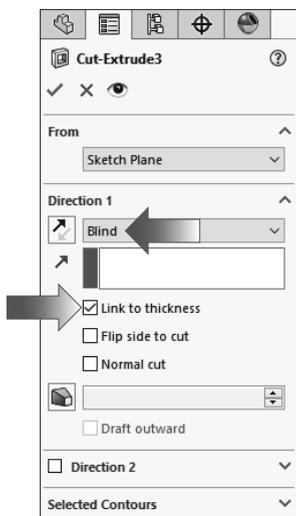
Use the Mirror tool to speed up the process and to keep sketch entities symmetrical to each other.

Add the dimensions shown to fully define the sketch.



Switch to the **Sheet Metal** tab and click **Extruded Cut**.

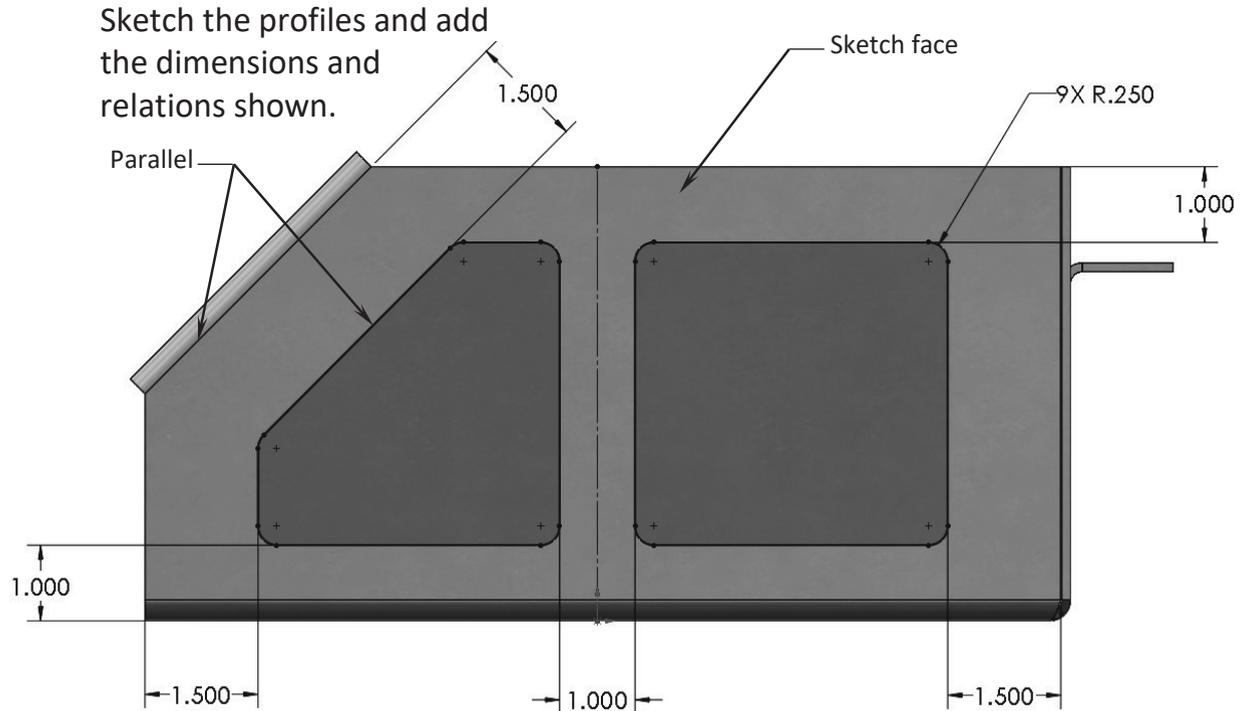
Use the default **Blind** type and enable the **Link To Thickness** checkbox.



Click **OK**.

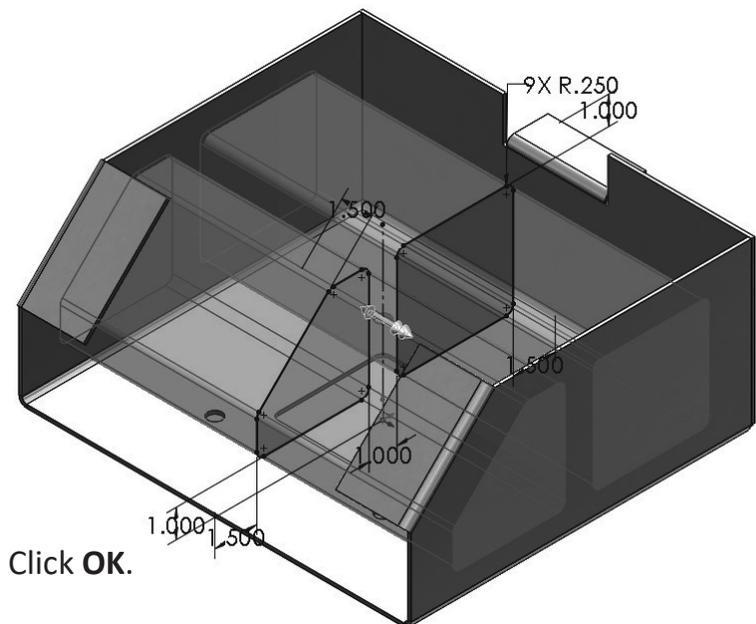
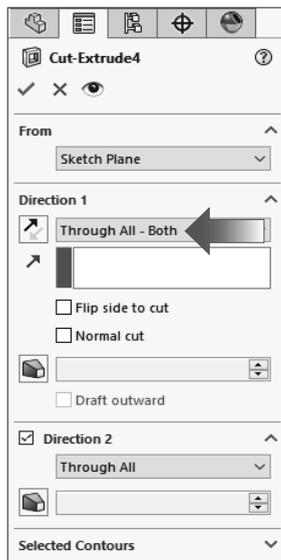
10. Adding the side cuts:

Open a **new sketch** on the side face.



