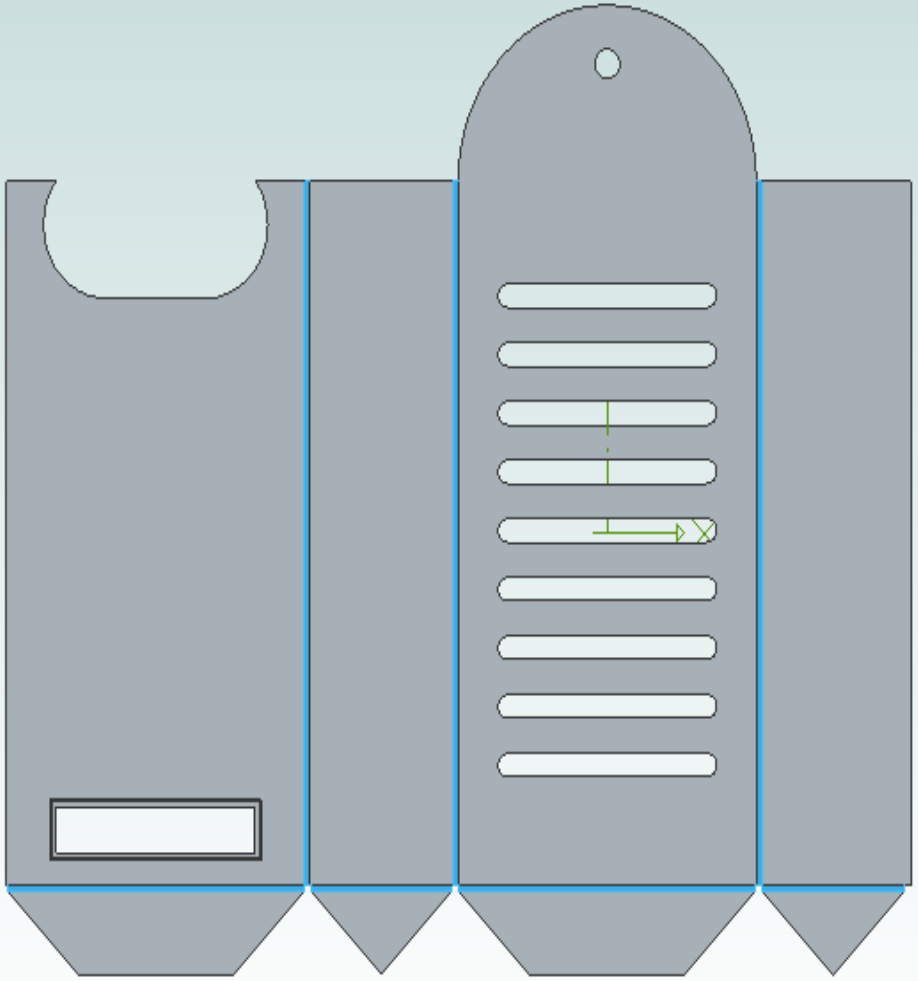
Alibre Design Exercise Manual

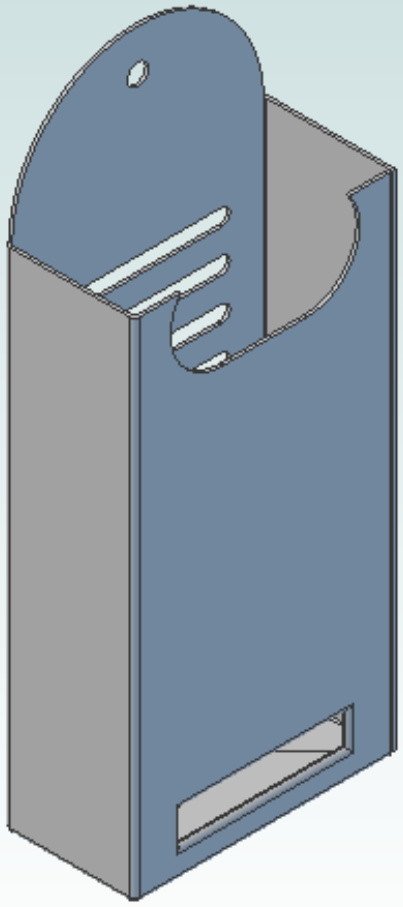
Introduction to Sheet Metal Design

Copyrights

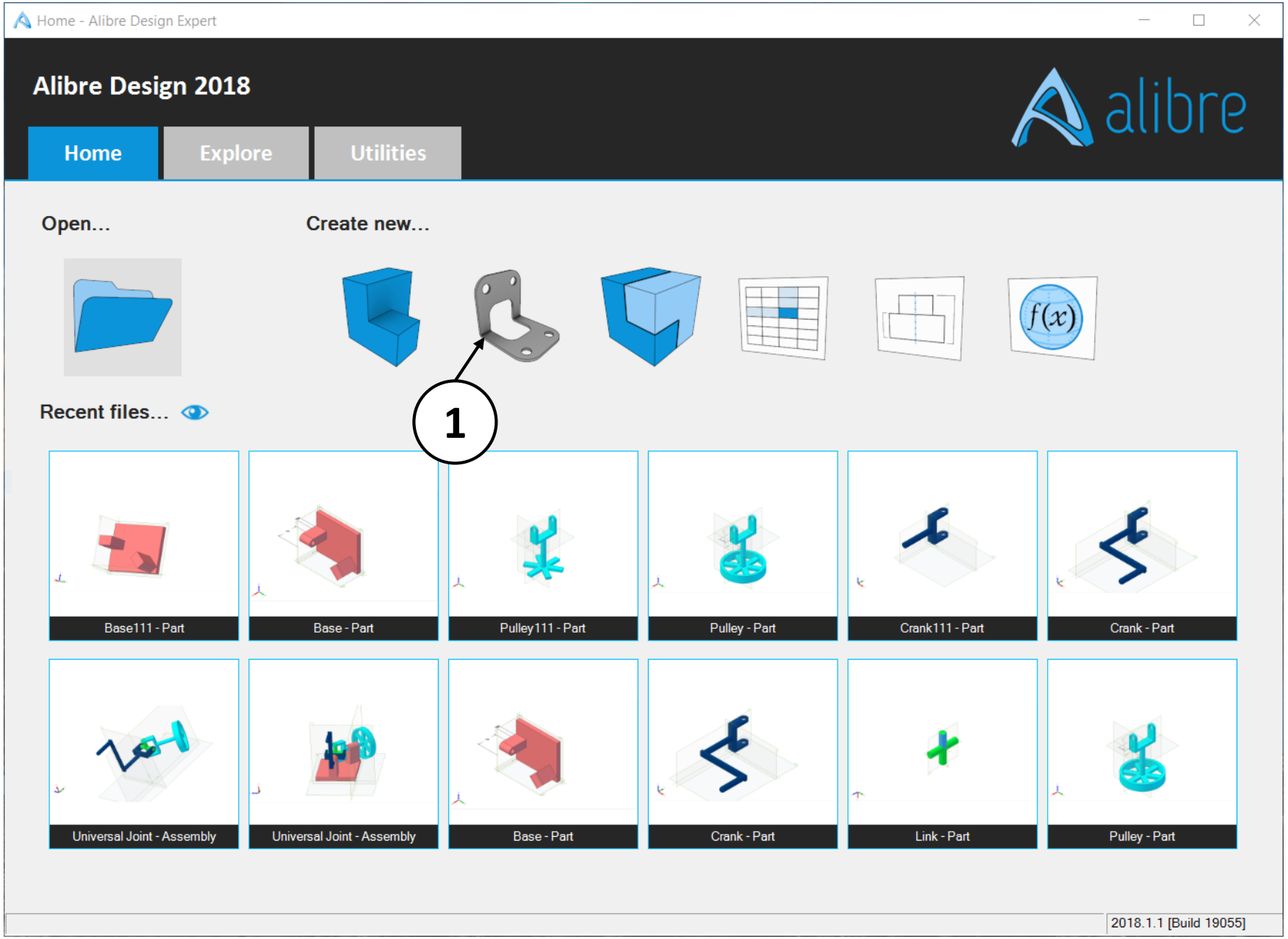
Alibre, LLC
1750 N Collins Blvd
Ste. 212
Richardson, TX 75080
U.S.A

www.alibre.com

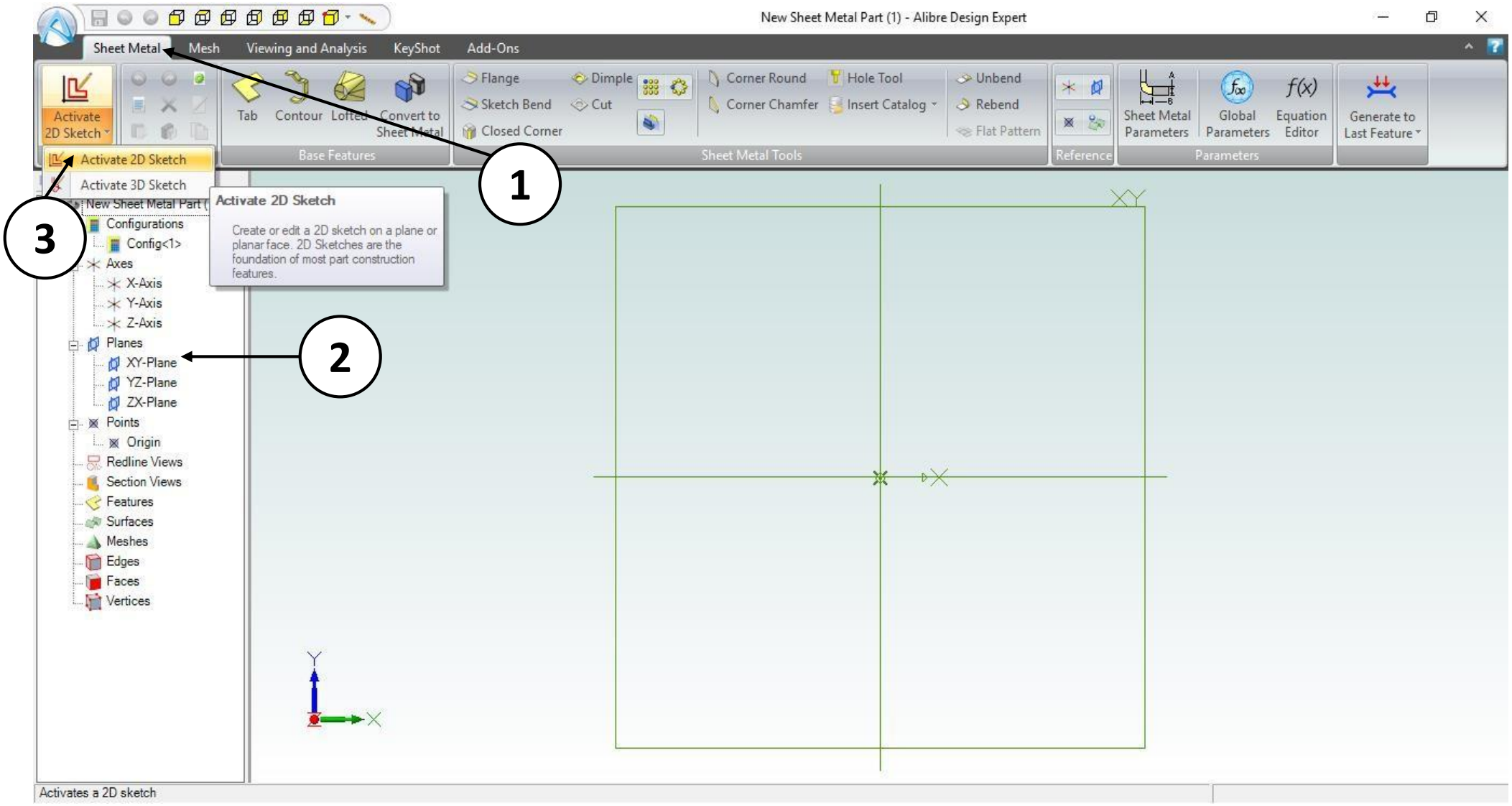




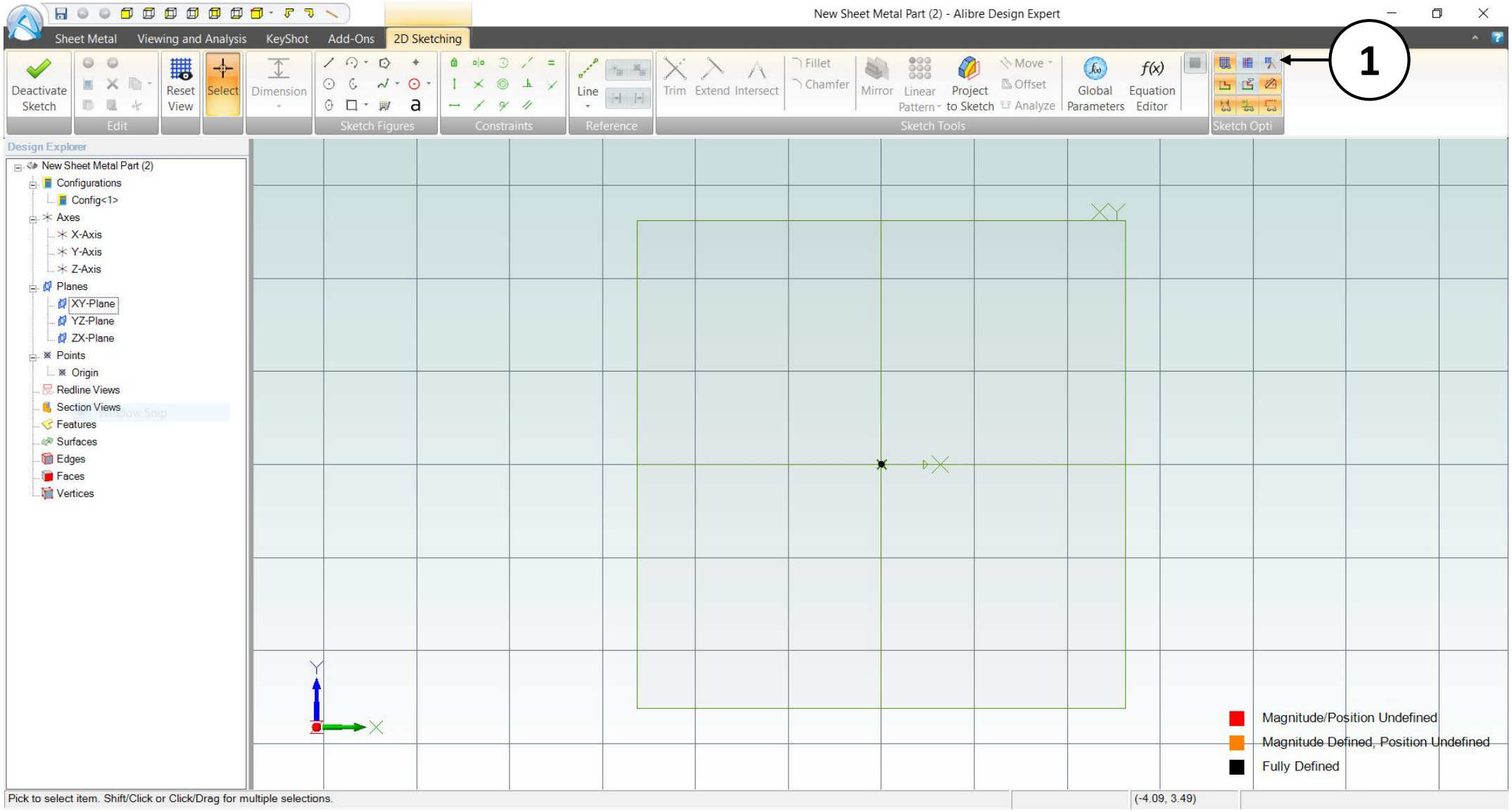
Creating a Tool Holder



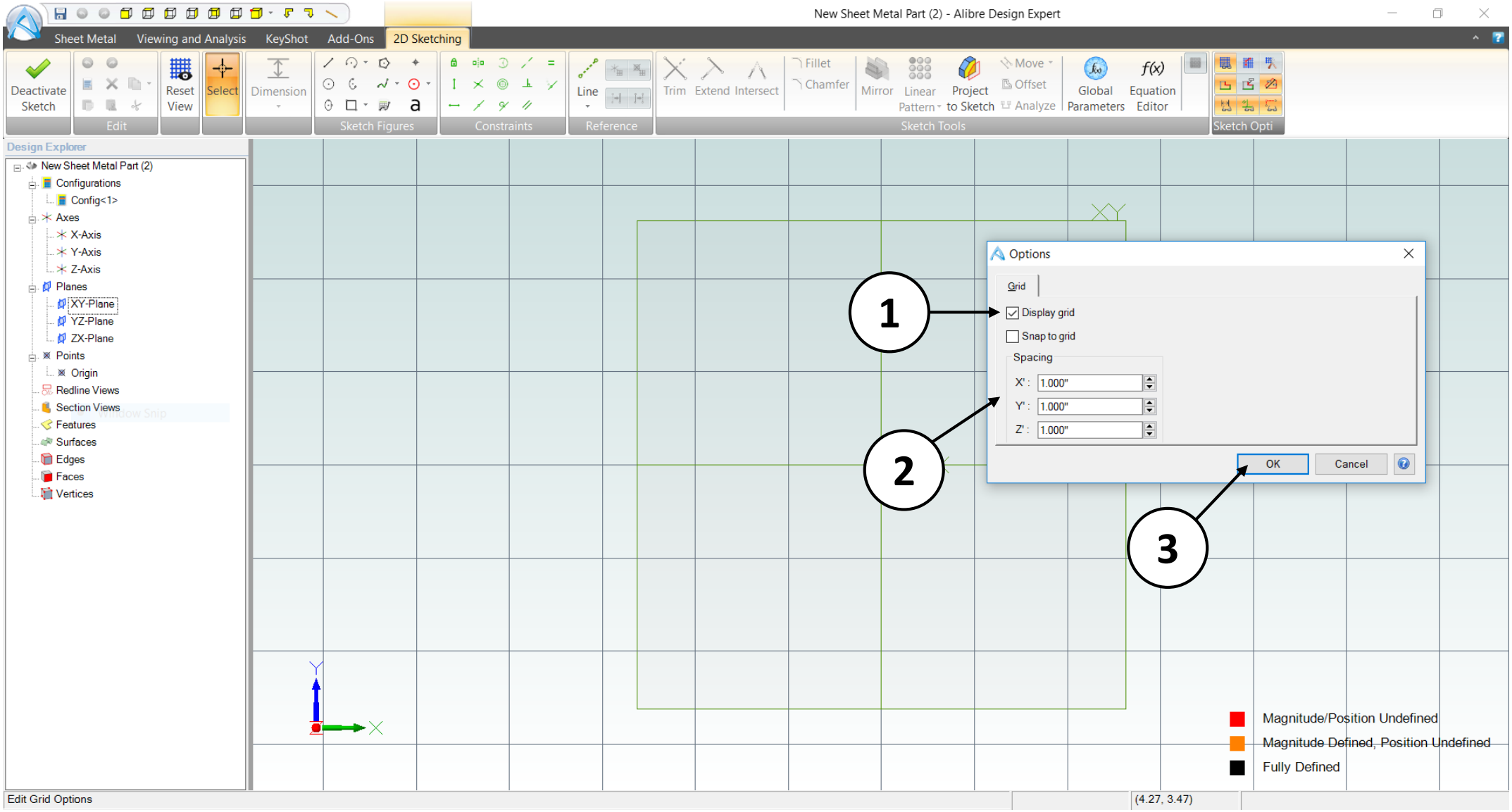
1. Open a new **Sheet Metal Part** workspace from Alibre Design's **Home** window.



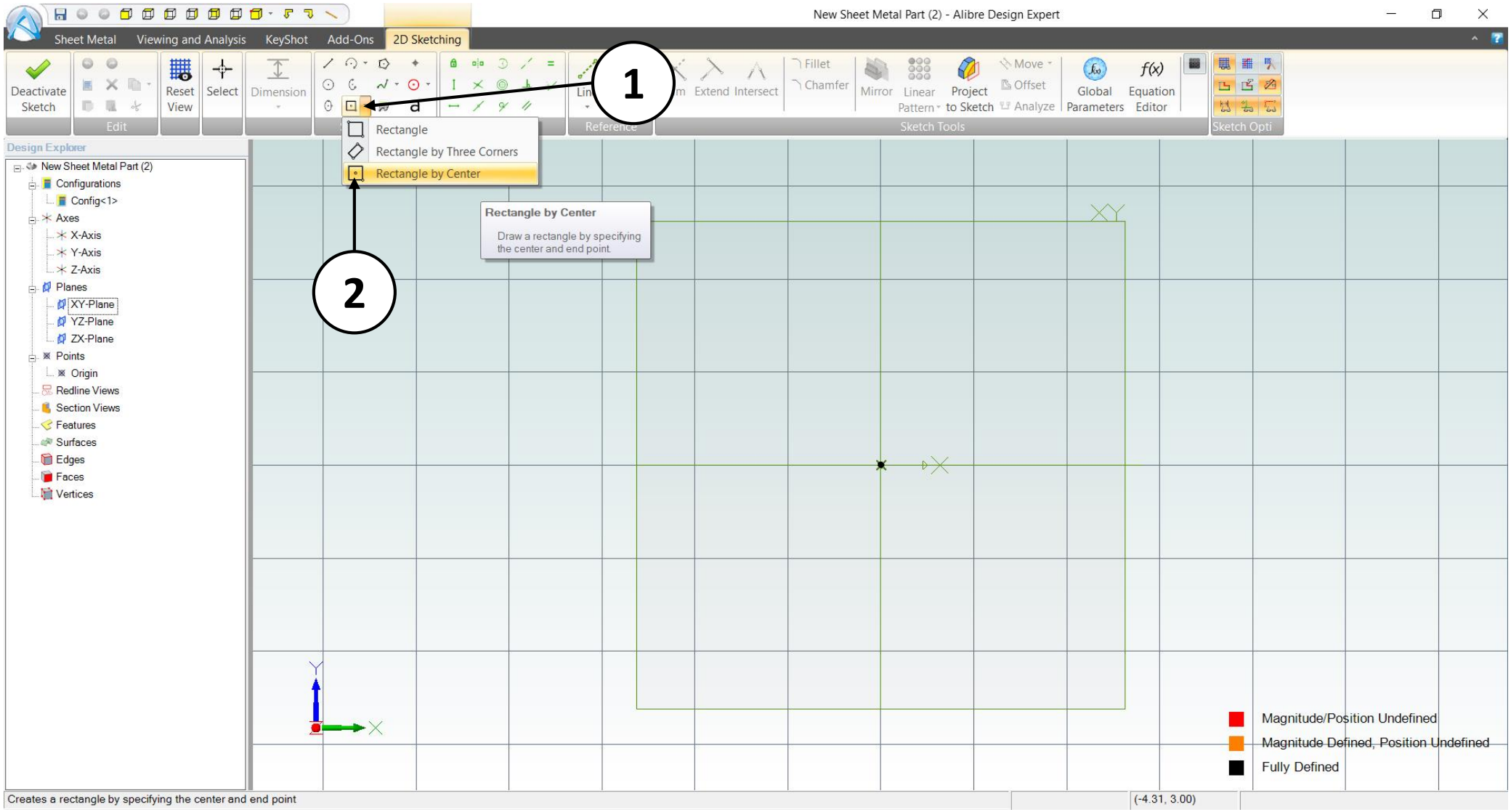
1. Click on the **Sheet Metal** tab in the ribbon.
2. Click on the **XY Plane** in the **Design Explorer**.
3. Click on the **Activate 2D Sketch** icon.