Switch back to the **Sheet Metal** tab and click **Extruded Cut**.

For Direction 1, select **Through All - Both**.

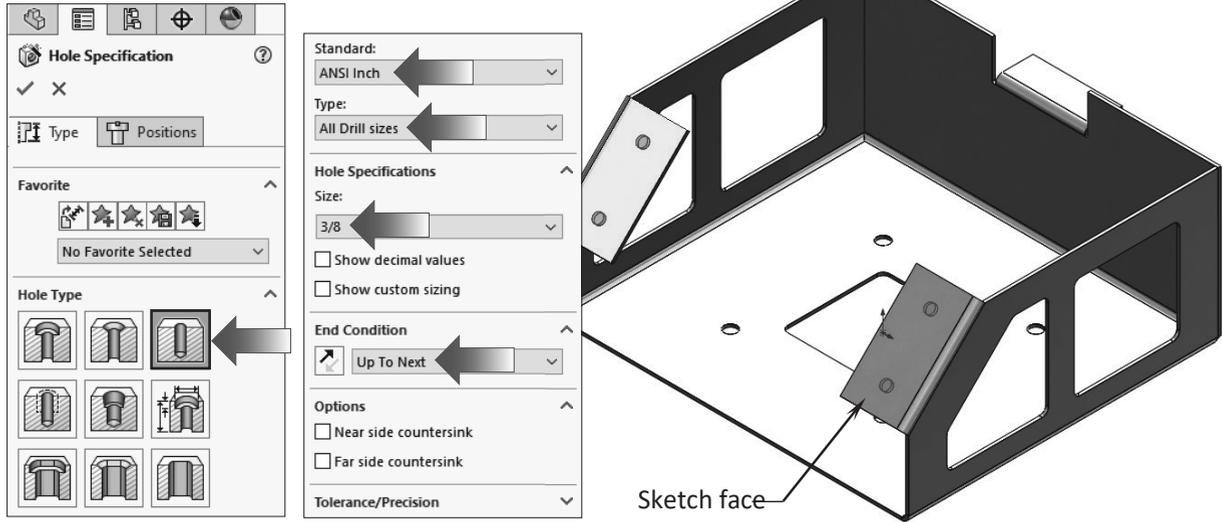


Click **OK**.

11. Adding the 3/8" holes:

Switch to the **Features** tab and click **Hole Wizard**.

For Hole Type, select **Simple Hole**.



Standard: **ANSI Inch**
Size: **3/8**

Type: **All Drill Sizes**
End Condition: **Up To Next**

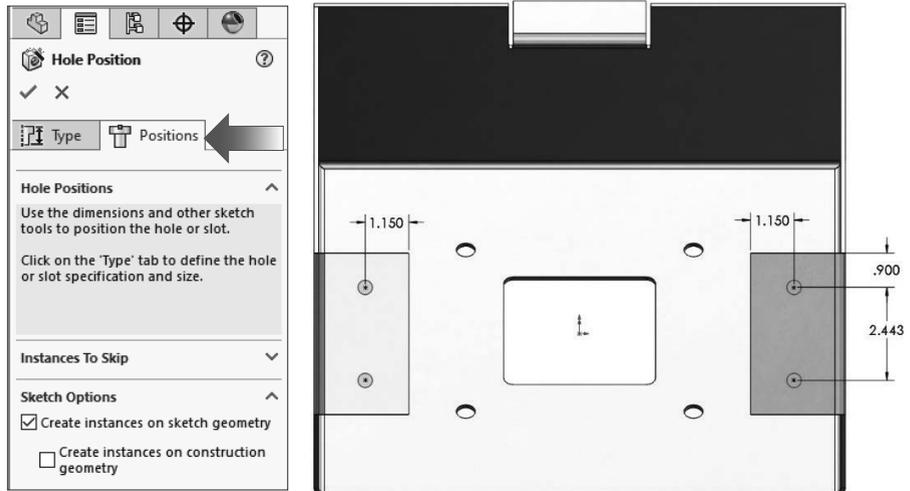
Click the **Position** tab and select the **Sketch Face** to rotate the view normal to you.

SOLIDWORKS enables the **Point** command automatically.

Place **two points** on each tab, and add

the vertical, horizontal relations between the points, then add dimensions.

Click **OK**.