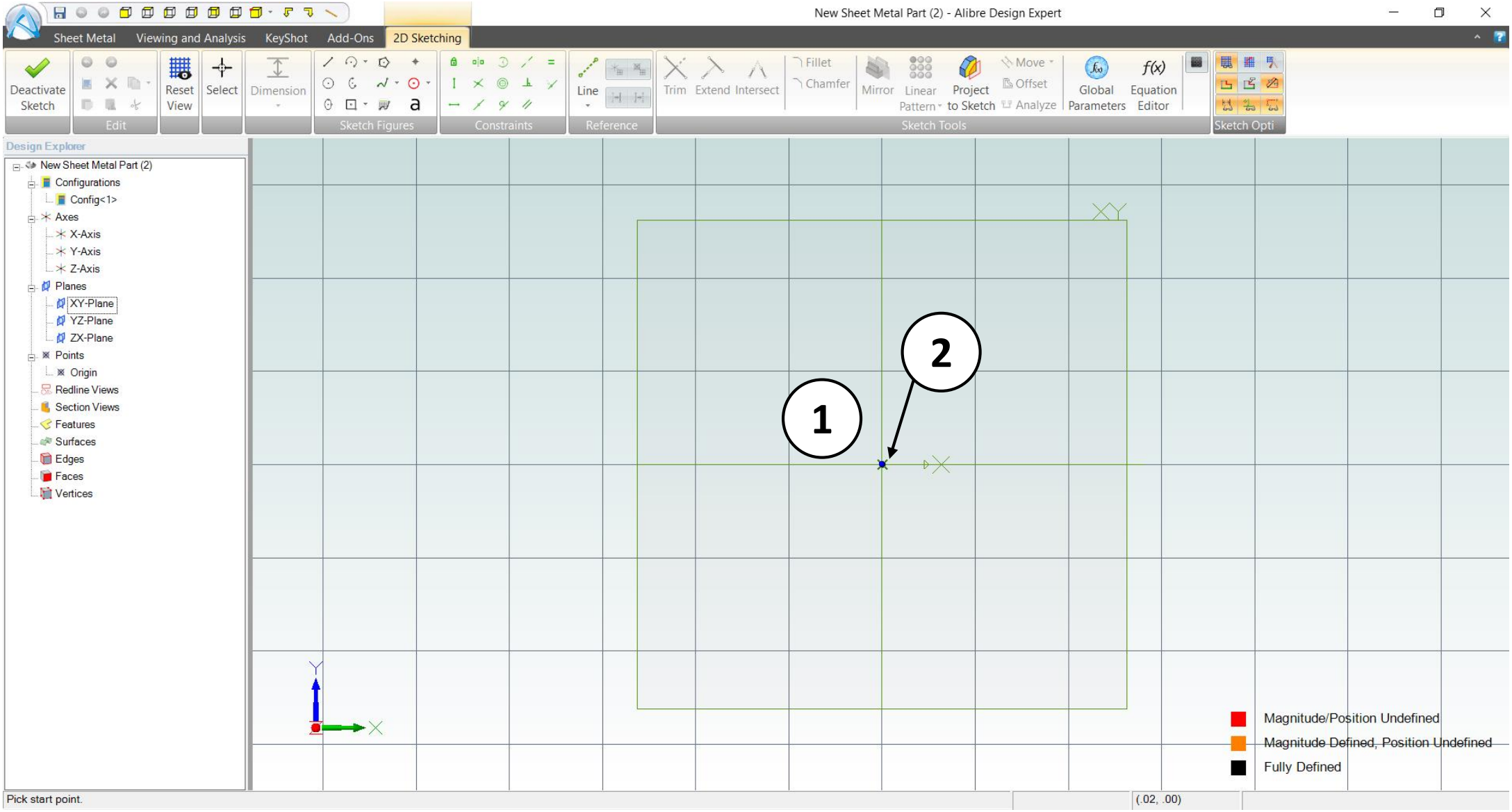
1. Select the **Grid** options.



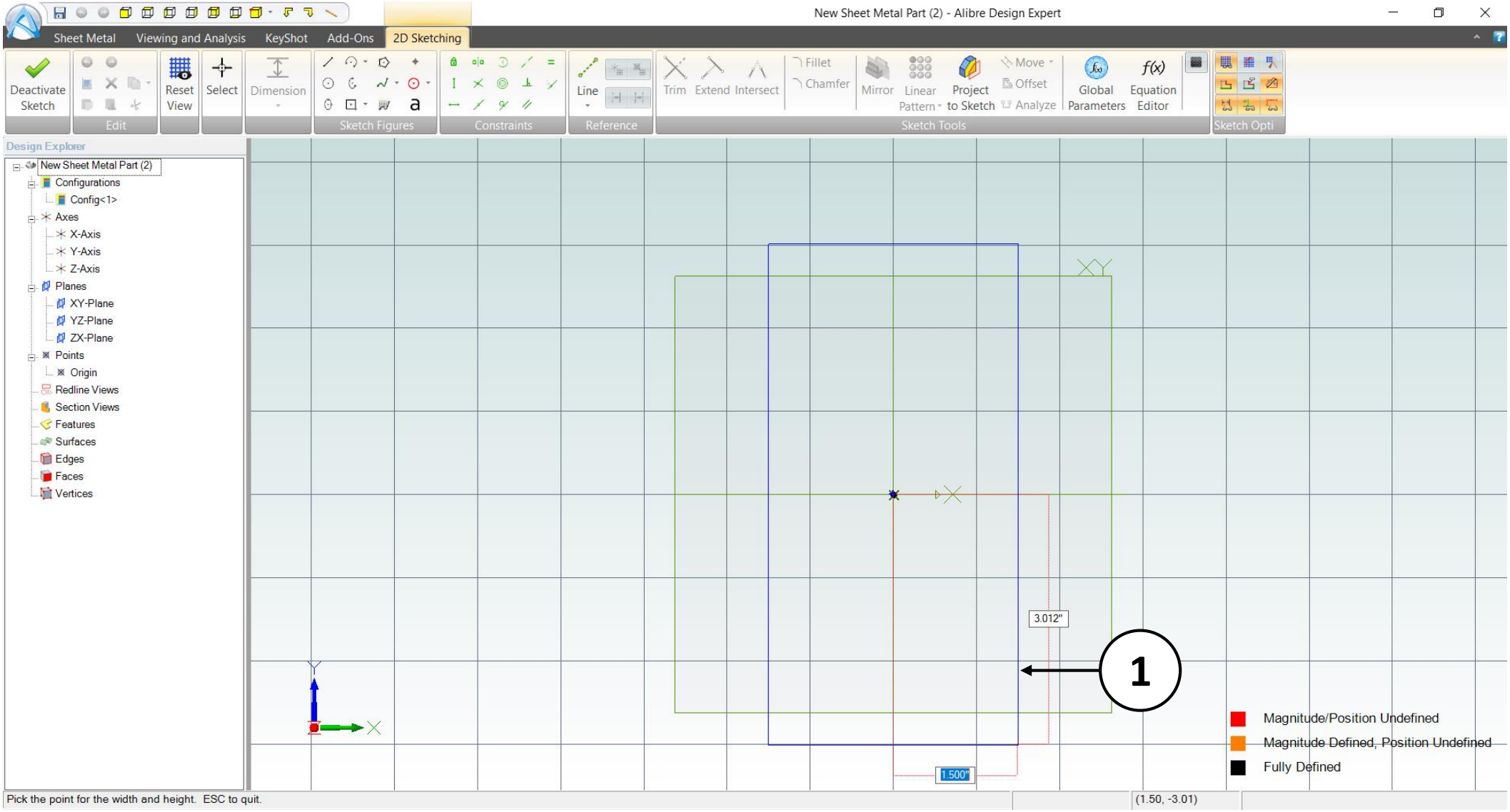
1. Enable the **Display Grid** option if it is unchecked.
2. If necessary, change the values to .500 or .25 inch for X, Y, and Z.
3. Press the **OK** button.



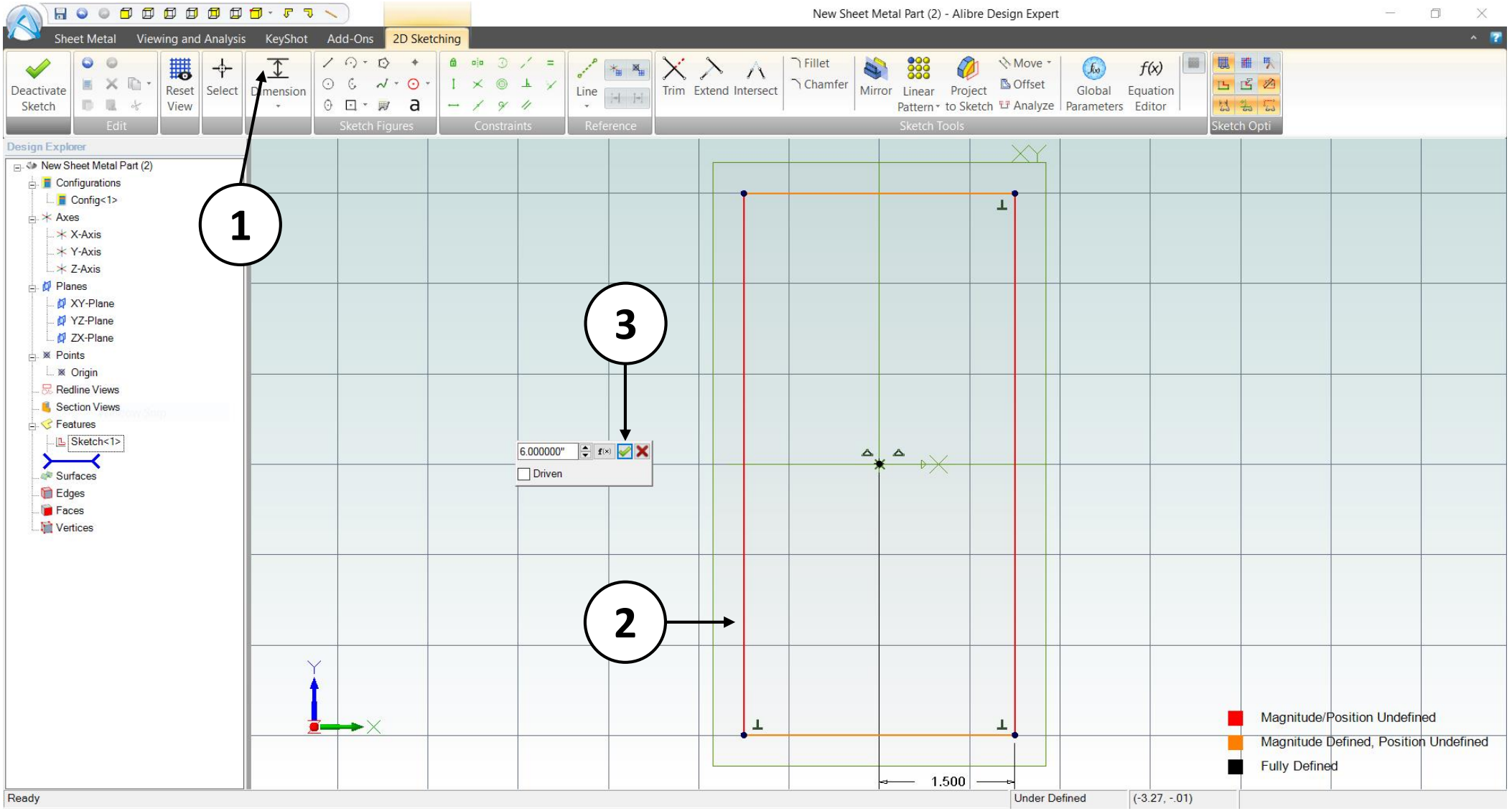
1. Click on the dropdown menu of **Rectangle** tool on the **Sketch Figures** tab in the ribbon.
2. Select the **Rectangle by Center** from the List.



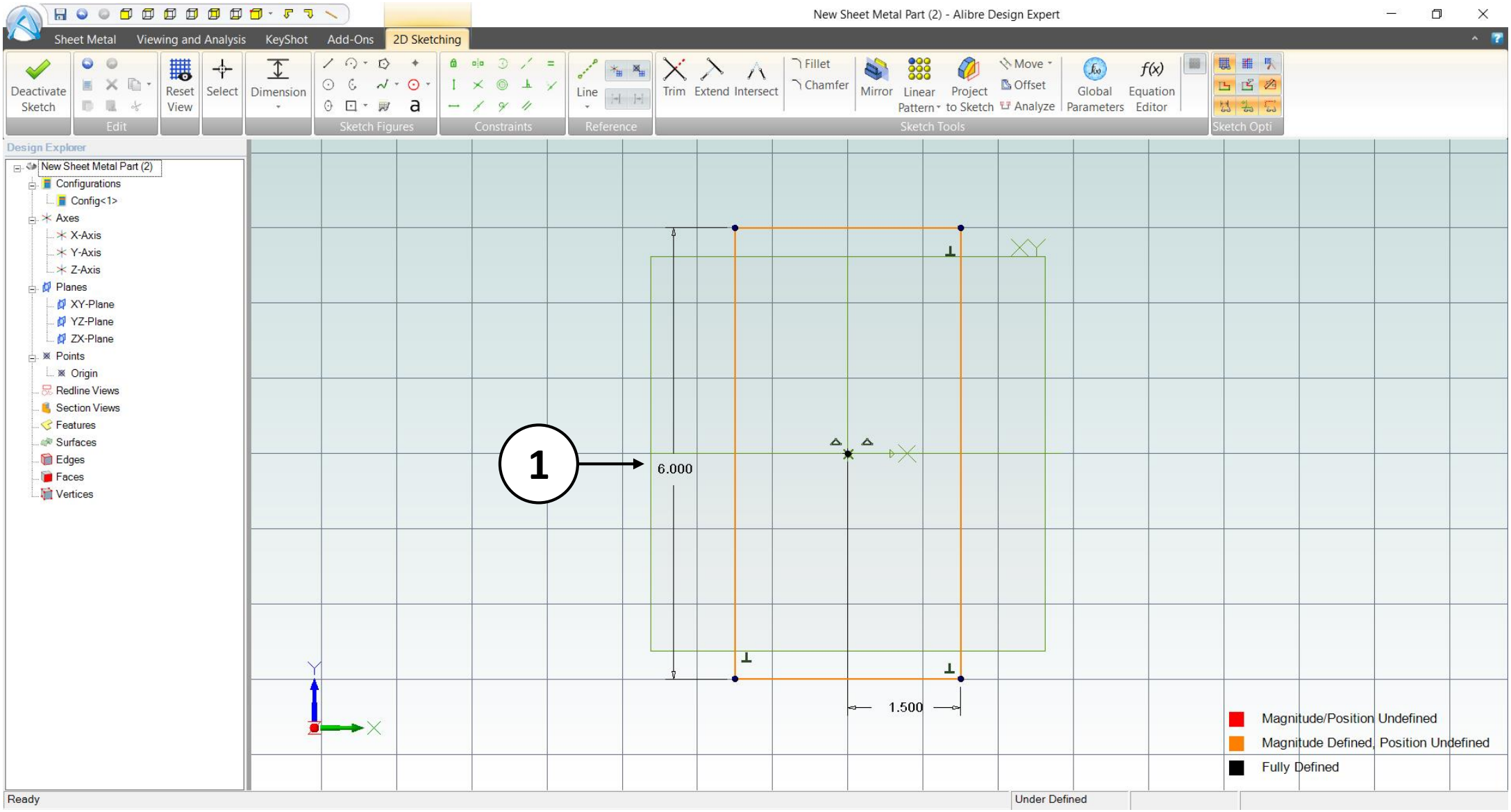
1. Position the mouse cursor near the **Origin** point. Roll the mouse wheel forward to zoom in. The view of the work area should be similar to the screen capture.
2. Position the mouse cursor over the **Origin** point. The point's color and cursor tool tip changes confirm that the mouse cursor is positioned properly. *CLICK AND RELEASE* to create the center point of the Rectangle.



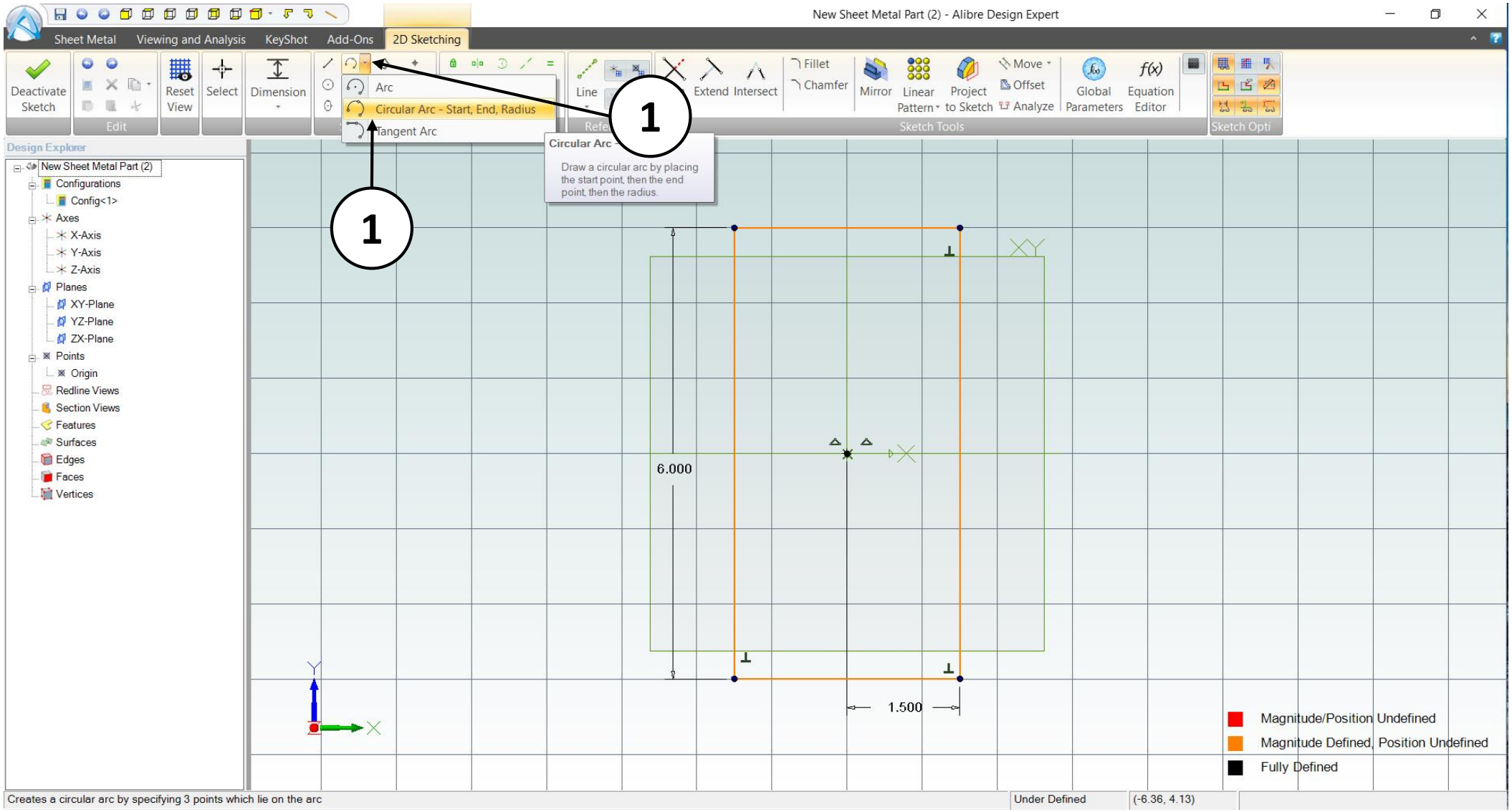
1. Move the mouse cursor horizontally to the right, input a dimension of **1.500 Inches** for the horizontal distance from the center of the rectangle to its edge, and then hit Enter.



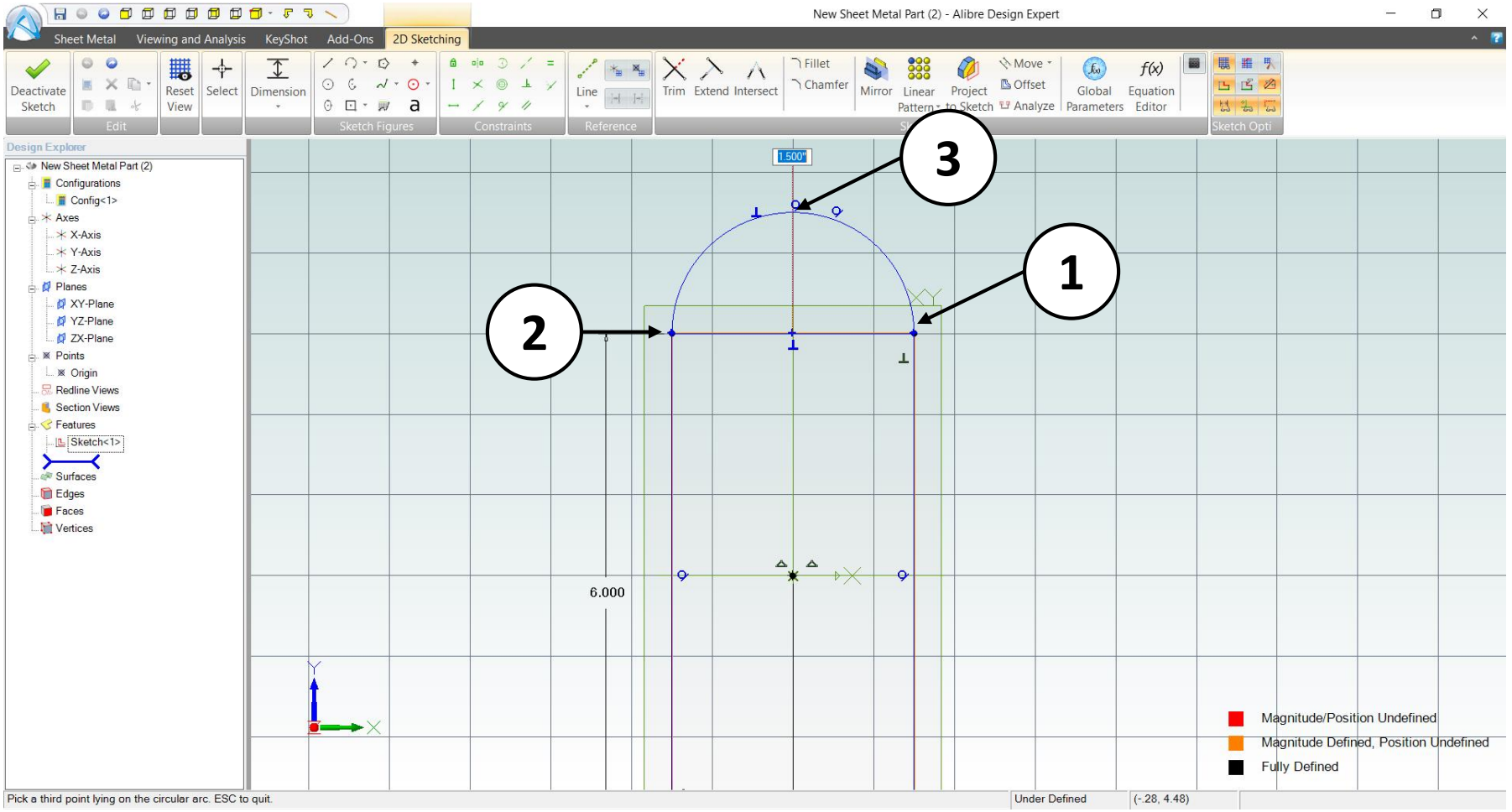
1. Click on **Dimension**.
2. Select the left or right vertical line of Rectangle, then click on the workspace location where you want to place the dimension and input a value of **6.000”**.
3. Click on the green checkmark or press the **Enter** key on the keyboard.



1. Completed rectangle view of the work area should be similar to the image above.



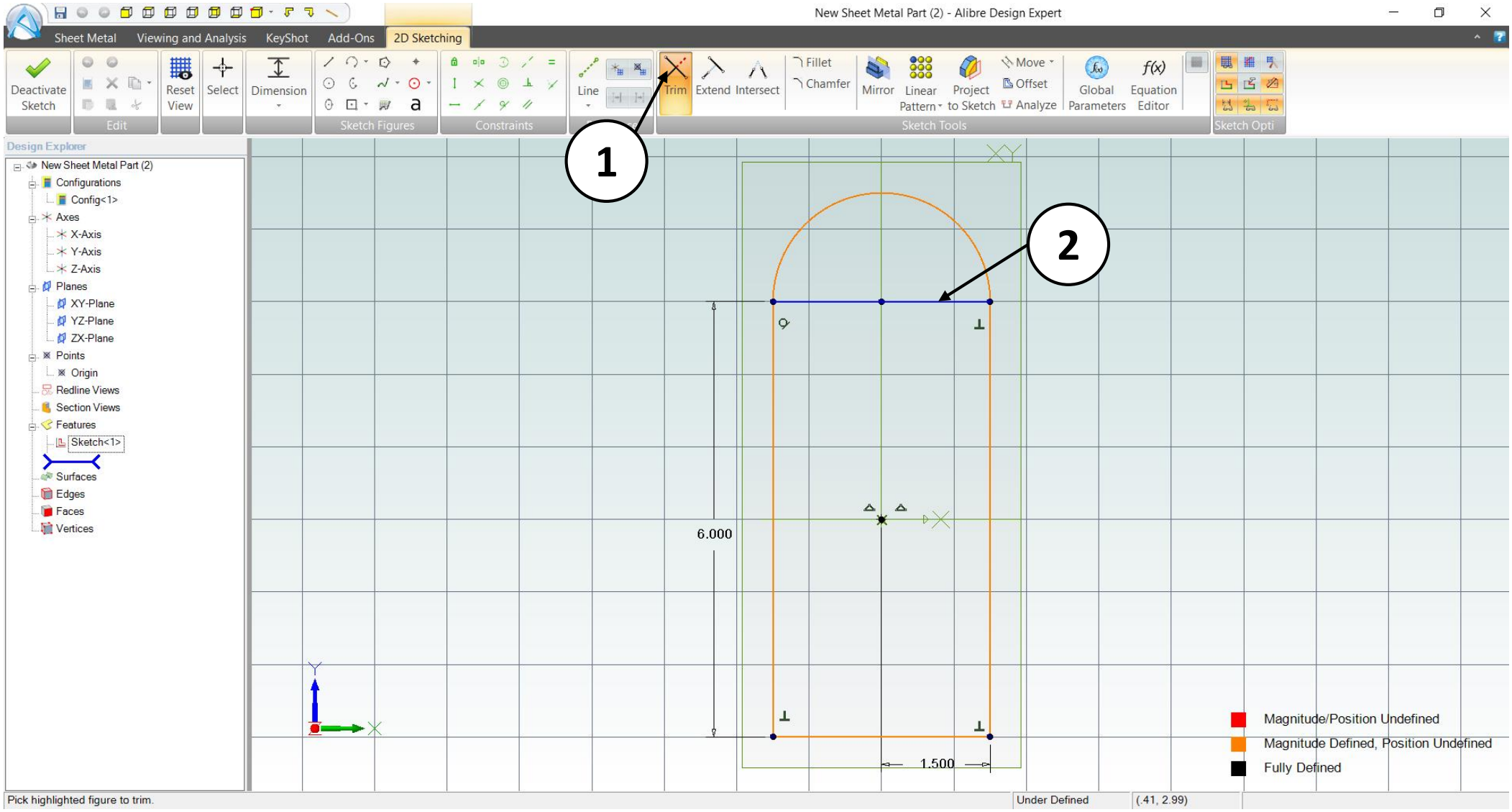
1. Click on the dropdown menu of **Circular Arc** tool on the **Sketch Figures** tab in the ribbon.
2. Select the **Circular Arc-Start, End, Radius** option from the List.



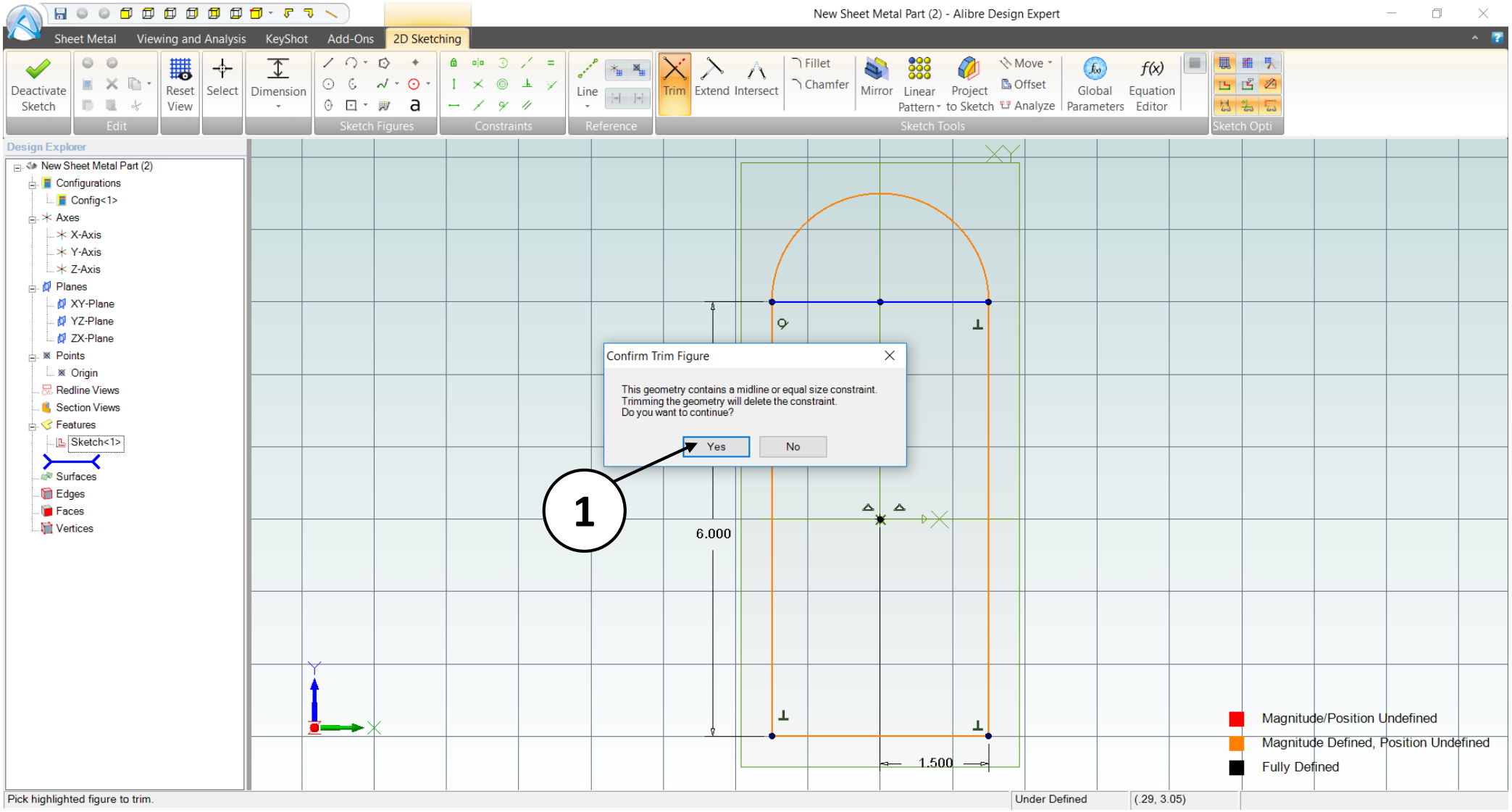
1. Move the mouse cursor over the top-right sketch node on the rectangular sketch **(1.50” to the right of the origin & 3.00” above the Origin). Click** and release to create the Start point of the Arc.

2. Move the mouse cursor over the top-left sketch node on the rectangular sketch**. Click** and release to create the End point of the Arc.

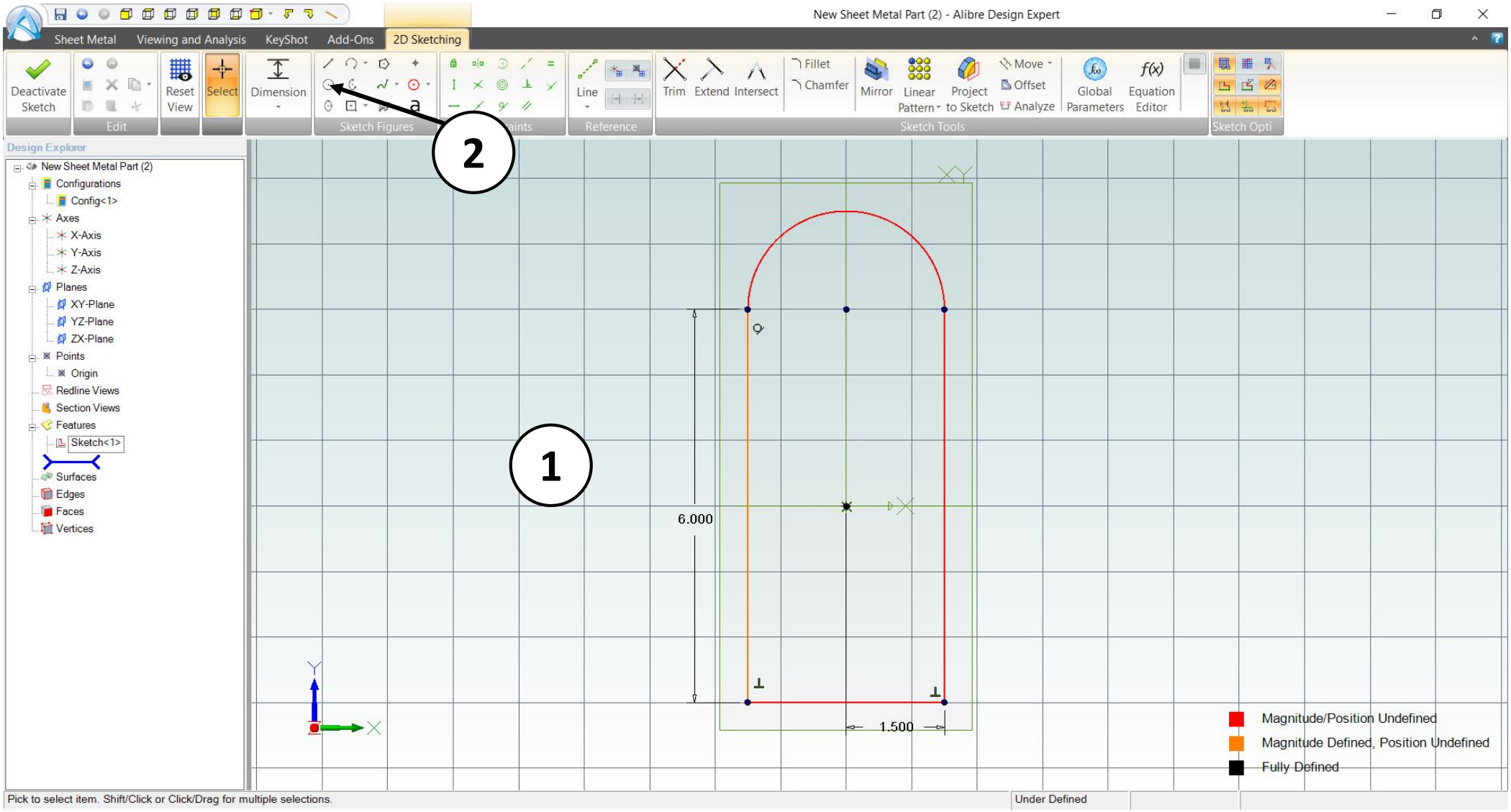
3. Move the mouse cursor in an upward direction until the tangential constraint symbols appear. **Click** and release to complete the Arc.



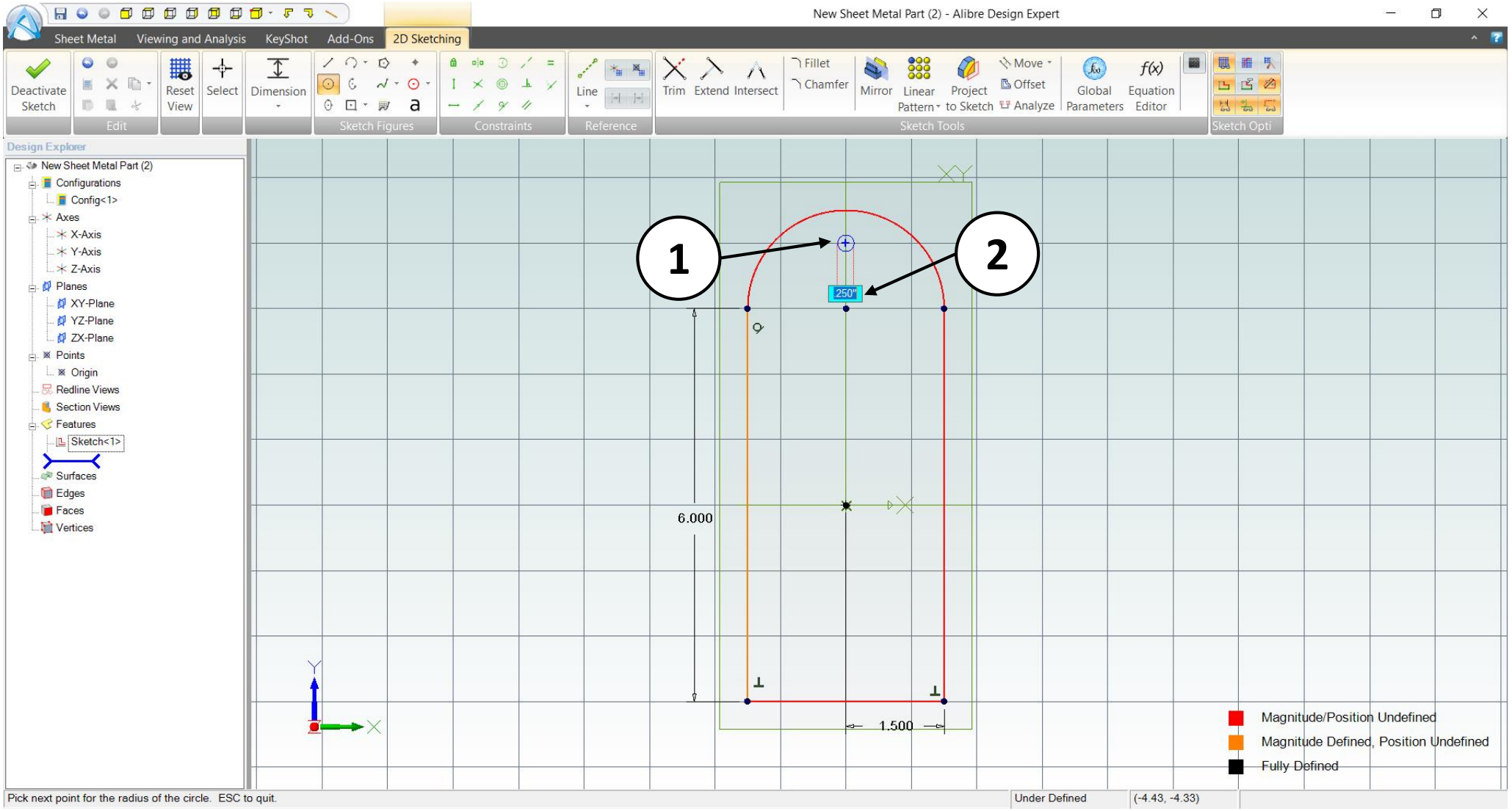
1. Click on the **Trim** tool on the **Sketch Tools** tab in the ribbon.
2. Click on the top horizontal sketch line to execute the **Trim**.



1. Click **Yes** on the pop-up window to complete the process.

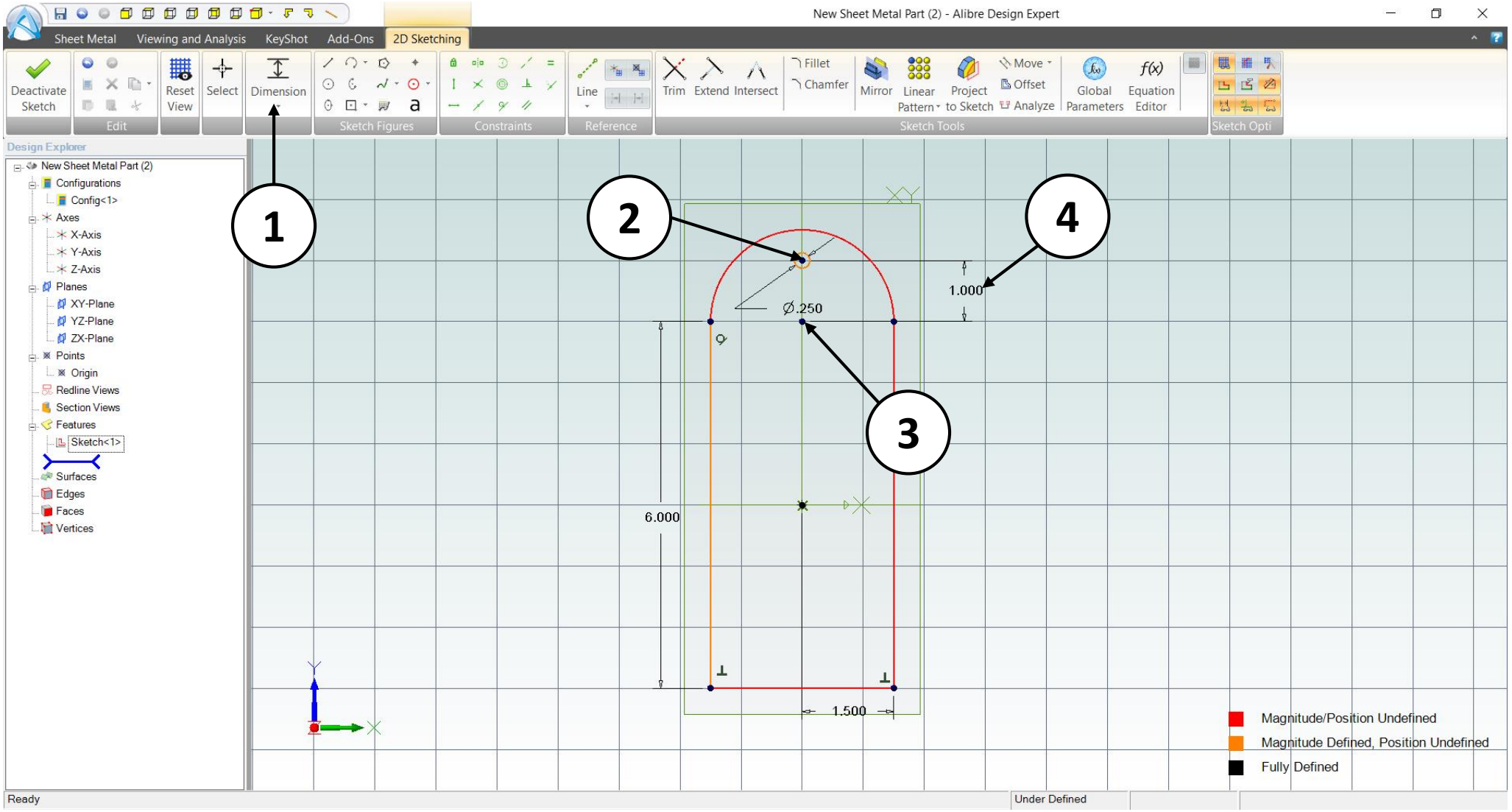


1. Completed Sketch view of the work area should be similar to the screen capture.
2. Click on the **Circle** tool on the **Sketch Figures** tab in the ribbon.