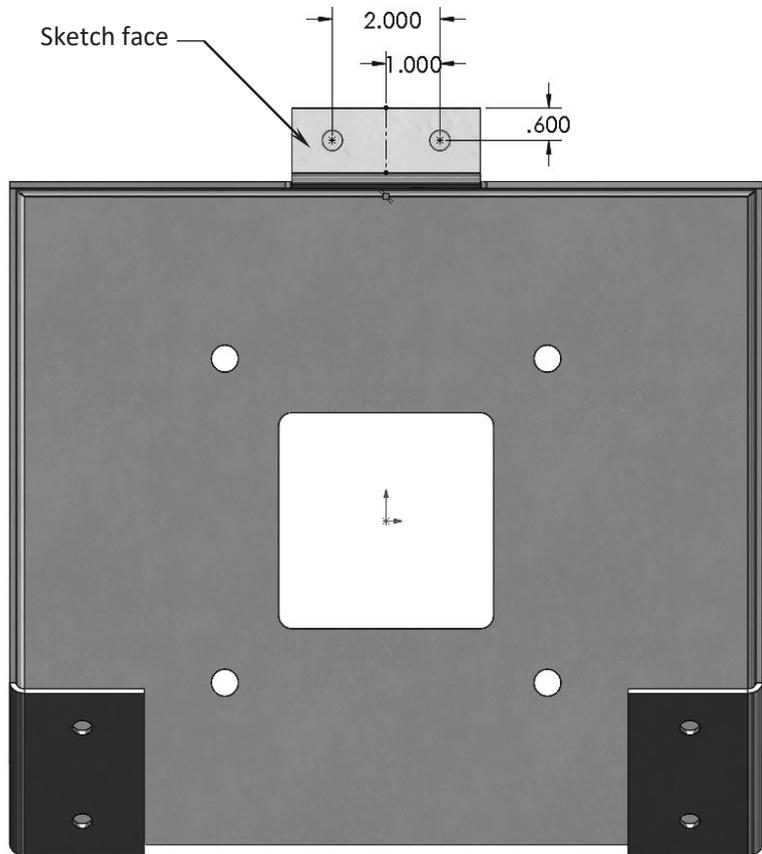
12. Adding the additional holes:

Change to the **Top** orientation (Control + 5).

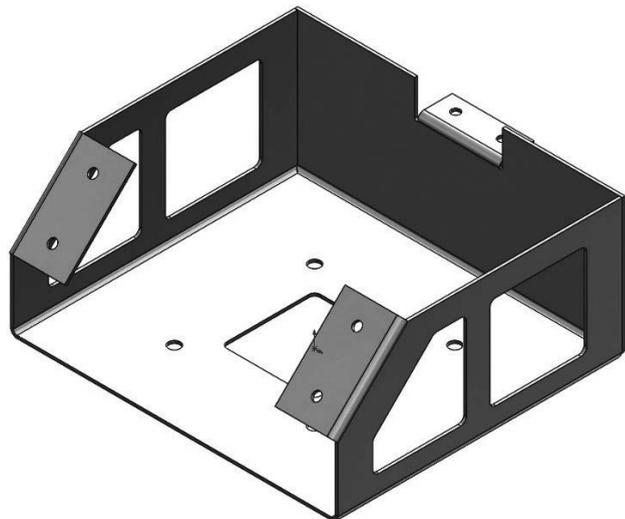
Click **Hole Wizard**.

Use the same hole settings as in the previous step and **add two additional holes** on the face of the tab, as shown in the image on the right.

Click **OK**.



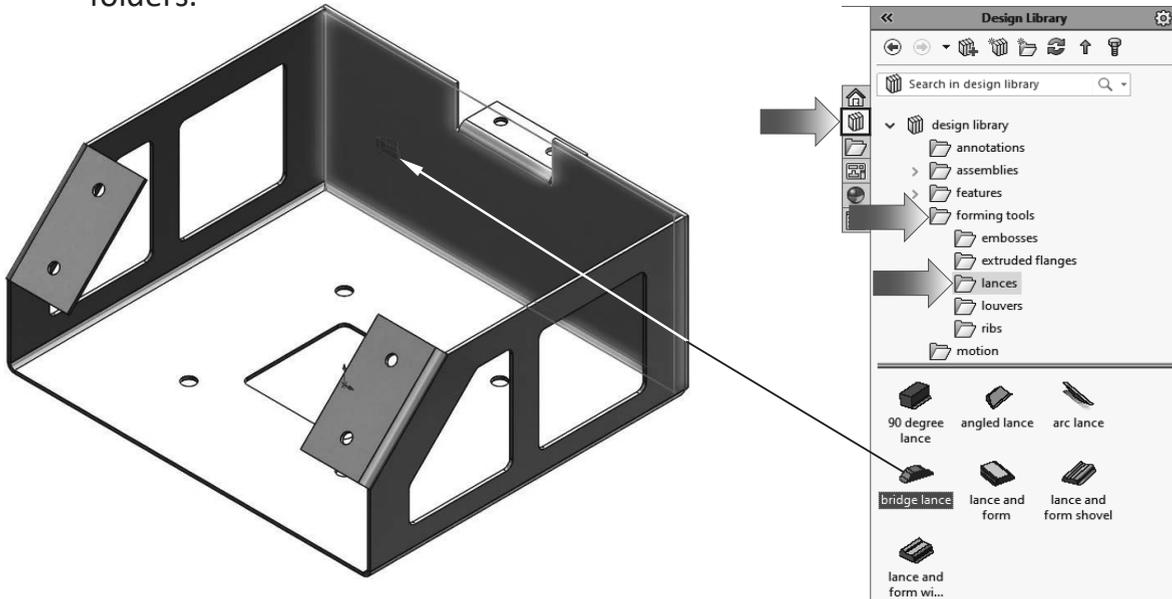
***Note:** When using the Hole Wizard, the 3D Sketch option allows the holes to be added to multiple surfaces, on multiple elevations or different planes.*



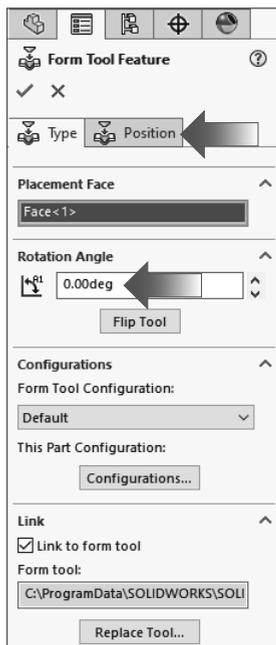
13. Inserting a form tool:

Forming tools are solid parts that act as dies that bend, stretch, or otherwise form sheet metal to create form features such as louvers, lances, flanges, and ribs.

Click the **Design Library** tab and expand the **Forming Tools** and the **Lances** folders.



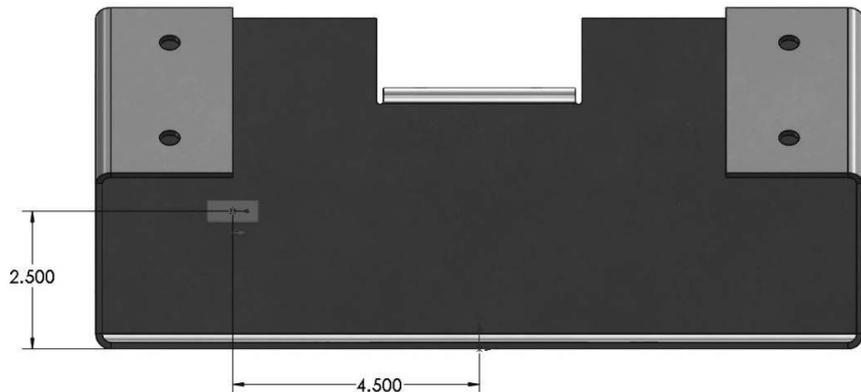
Locate the **Bridge Lance** form tool and drag/drop it to the inside face of the model.



The form tool should protrude outwards and orients horizontally. Change the **Angle** or click **Flip Tool** if needed.

Click the **Position** tab and add the location dimensions.

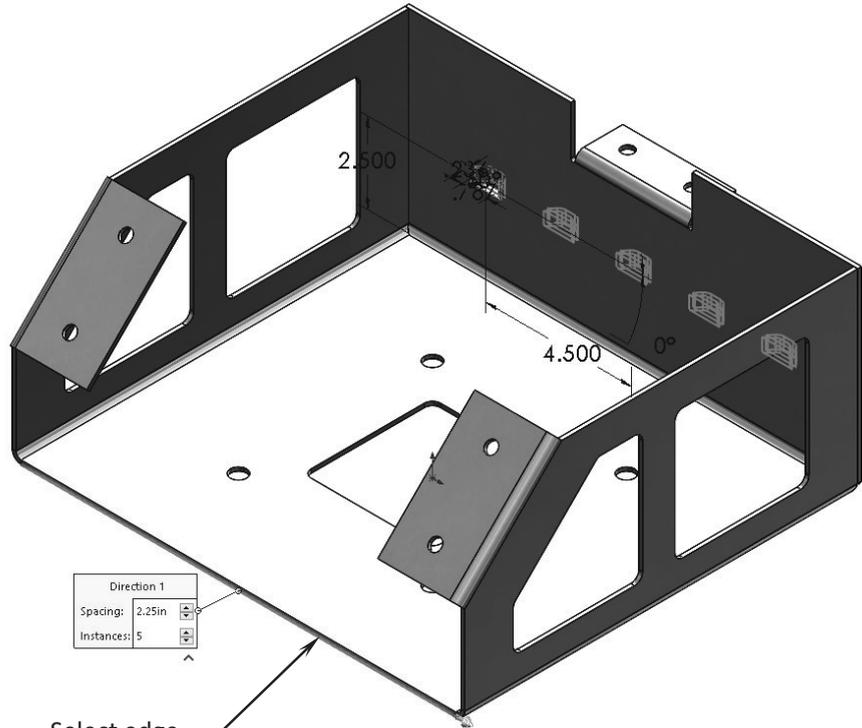
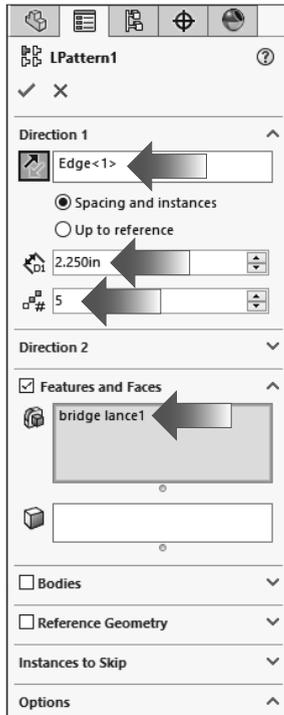
Click **OK**.



14. Patterning the Bridge Lance:

Switch to the **Features** tab and click **Linear Pattern**.

For Direction 1, select the horizontal edge as indicated in the image below.



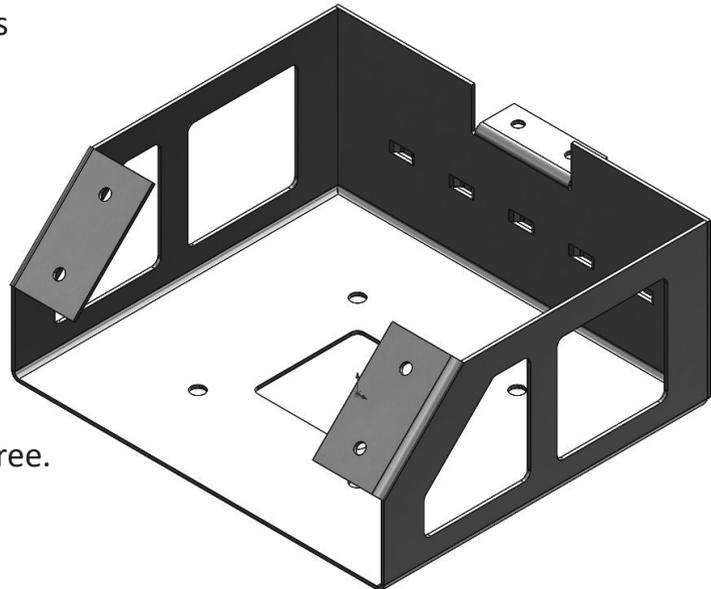
Select edge
for Direction

Click Spacing and Instances and enter **2.250in**.