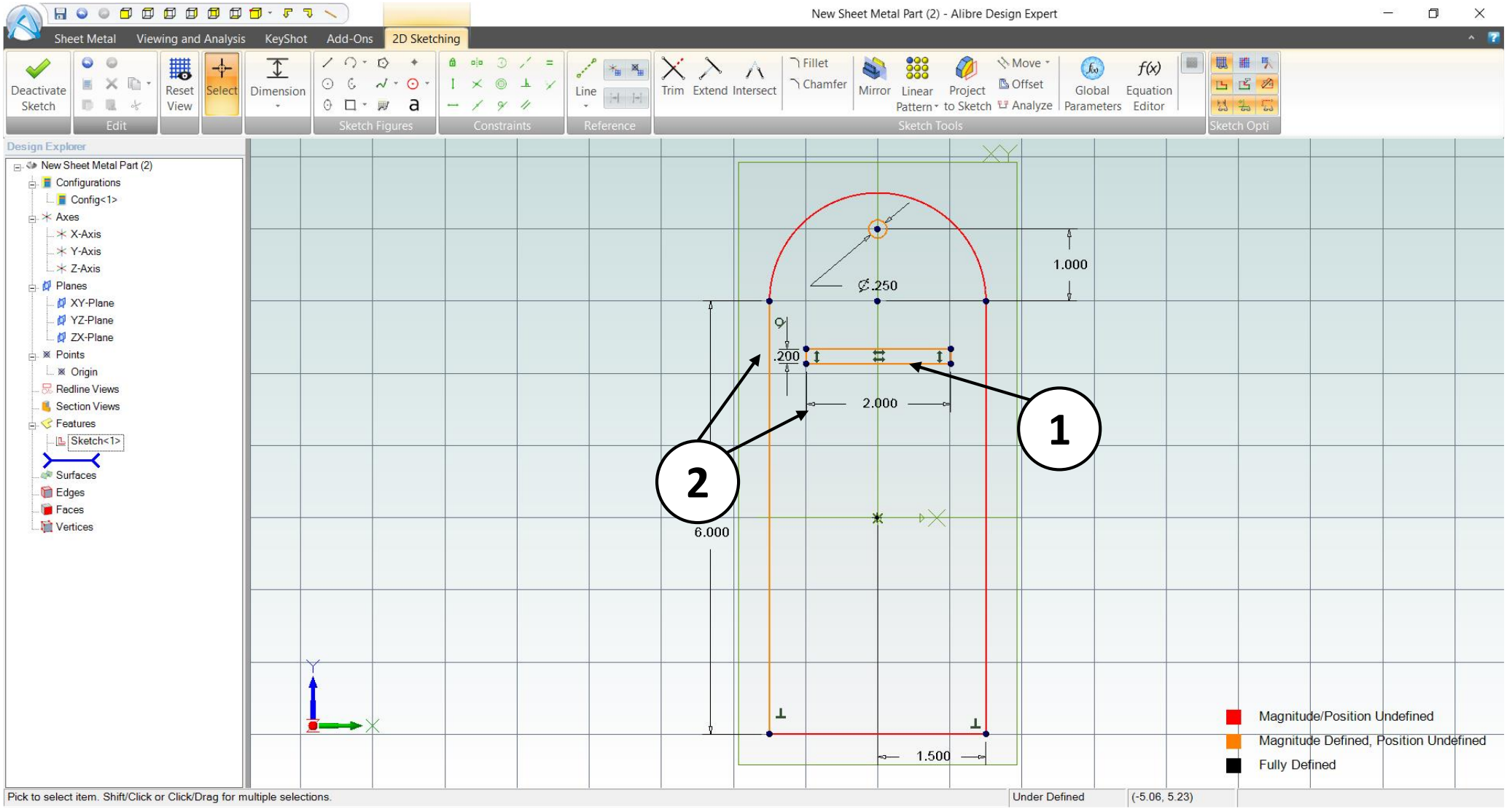


1. Move the mouse cursor close to the position shown **(4.00” over the Origin). Click** and release to create the Center point of the **Circle**.

2. Enter a value of 0.25” for the circle’s diameter, and then hit Enter on your keyboard to apply the value.

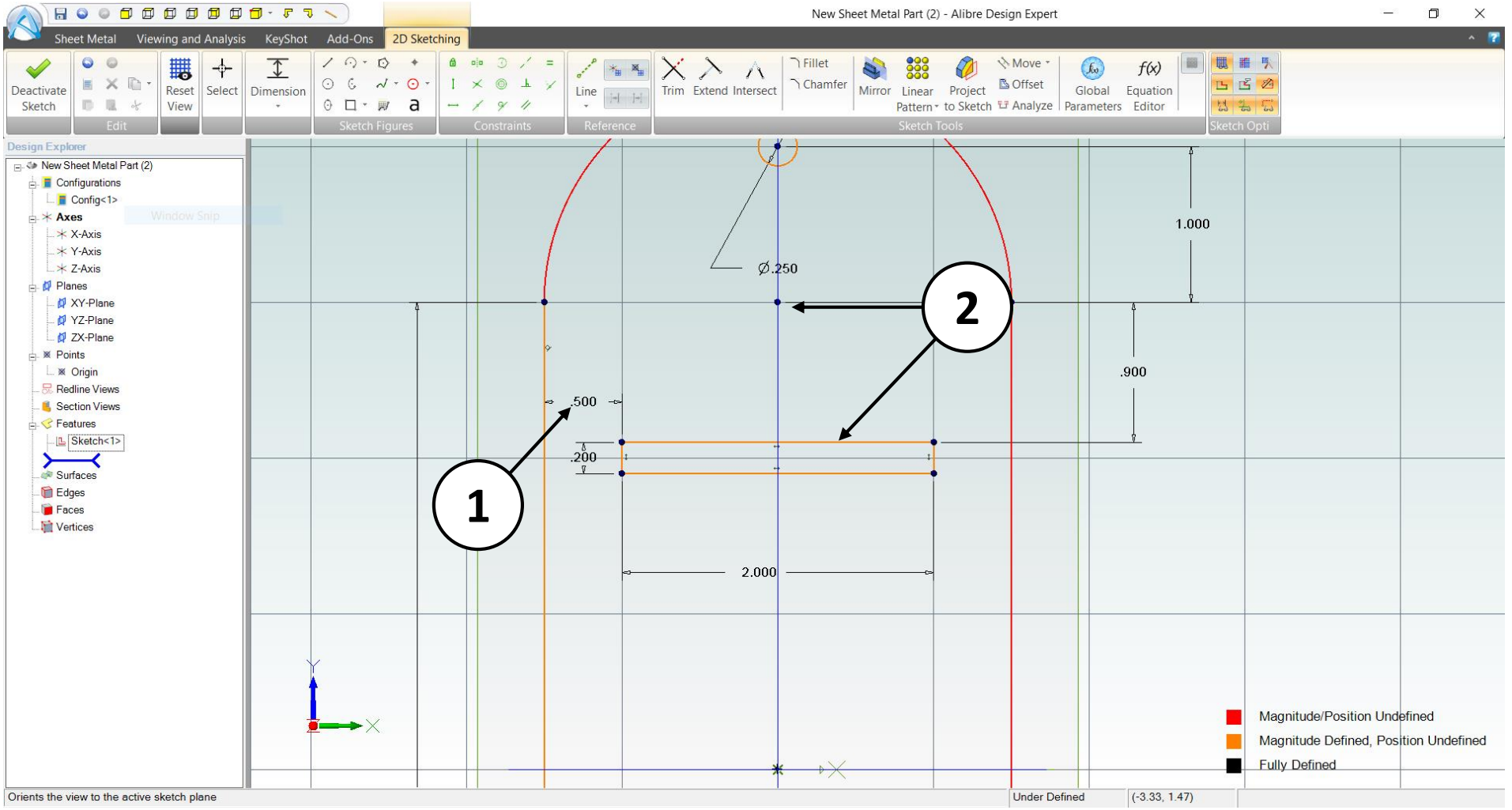


1. Click on **Dimension**.
2. Click on the center point of the circular sketch you just created.
3. Click on the **node** point referenced by Callout “3” in the image above.
4. Enter a dimension of 1.00” and hit Enter.



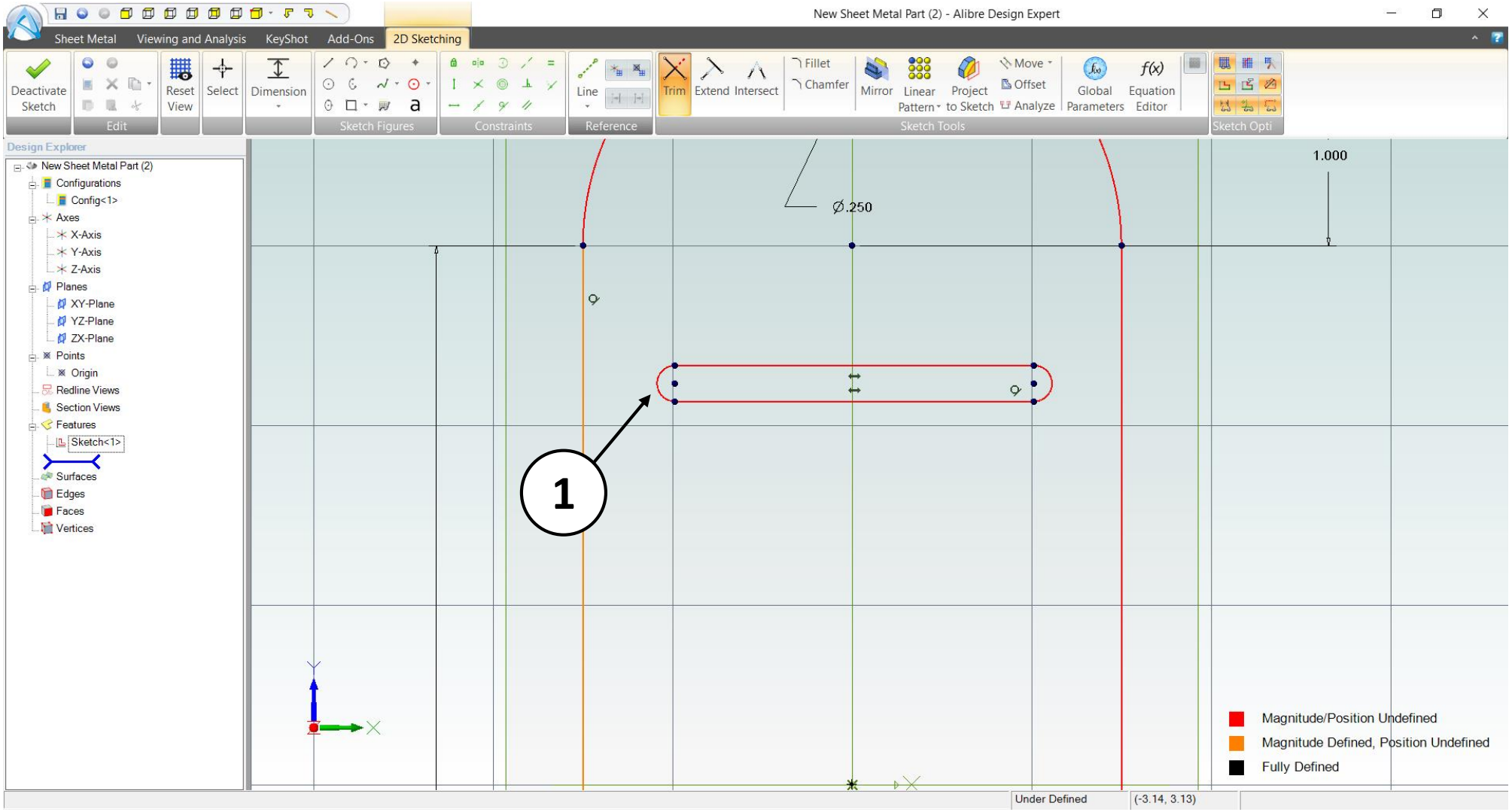
1. Use the **Rectangle sketch** tool to create a rectangular sketch similar to what is shown above.

2. Use the **Dimension** tool and apply a width of **.200”** and a length of **2.000”** to the rectangular sketch.

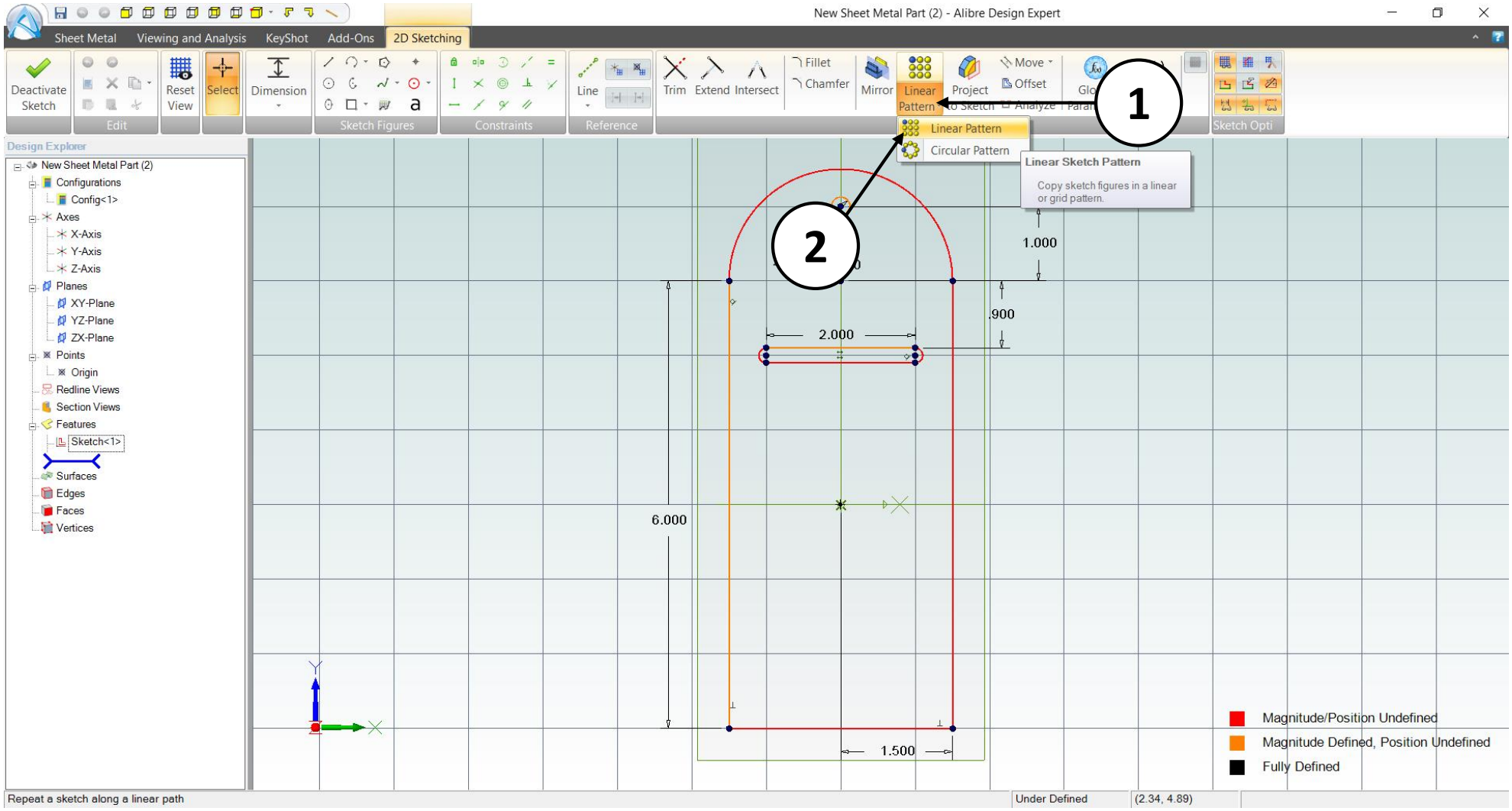


1. Use the **Dimension** tool to position the rectangular sketch accordingly so that it is centered horizontally on the workspace. Use a value of 0.500” for the distance from the left most vertical sketch line on the canvas to the left vertical sketch line of the rectangular sketch.

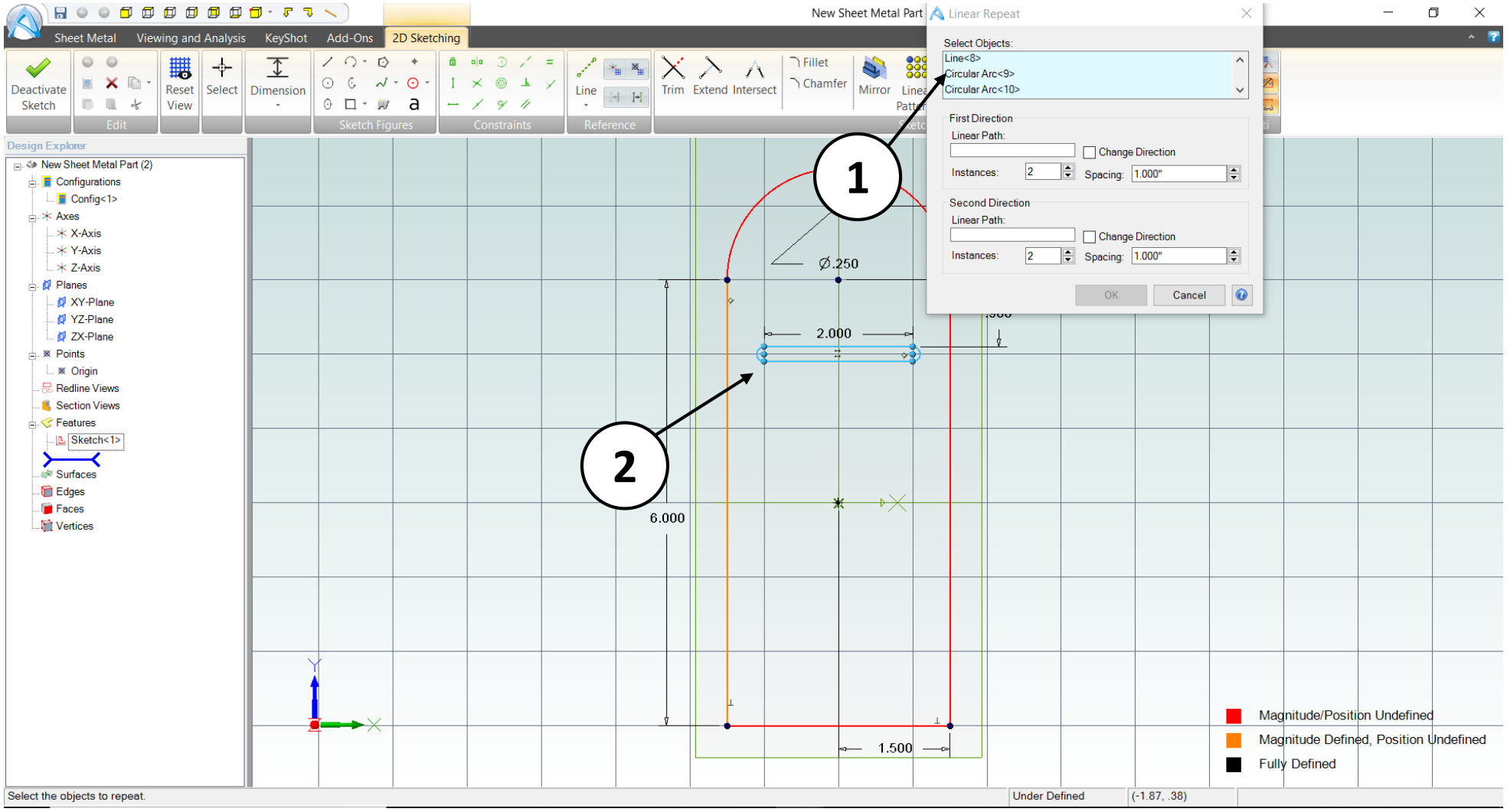
2. Use the **Dimension** tool and apply a 0.900” distance from the top horizontal line of the rectangular sketch to the node point.