For Number of Instances, enter: **5**.

For Features to Pattern, select the 1st **Bridge-Lance** and the **Pattern1**, either from the graphics area or from the Feature tree.

Click **OK**.

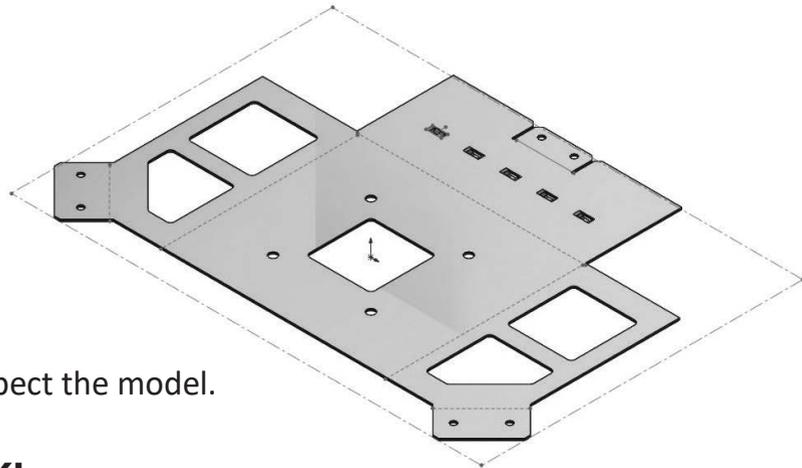
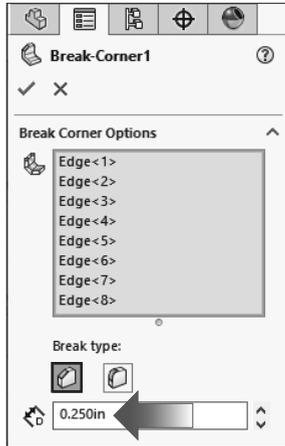
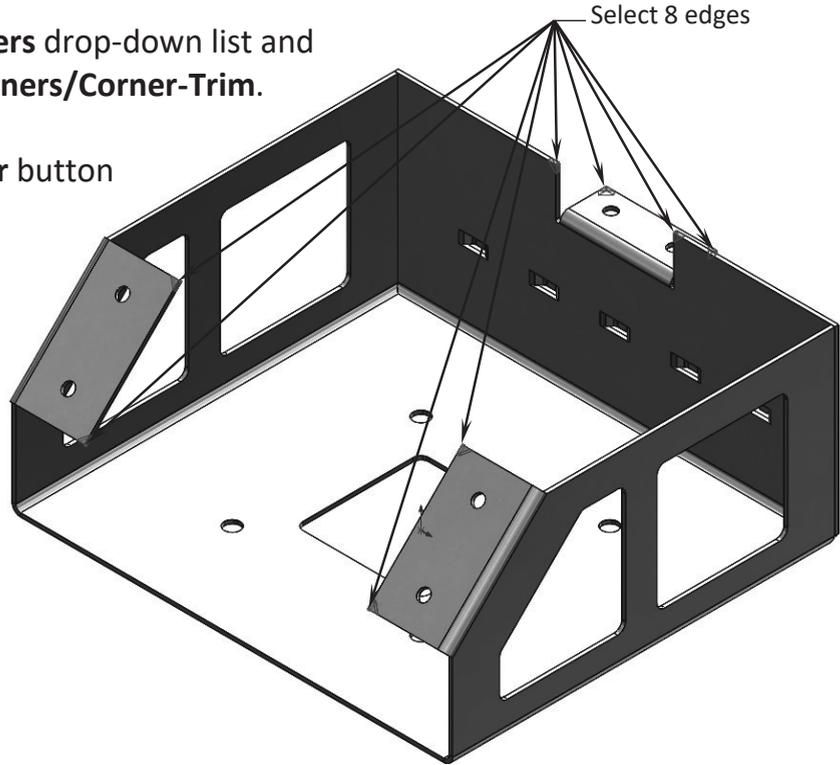


15. Adding chamfers:

Expand the **Corners** drop-down list and select: **Break-Corners/Corner-Trim**.

Click the **Chamfer** button and enter **.250in** for Distance.

Select **8 edges** as shown in the image on the right.



Click **OK**.

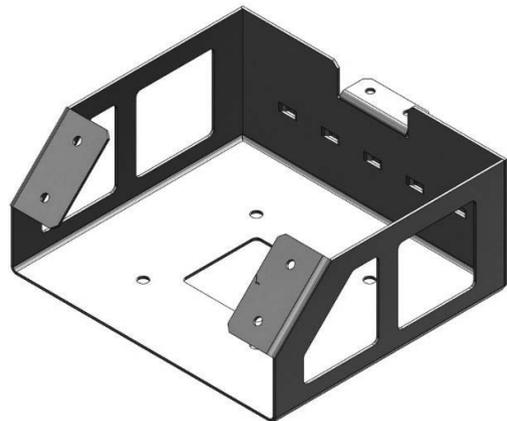
Click **Flatten** to inspect the model.

16. Saving your work:

Select **File, Save As**.

Enter: **Sheet Metal_Exe1_Completed** for the file name.

Click **Save**.

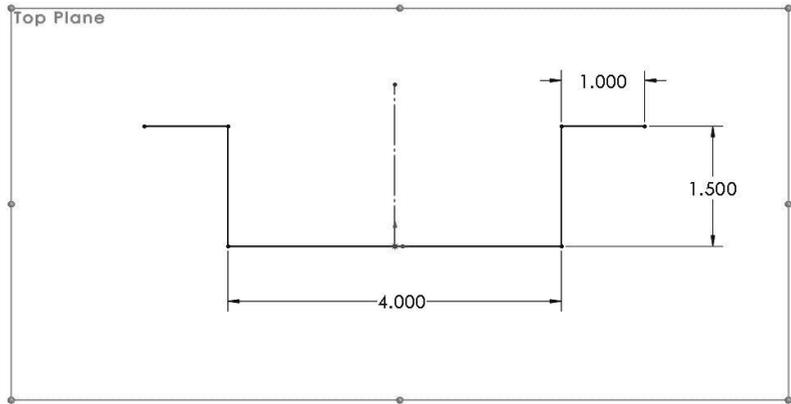


Exercise: Basic Sheet Metal Parts

1. Making the Base Sketch:

Select the **TOP** plane and open a **new sketch**.

Sketch the profile shown on the right. Use the **Mirror** option to keep the entities symmetrical to one another.



Add dimensions to fully define the sketch.

2. Creating a Base Flange:

Click **Base Flange**.

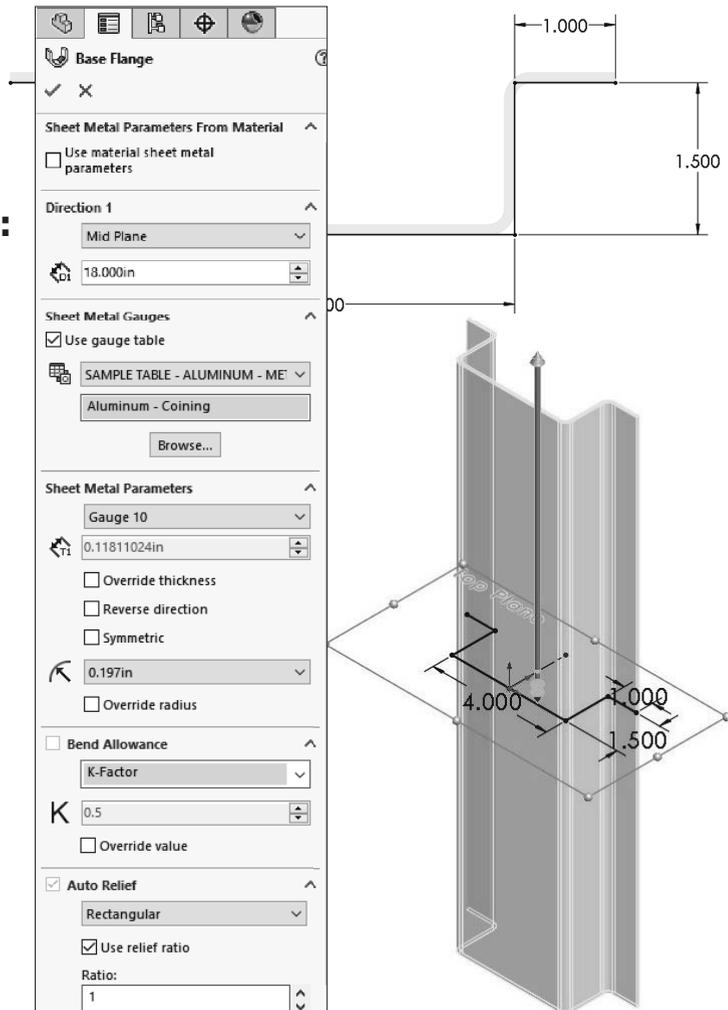
For Direction 1, use **Mid Plane** and **18.00in**.

Enable the **Use Gauge Table** checkbox and select:

Sample Table – **Alum.**
Gauge **10**
Bend Radius **.197in**

K-Factor: **0.5**

Auto Relief **Enabled**
Rectangular Ratio **1**
Click **OK**.



3. Adding the Hole Wizard holes:

Switch to the **Features** tab and click **Hole Wizard**.

For Hole Type, select **Hole**.

Standard:
ANSI Inch.