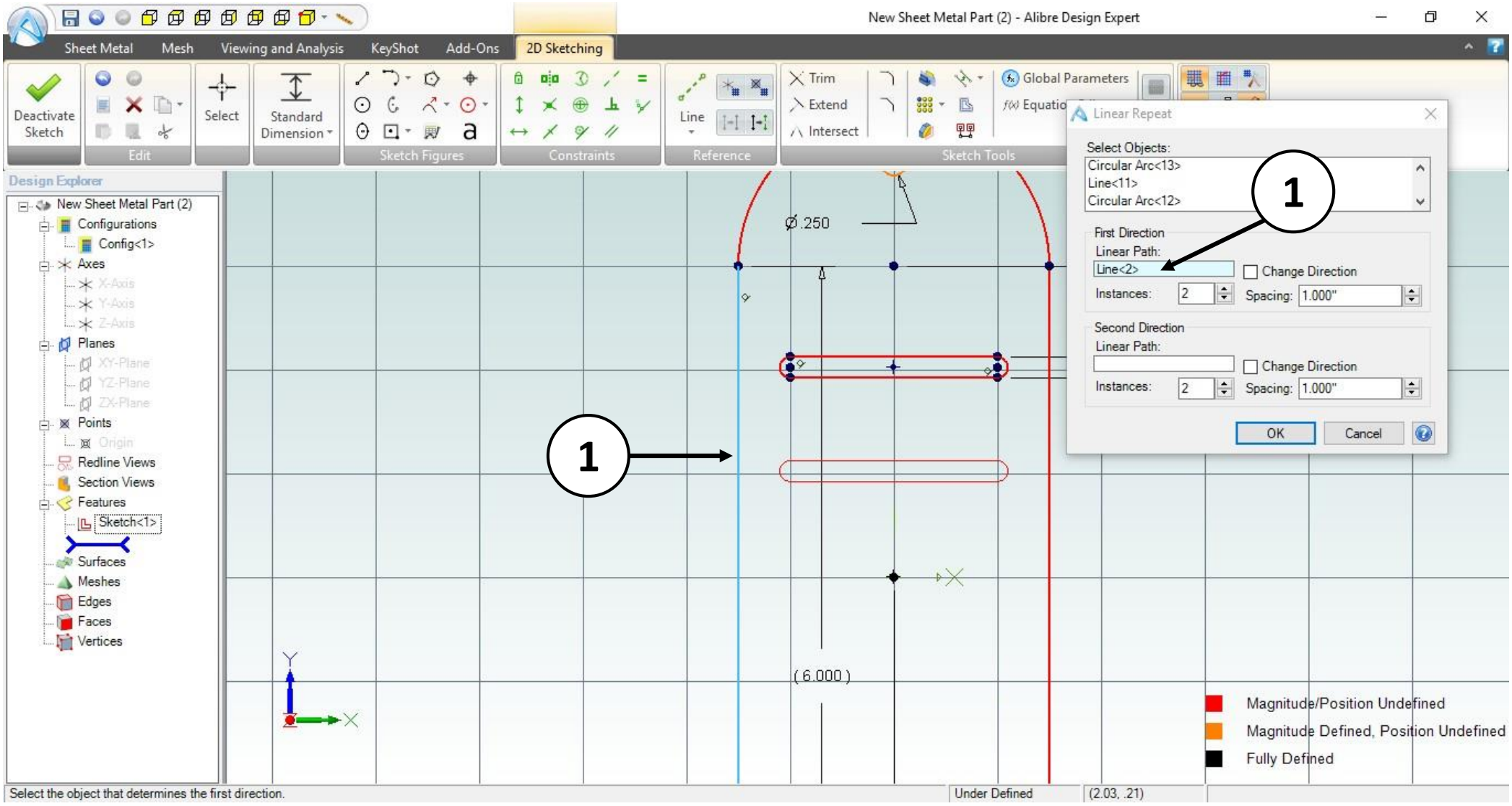
1. Use the Circular **Arc** – Start, End, Radius sketch tool and the **Trim** tool to edit the Rectangular sketch into an Obround sketch as shown in the image above.



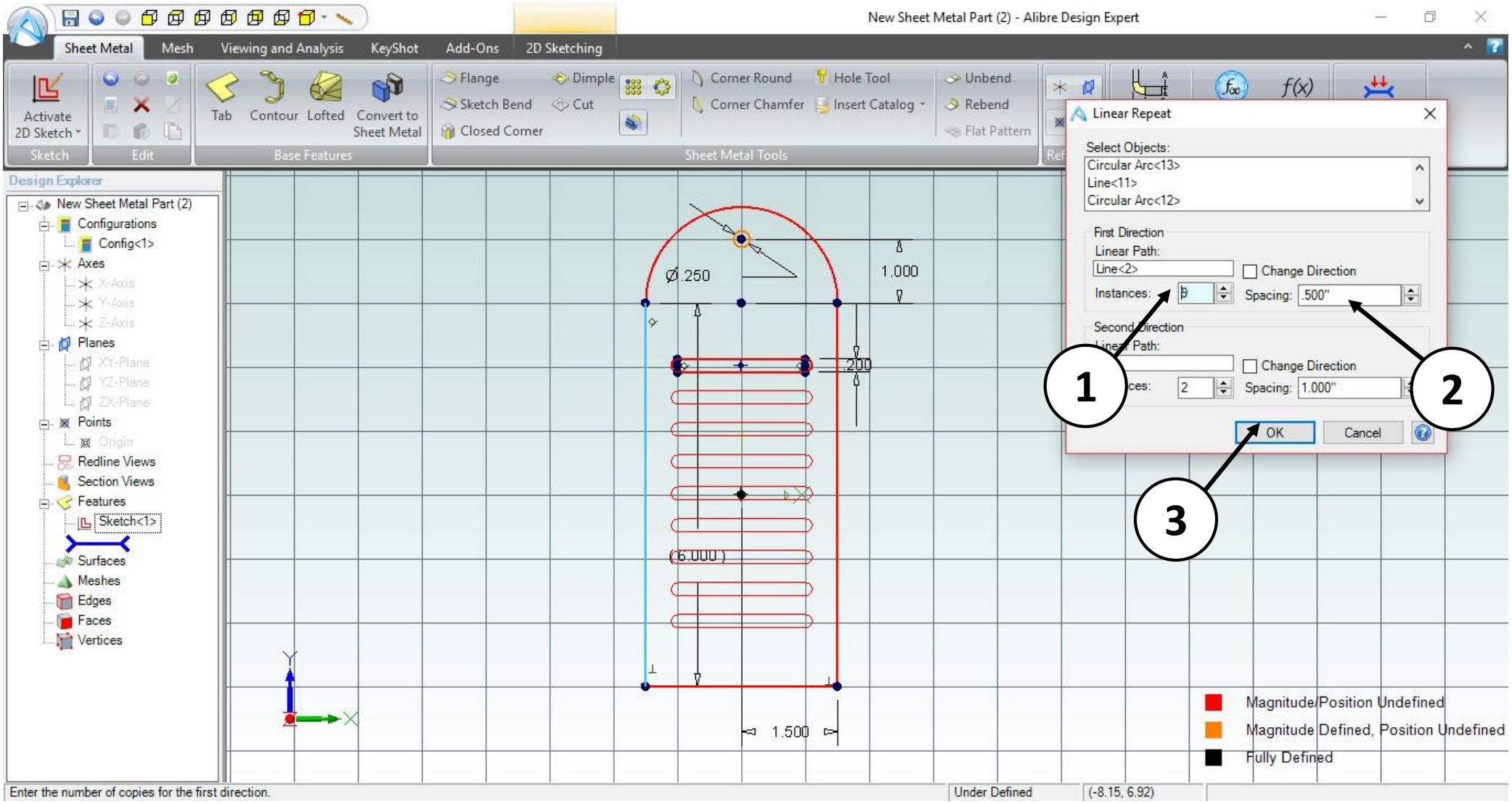
1. Click on the Dropdown menu of the **Pattern** tool on the **Sketch Tools** section of the ribbon.
2. Select **Linear Pattern**.



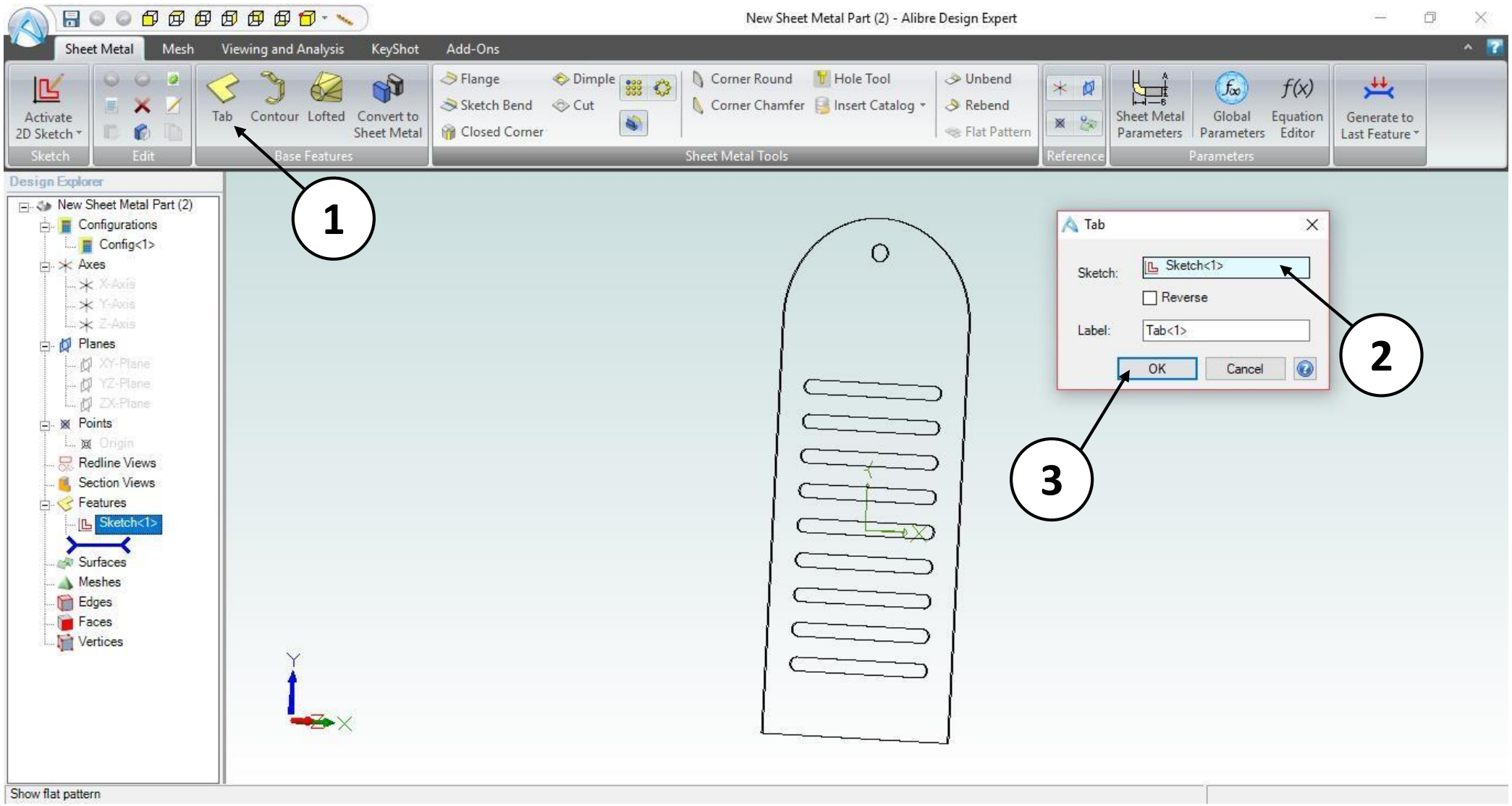
1. In the **Linear Repeat** dialog box, click on the **Select Objects** field.
2. Select the sketch lines to be patterned



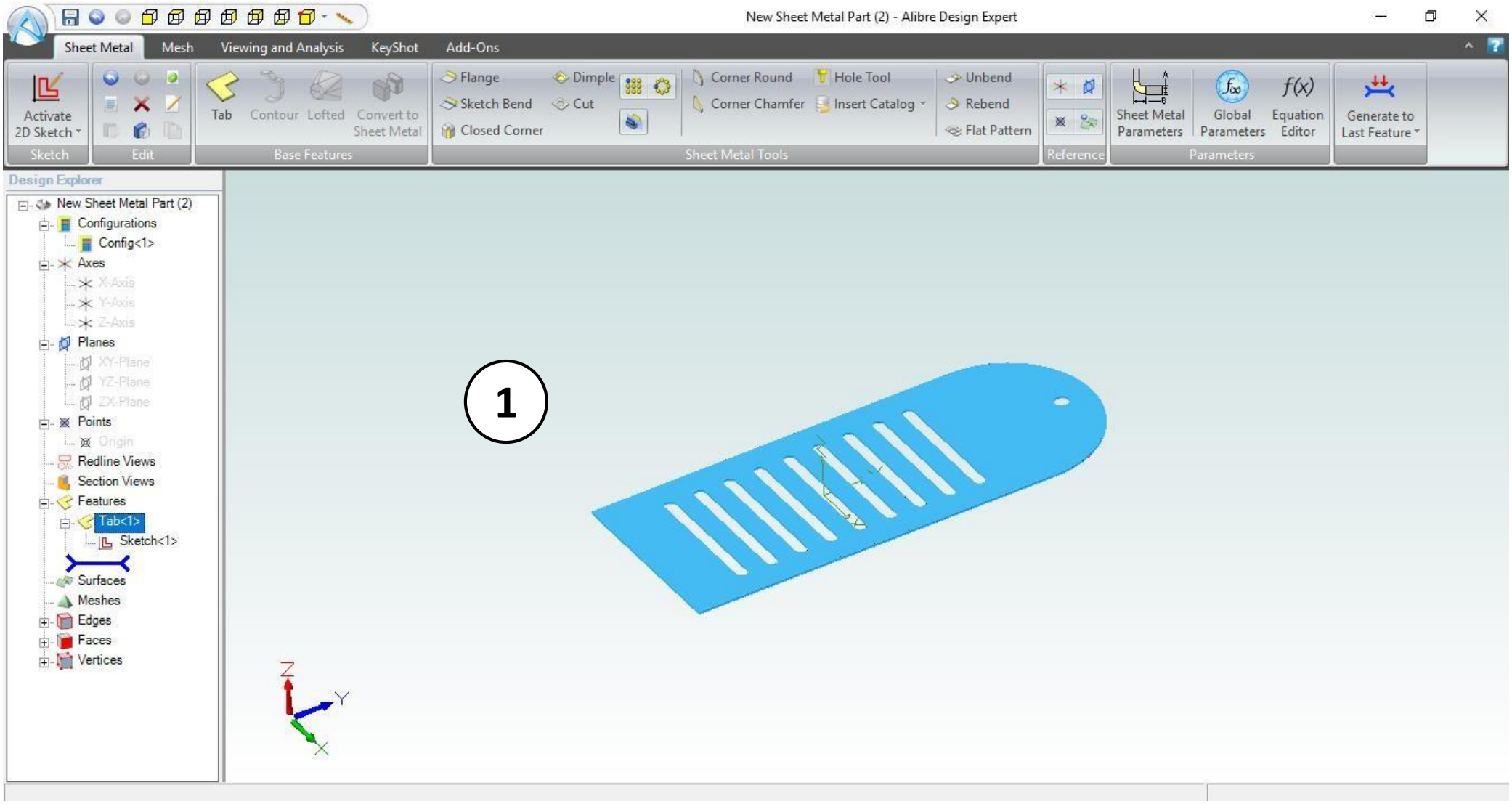
1. In the **Linear Repeat** dialog box, click on the box named **Linear Path**.
2. Select the highlighted edge as the **Pattern Direction**.



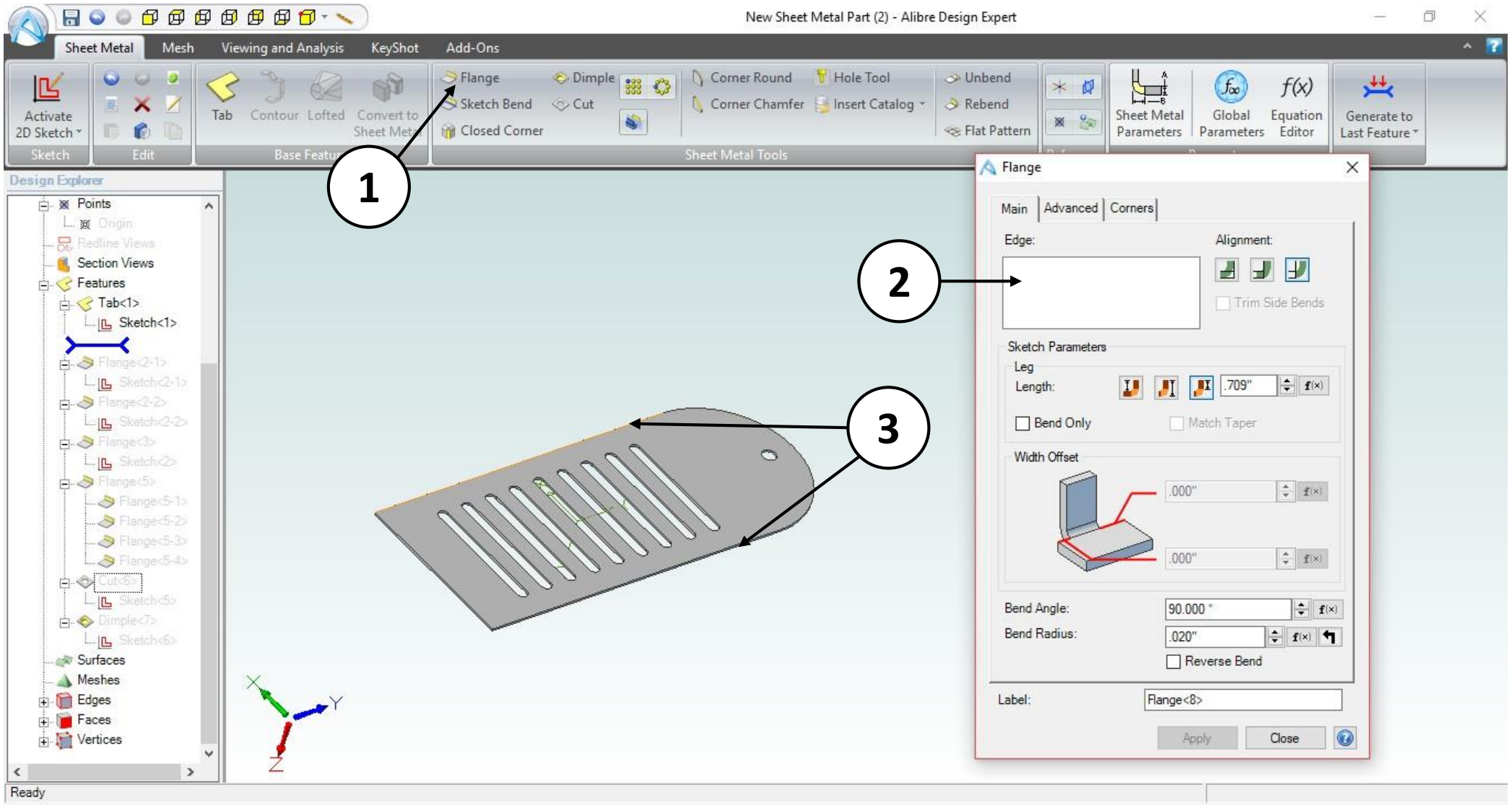
1. In the **Linear Repeat** dialog box, Increase the **Instances** to 9.
2. In the **Linear Repeat** dialog box, Change the **Spacing** to 0.500”.
3. Click **Ok** and then click **Deactivate Sketch** on the ribbon.



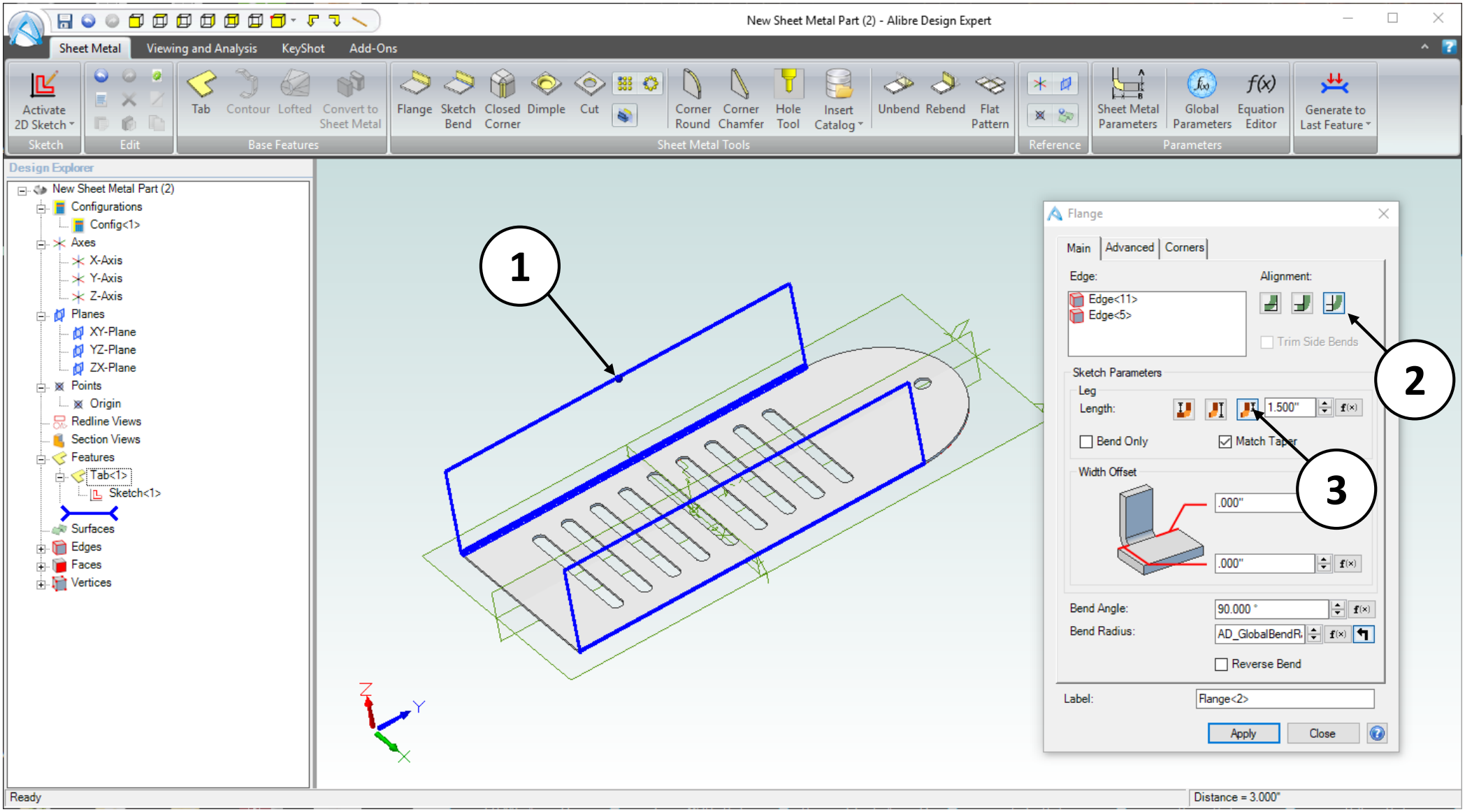
1. Click on the **Tab** in **Base Features** section in the ribbon.
2. Confirm that **Sketch<1>** appears in the **Tab;** entry box.
3. Click **OK.**