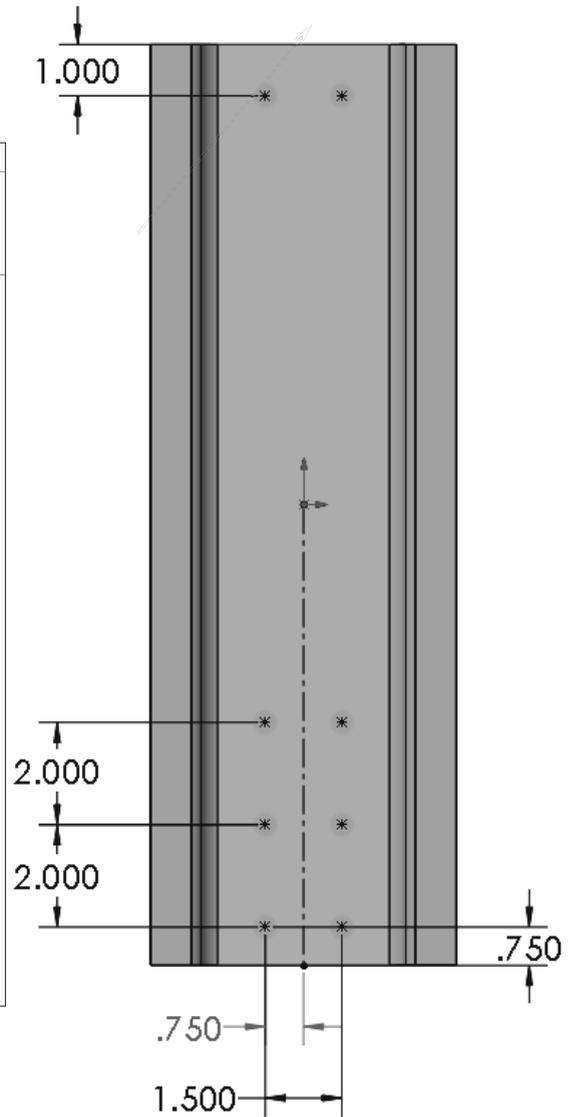
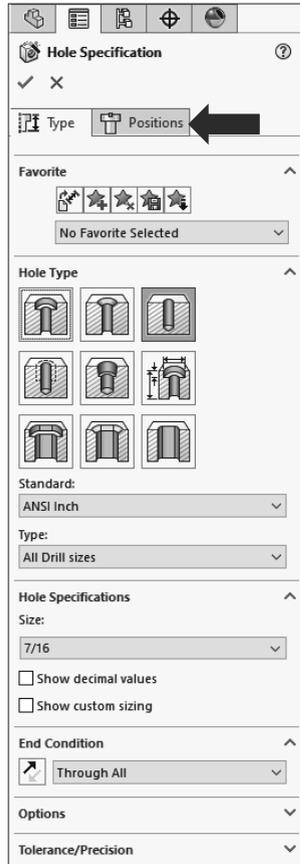
Type: **All
Drill Sizes.**

Size: **7/16.**

End Condition:
Through All.

Click the
Position tab
(arrow).

Select the
center-face
to rotate the view normal to you.



Place 4 dots on one side and sketch a **Centerline** to mirror the dots.
(Each dot represents the center of a hole.)

Add the dimensions shown in the image to fully define the sketch.

Click **OK**.

4. Adding the Slotted-Holes (Clearance Holes):

Click **Hole Wizard** once again.

For Hole Type, select **Slot**.

Standard: **ANSI Inch**.

Type: **All Drill Sizes**.

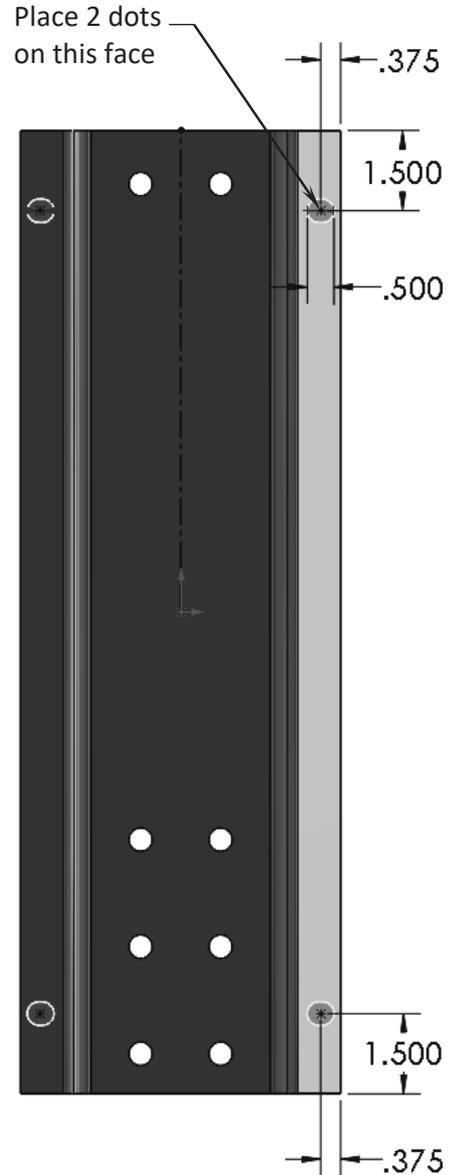
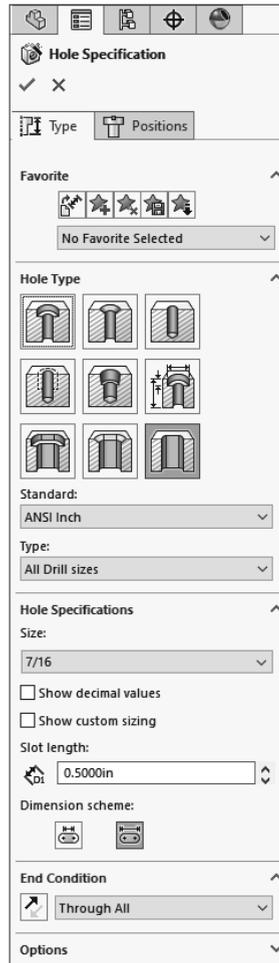
Size: **7/16**.

Slot Length: **.500in**.

Dimension Scheme:
Arc Tangent.

End Condition:
Through All.

Click the **Position** tab
(arrow).



Click the side-face to rotate the view normal to you.

Place **2 slotted holes** on one side and sketch a **Centerline** to mirror the slotted holes.

Click **OK**.

5. Adding the corner fillets:

Switch to the **Features** tab and **Fillet**.

For Fillet Type, use the default **Constant Size** option.

For Radius, enter **.250in**.