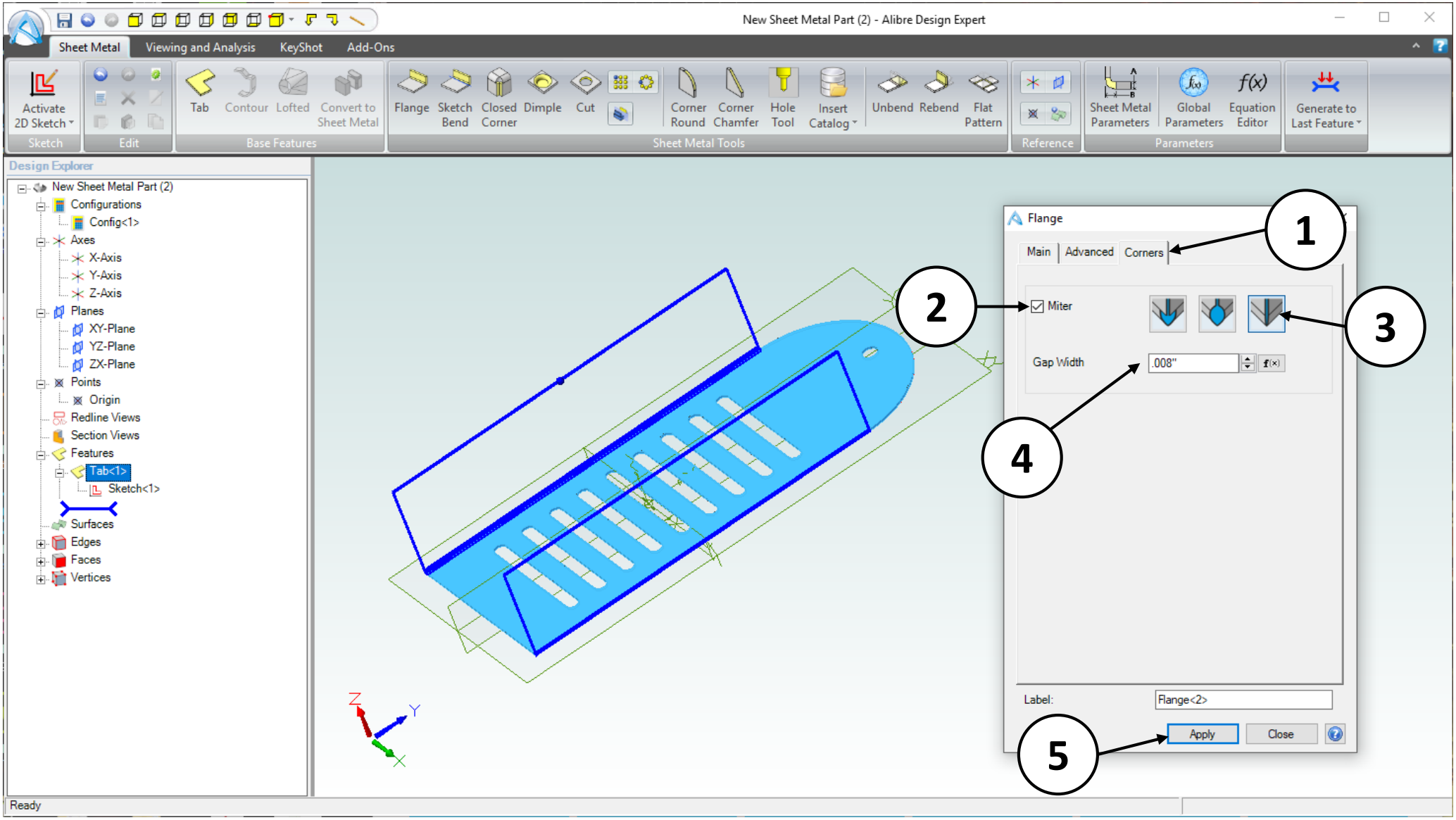
1. Confirm similar results for the **Tab** operation.



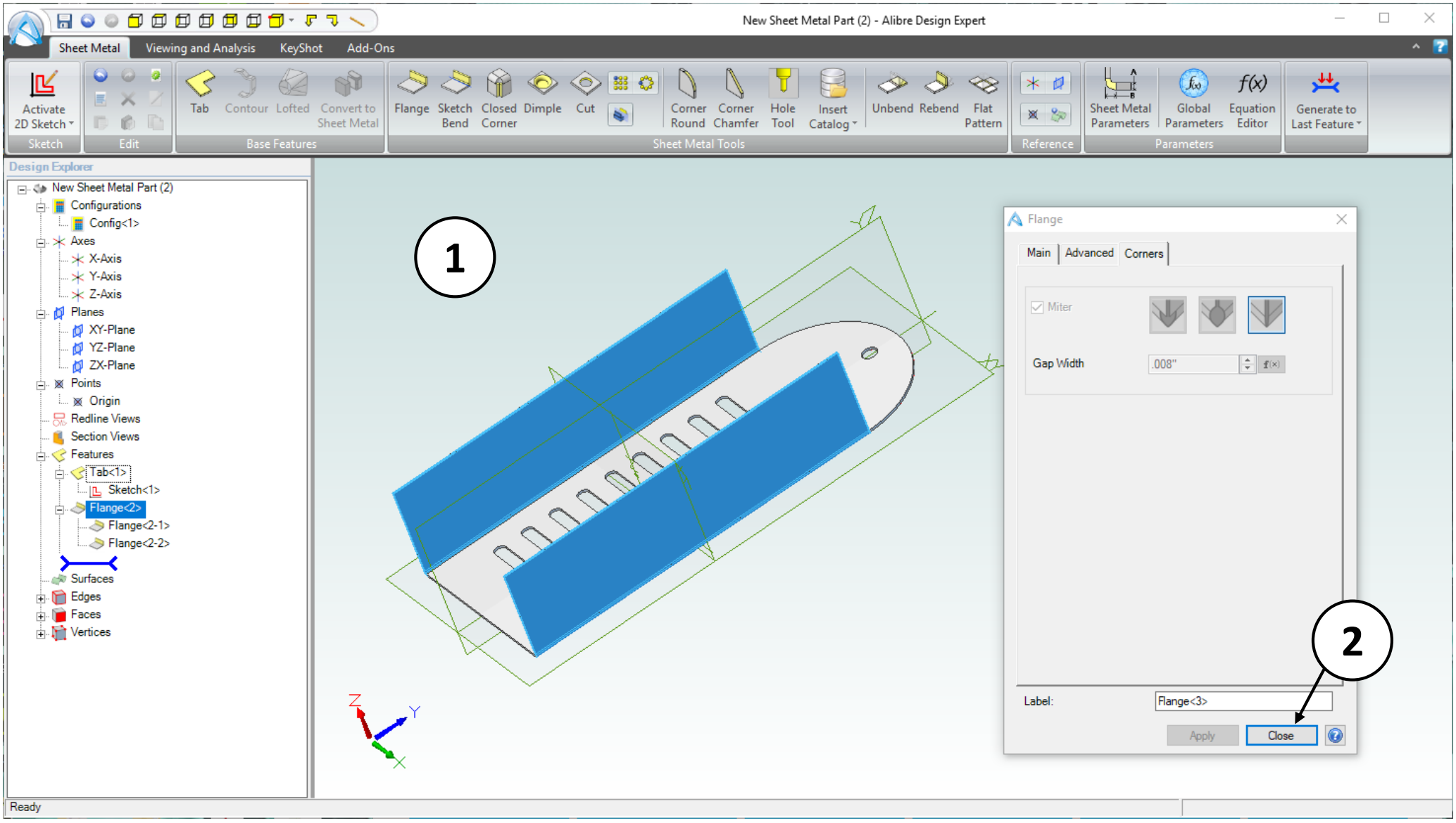
1. Click on the **Flange** in **Sheet Metal Tools** section in the ribbon.
2. Click the **Edge** box inside the Dialog
3. Click on the edges shown to select them.



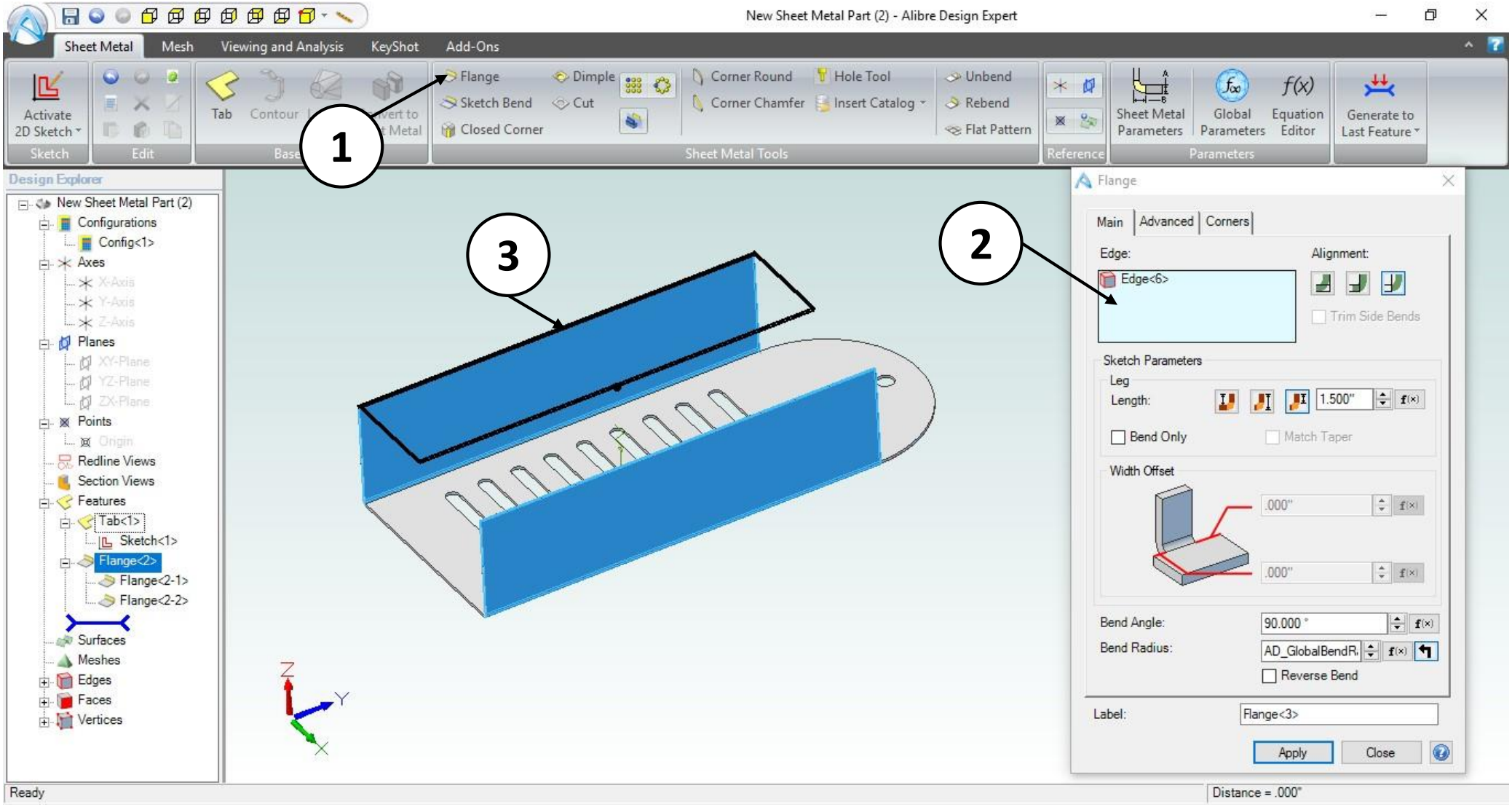
1. Move the mouse cursor over the point on the flange preview & click and drag until the measurement indicator in the **Flange** window displays **1.500**. *CLICK AND RELEASE* to complete the **Flanges**.
2. Select the **Adjacent** option under **Alignment** in **Main** tab of **Flange** dialog box.
3. Select the **Tab** under the **Length** option in **Main** tab of **Flange** dialog box.



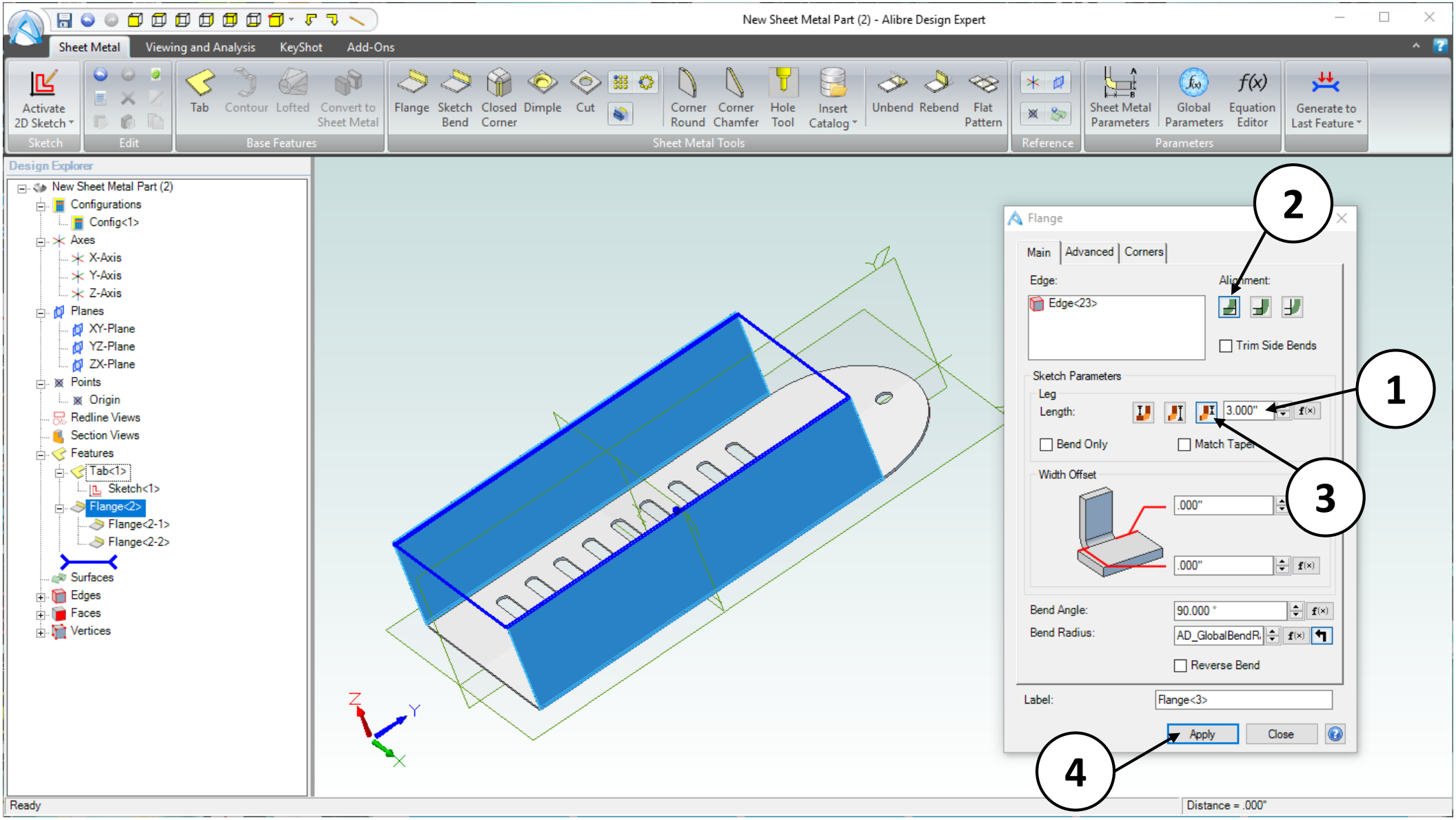
1. Click on the **Corners** tab in the **Flange** dialog box.
2. Click on the checkbox and Enable the **Miter**.
3. Select the **Corner Closed** under the **Miter** in **Corners** tab of **Flange** dialog box.
4. Input a value of **.008”** for the **Gap width**
5. Click **Apply.**



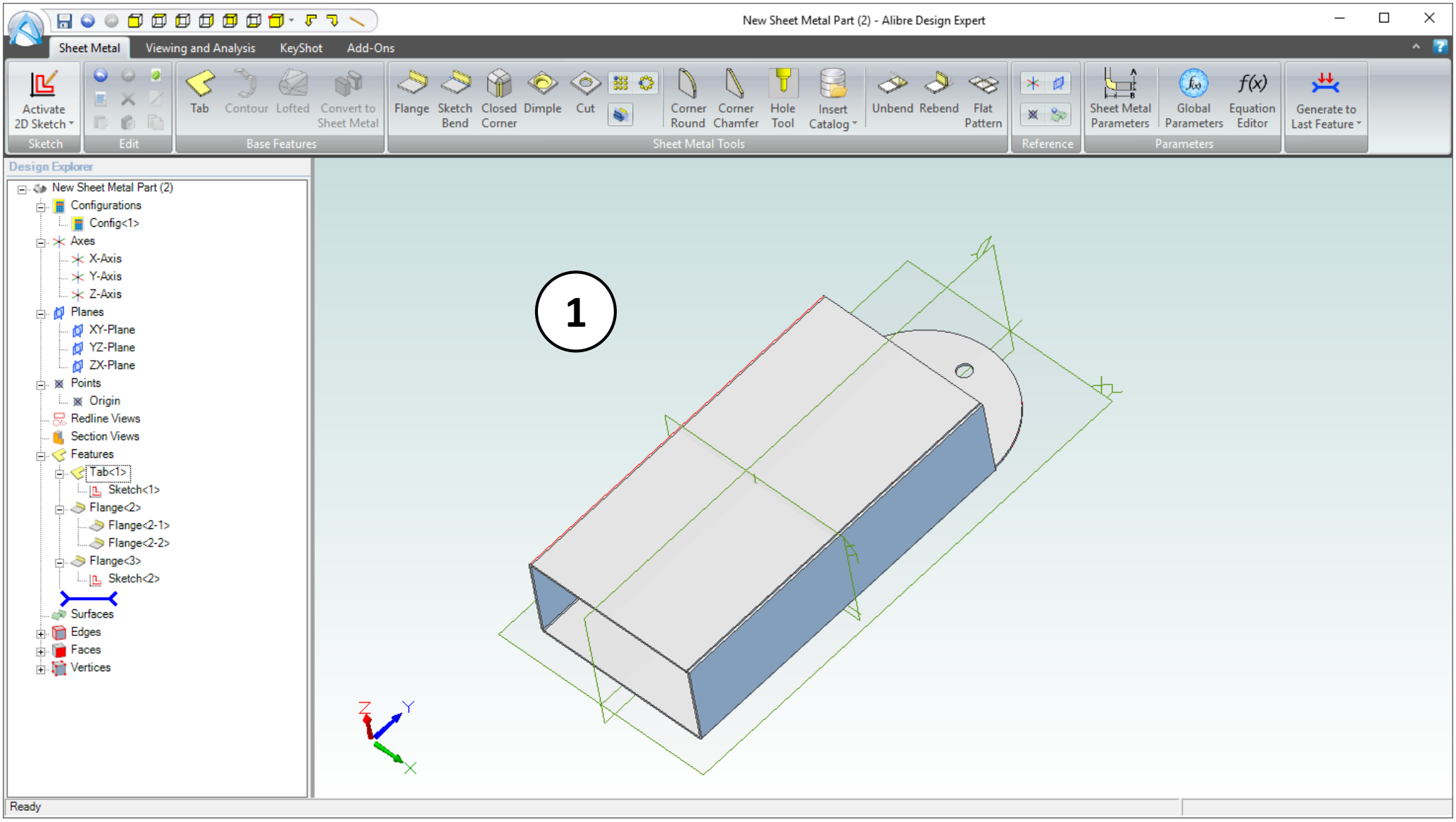
1. Confirm similar results for the **Flange** operation.
2. Click on the **Close**.



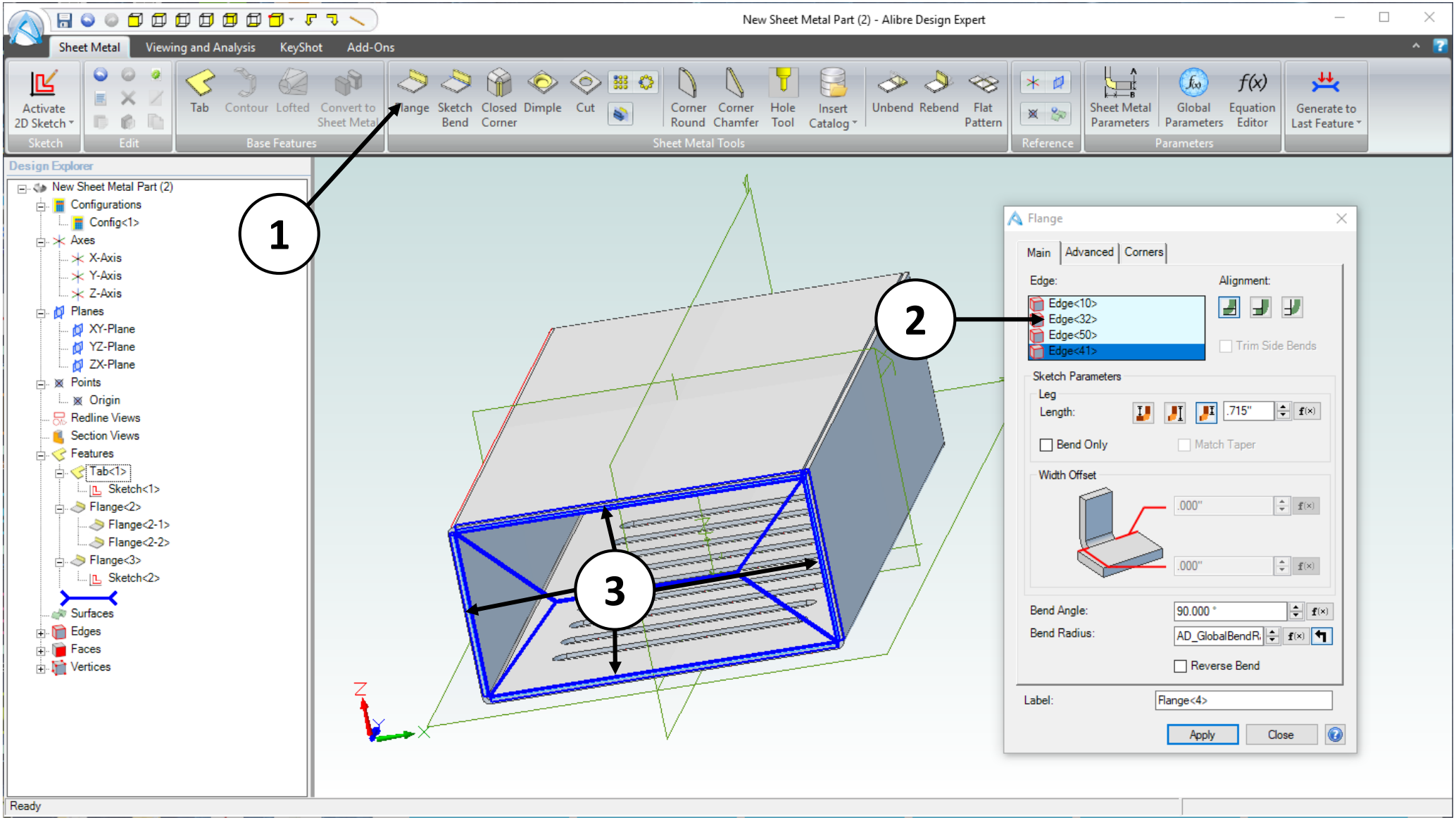
1. Click on **Flange** in **Sheet Metal Tools** section in the ribbon.
2. Click on the **Edge** box in the dialog.
3. Position the mouse cursor over the inside edge of the left flange, and then click to select it.



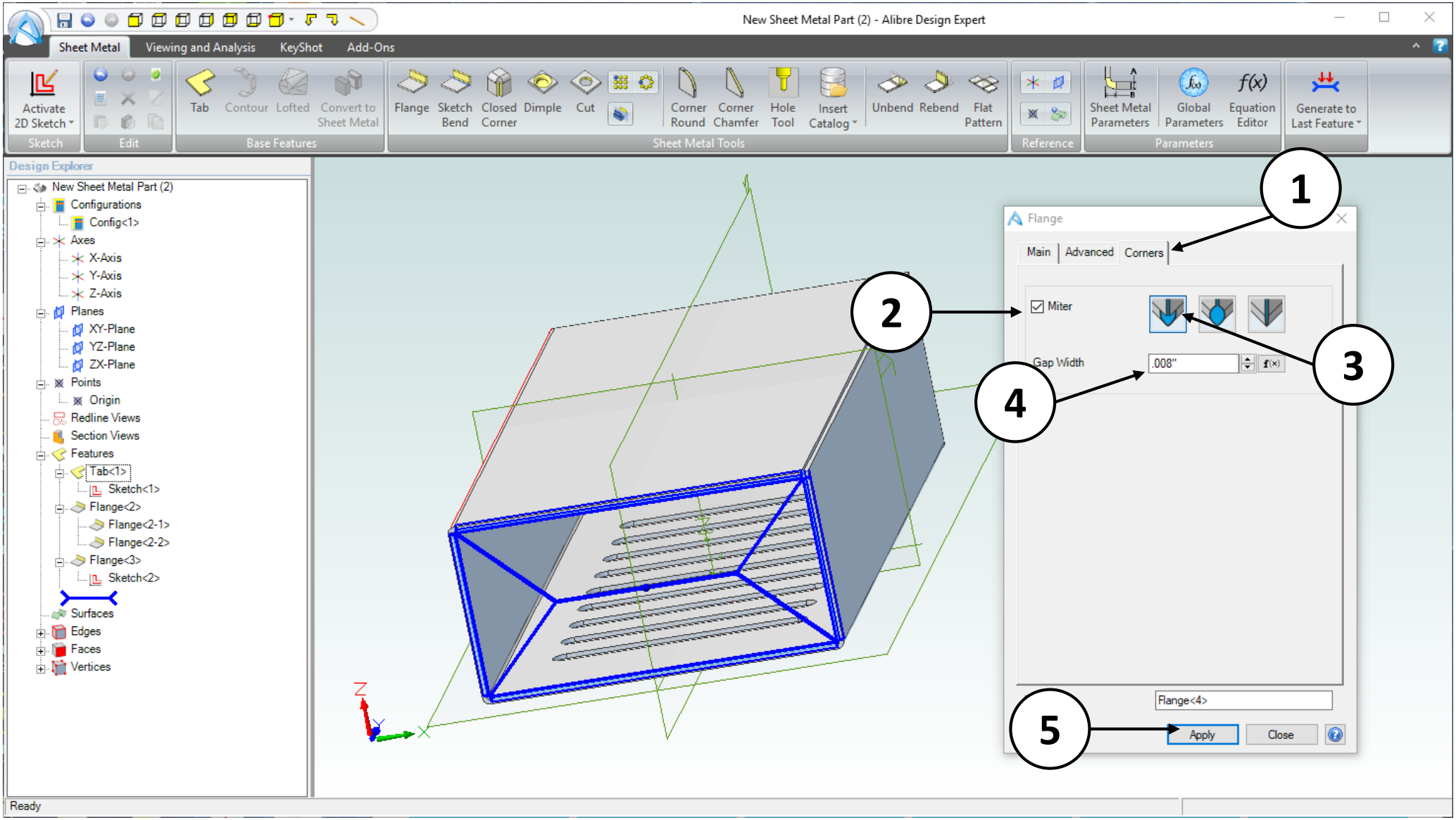
1. Enter a value of **3.000 inches** for the leg length in the dialog.
2. Select the **Inside** option under **Alignment** in **Main** tab of **Flange** dialog box.
3. Select the **Tab** under **Length** in the **Main** tab of **Flange** dialog box.
4. Click **Apply** then **Close.**



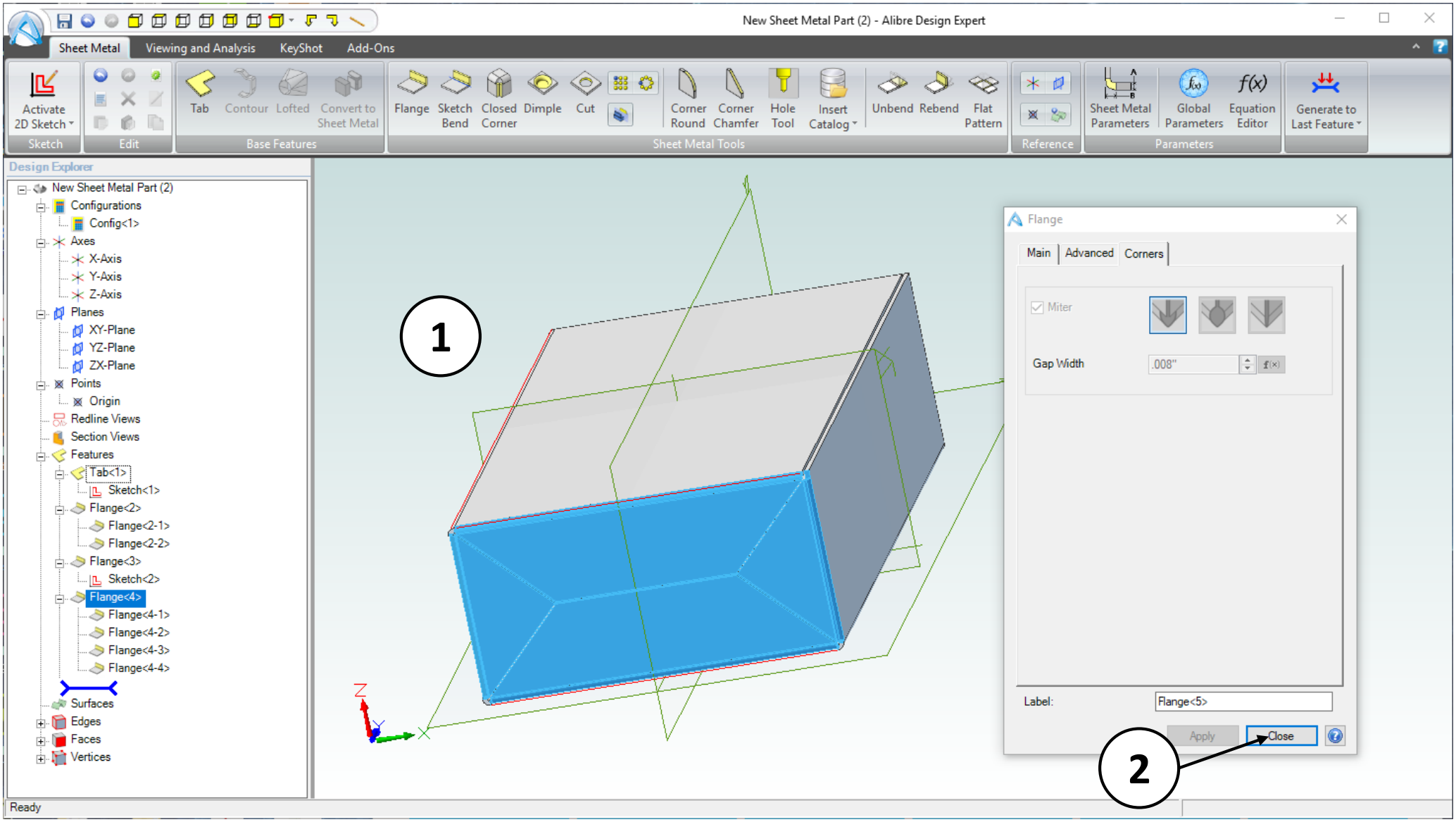
1. Confirm similar results for the **Flange** operation.



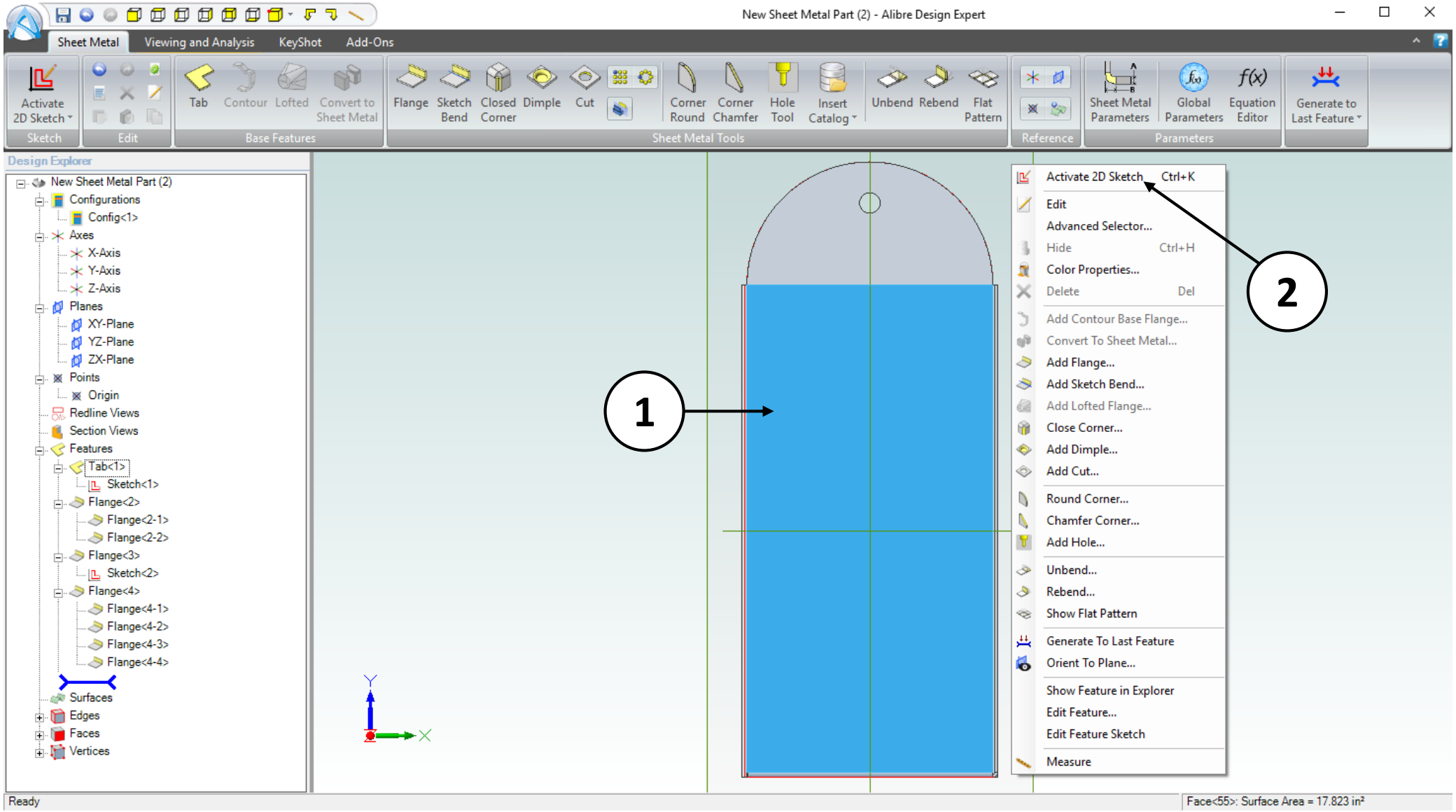
1. Click on **Flange** in **Sheet Metal Tools** section in the ribbon.
2. Click the **Edge** box in the dialog.
3. Click on all four inside edges of the bottom of the sheet metal part
4. Set the length value to **.715”**



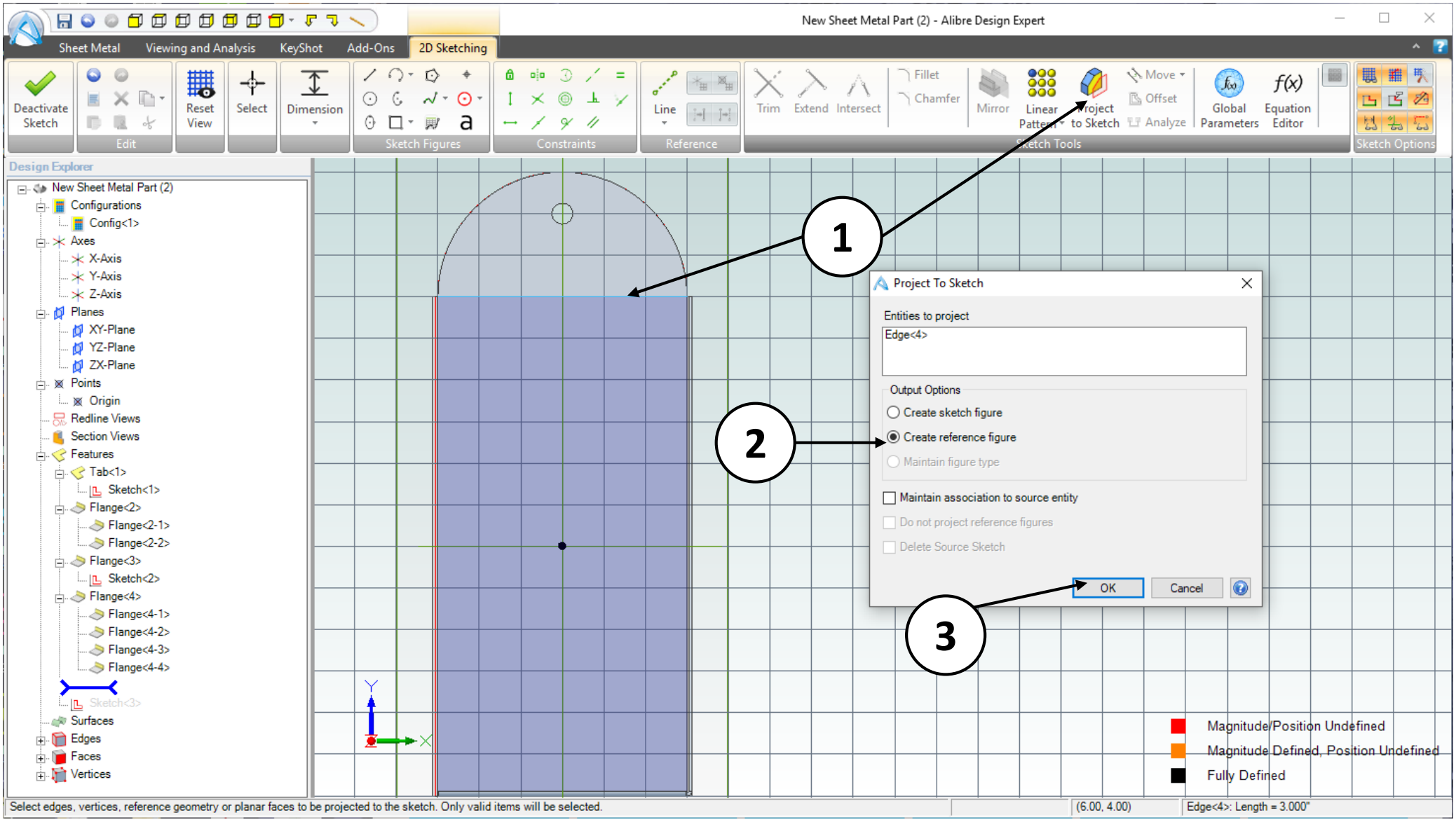
1. Click on the **Corners** tab in the **Flange** dialog box.
2. Click on the checkbox and Enable the **Miter**.
3. Select **Corner Relief** under **Miter** in the **Corners** tab of **Flange** dialog box.
4. Confirm the **Gap width** value in the **Flange** dialog box is **0.008 inches**.
5. Click **Apply.**



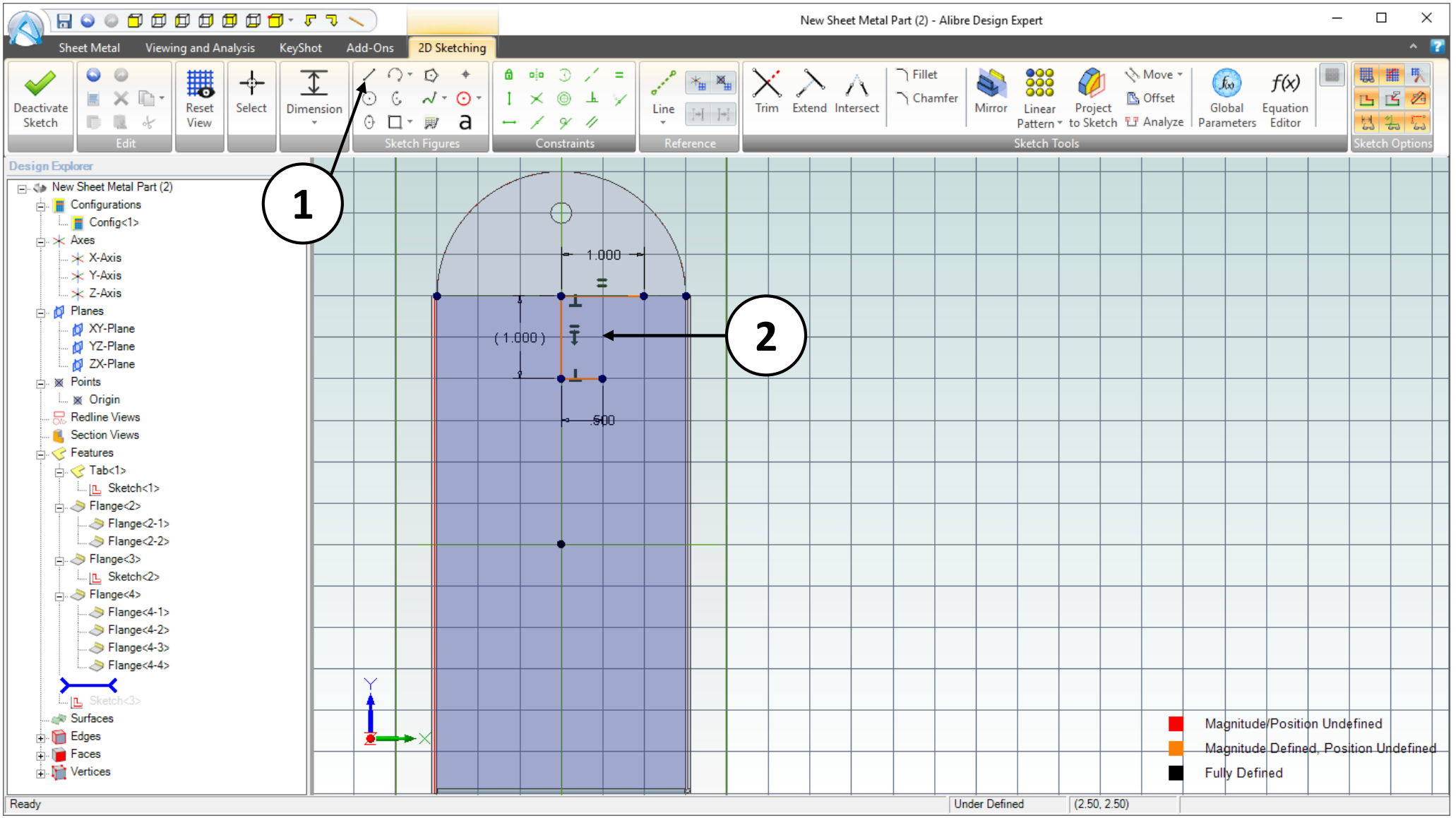
1. Confirm similar results for the **Flange** operation.
2. Click on the **Close**.