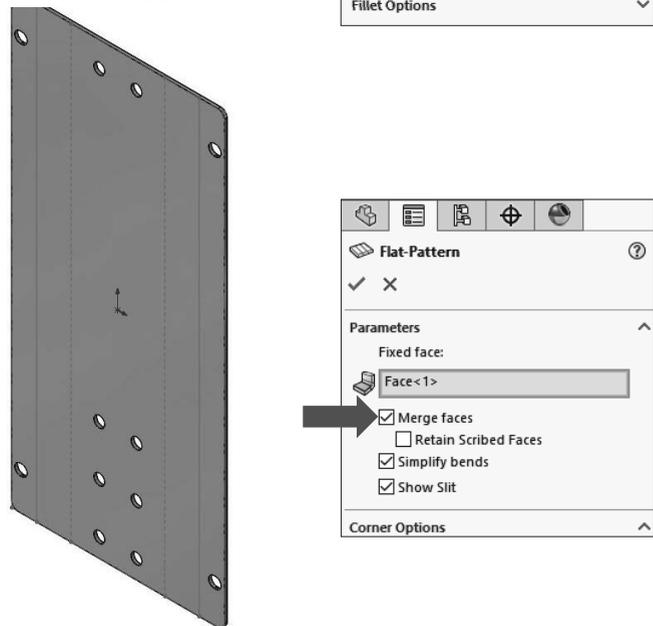
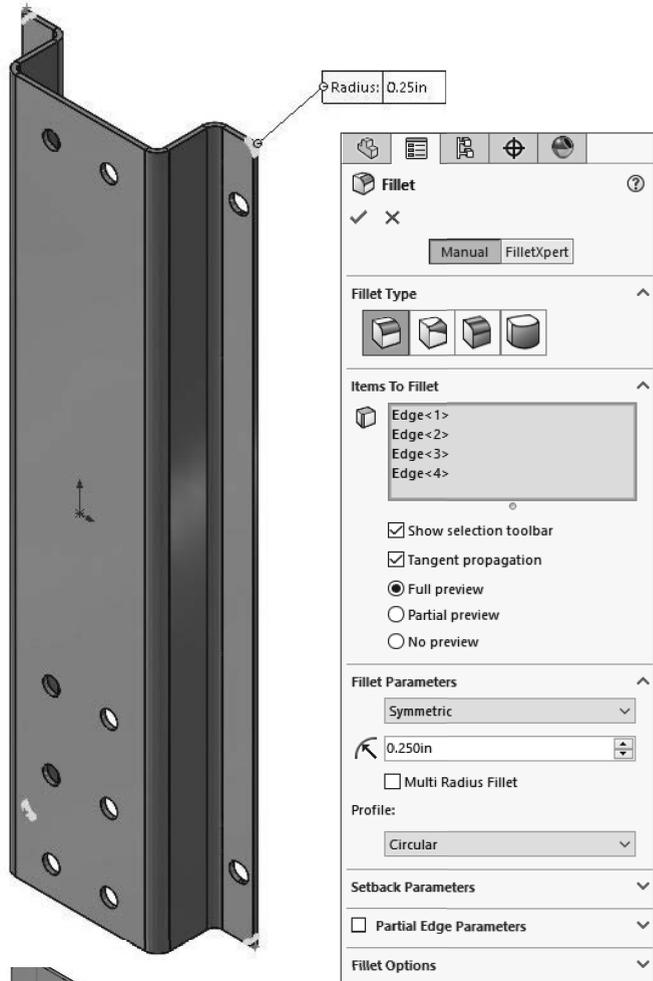
For Items to Fillet, select the 4 edges at the 4 corners of the part.

Click **OK**.

Switch to the **Sheet Metal** tab and click: **Flatten**.

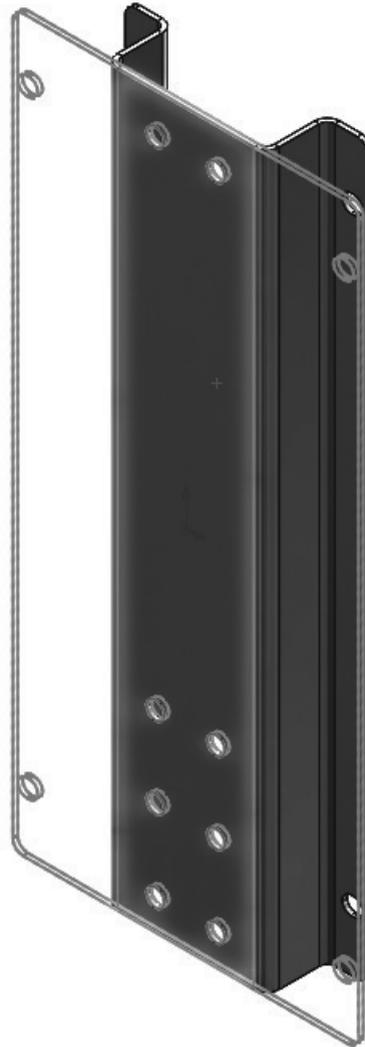
The part should show the flat pattern without any errors on the FeatureManager tree.

The Bend-Lines can be toggled on/off by editing the **Flat-Pattern1** and enabling the **Merge-Faces** checkbox.



Right-click anywhere on the part and select:
Toggle Flat Pattern.

The Flat Pattern is projected onto the Fold Pattern. This is one of the quicker ways to view the flat pattern of a sheet metal part.



6. Saving your work:

Select: **File, Save As.**

Enter: **Basic Sheet Metal Parts.sldprt** for the file name.

Click **Save.**