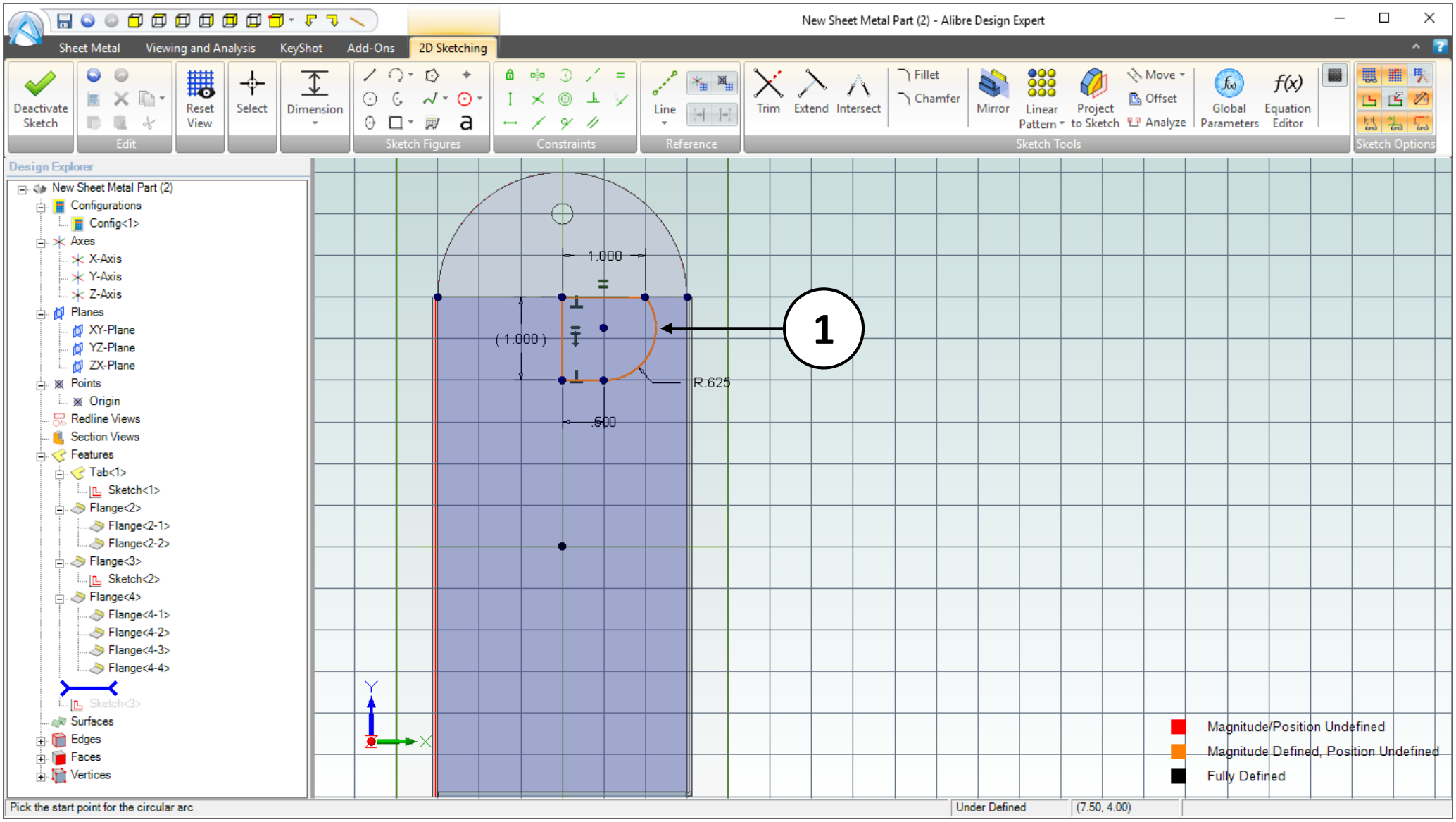
1. Right-click on the front face of the sheet metal part
2. Select the **Activate 2D Sketch** from the Menu or Press **Ctrl+K** on Keyboard.



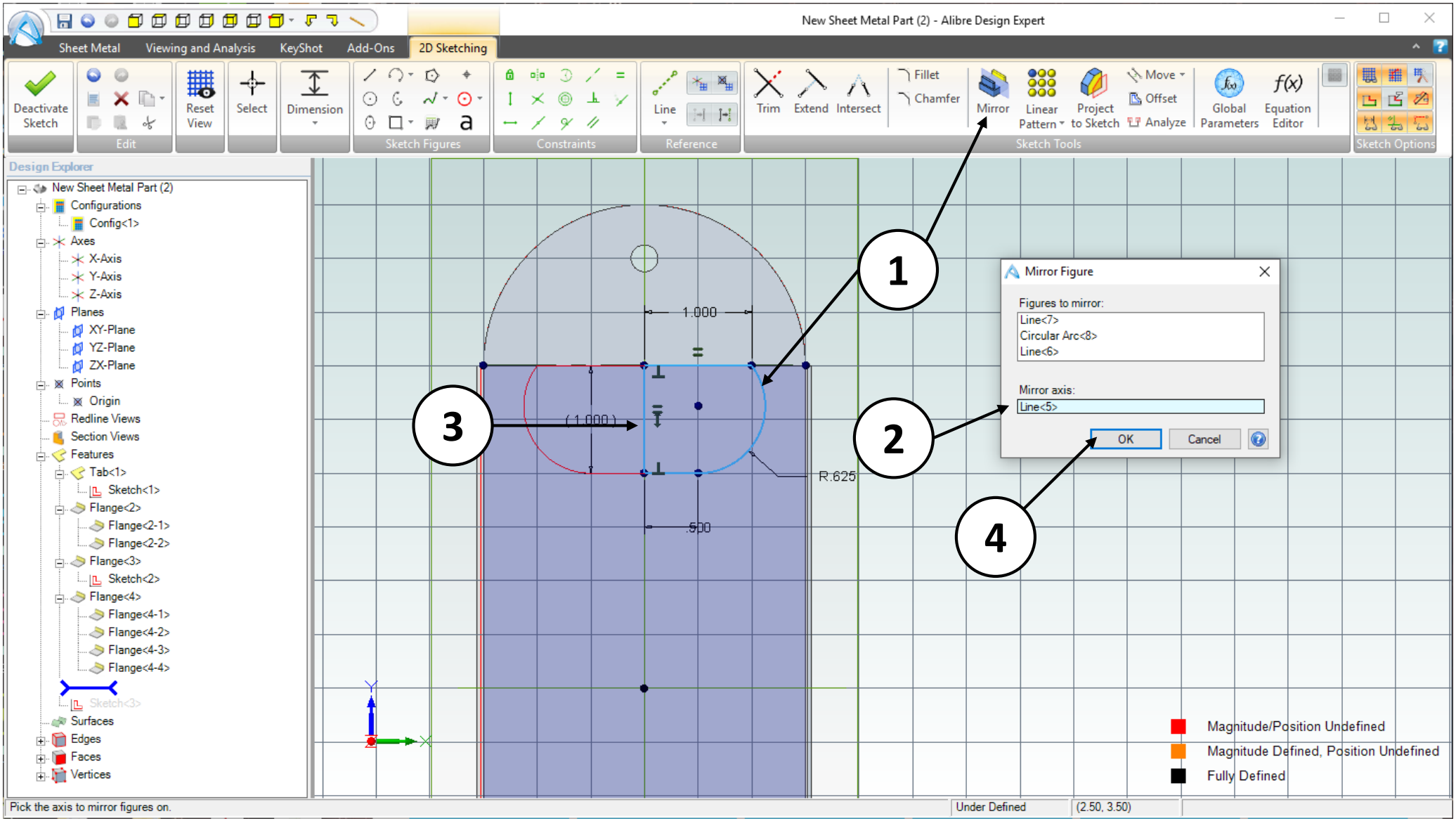
1. Click on the **Project to Sketch** tool and then click on the top edge of the front face of the model.
2. Choose the **Create reference figure** option in the dialog
3. Click **OK**



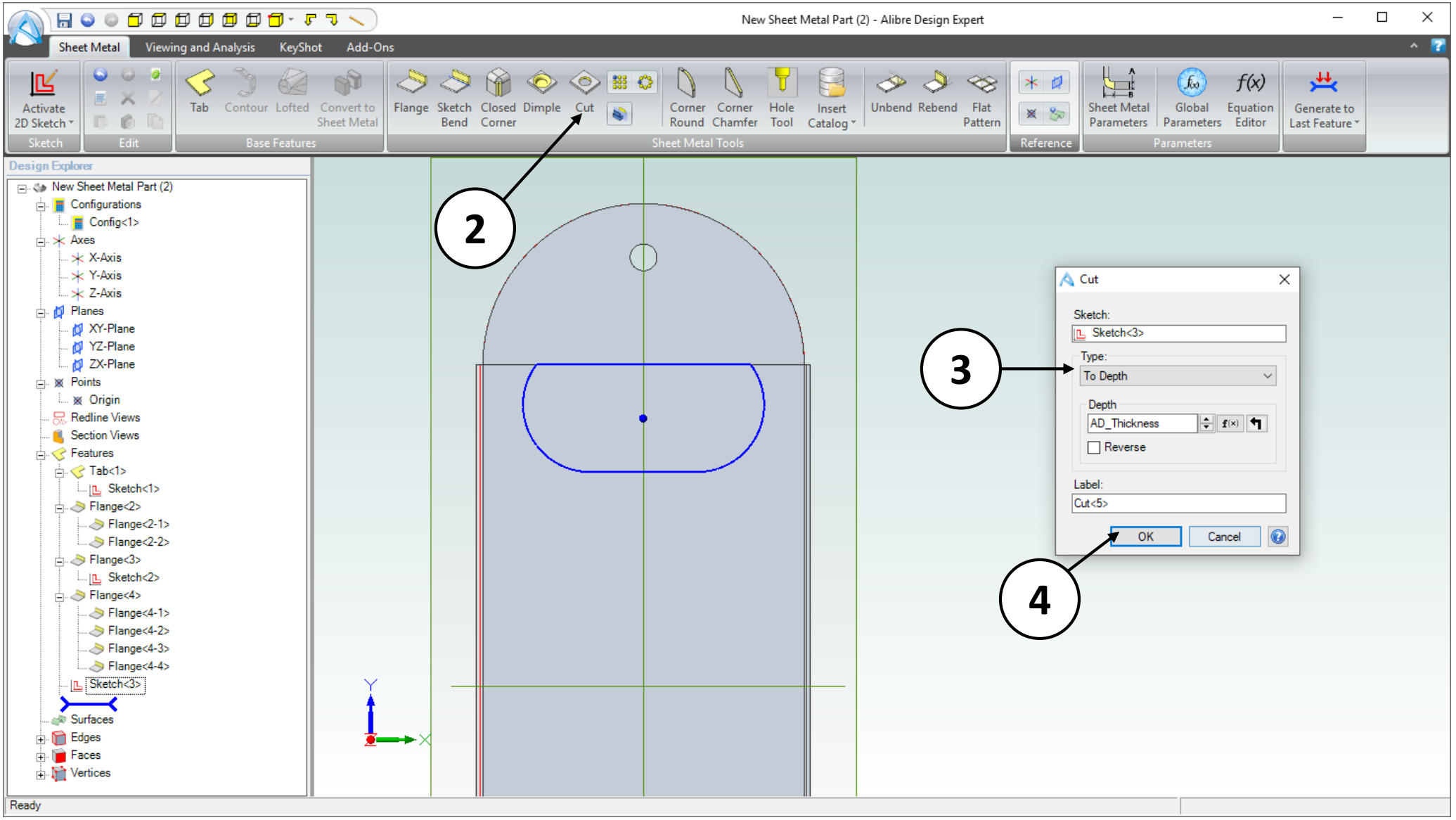
1. Click on the **Line** tool, then hover your cursor over the center point of the reference line.
2. Create the sketch lines as shown.



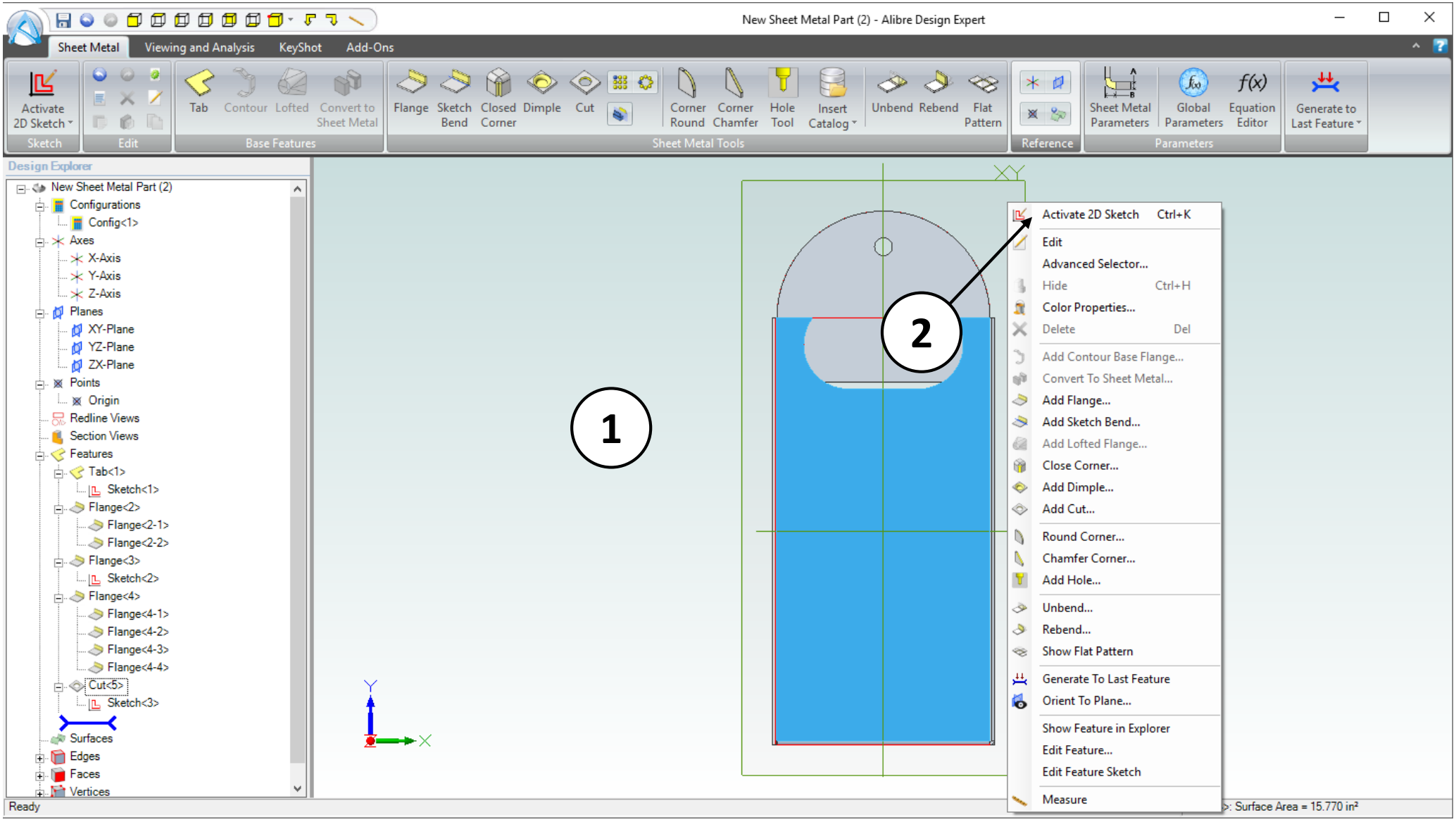
1. Use the **Circular Arc-Start, End, Radius** to create an arc with a radius of **.625”** to close the sketch.



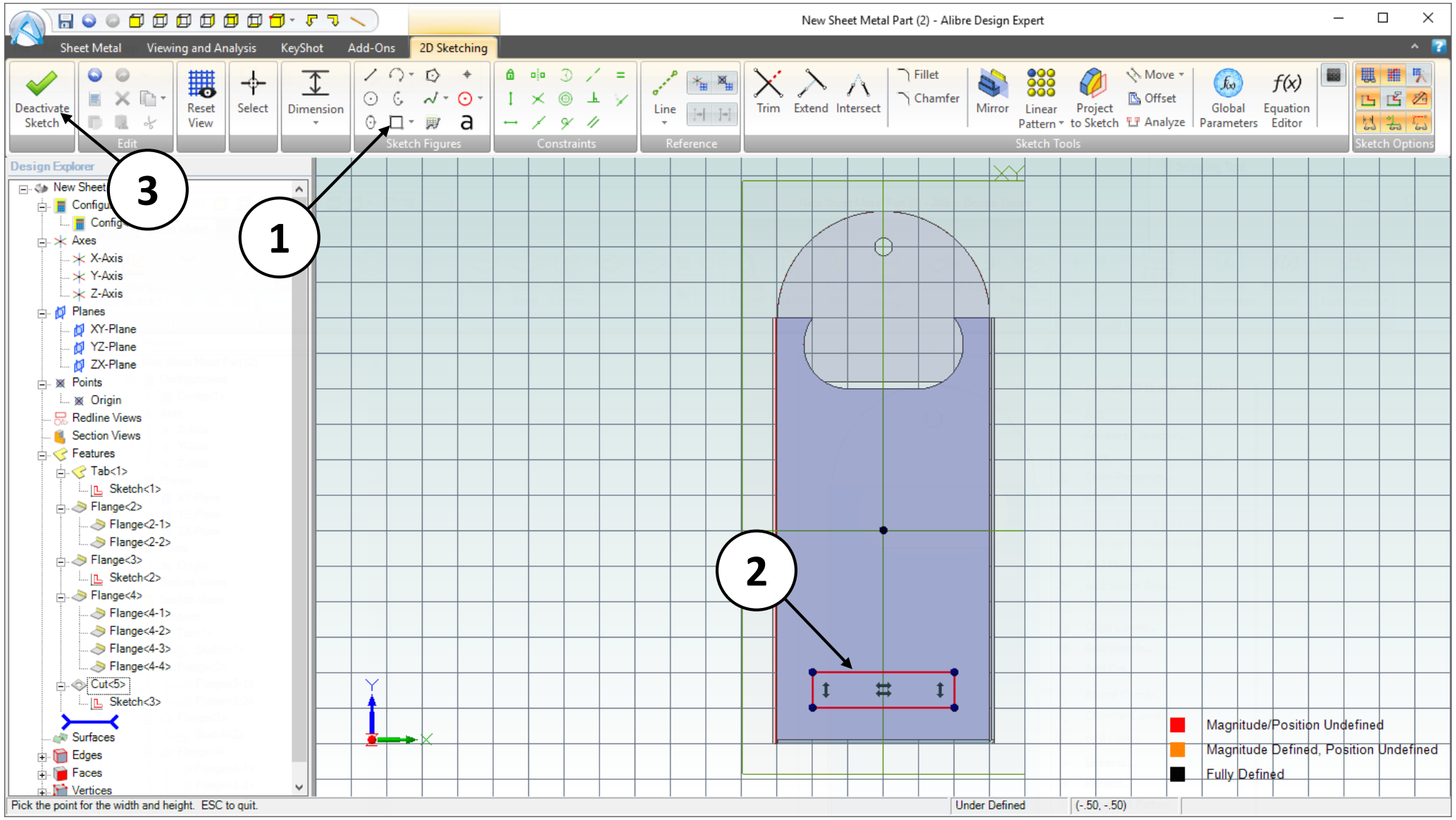
1. Click on the **Mirror** tool and select the sketch lines you just created except for the vertical line, and populate the **Figures to Mirror** box of the Mirror dialog
2. Click on the **Mirror Axis** box of the mirror figure dialog.
3. Select the vertical line of the sketch for the **Mirror axis** field in the dialog**.**
4. Click **OK**.



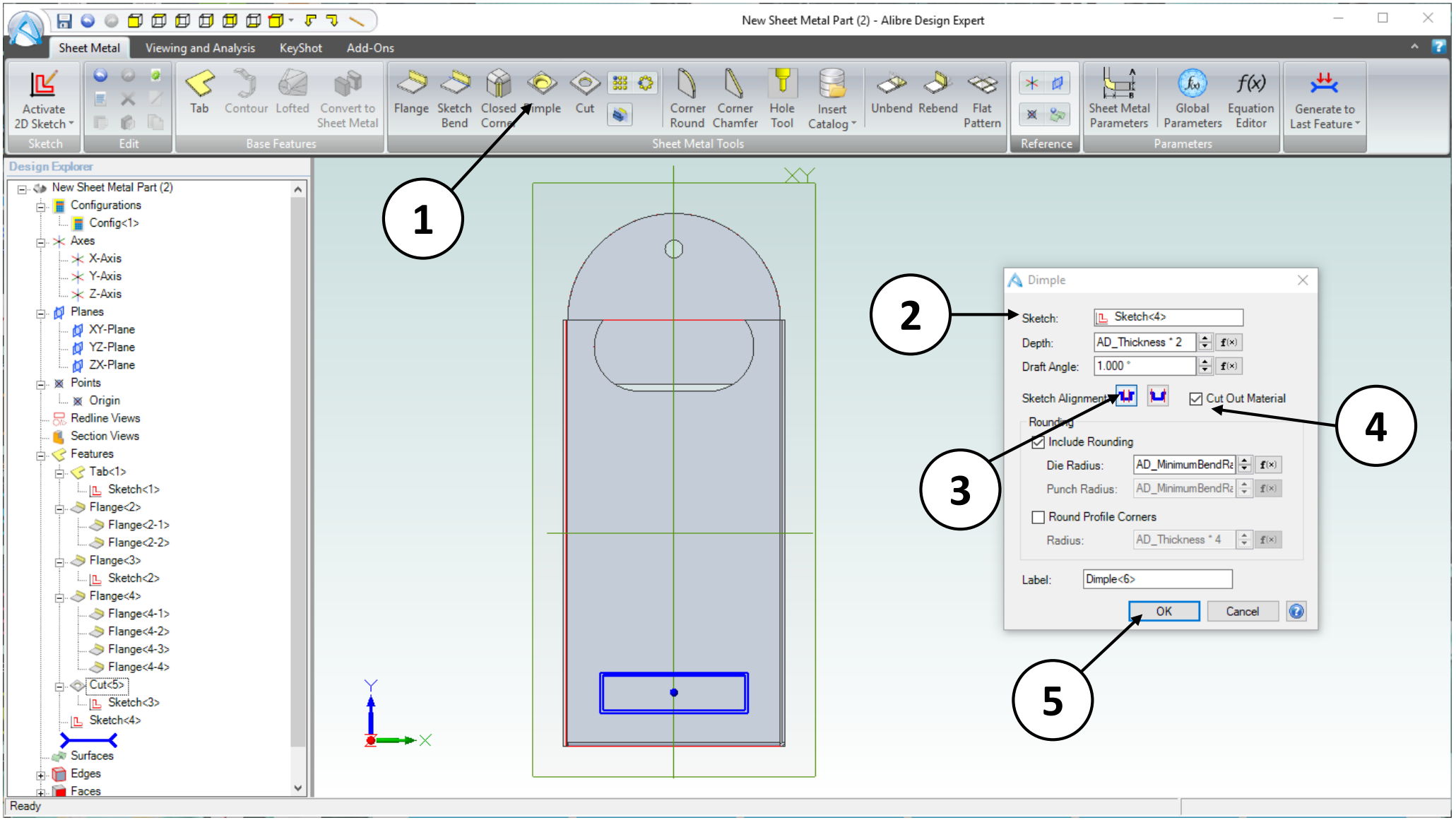
1. Use the **Trim** tool to remove the center line. Click **Deactivate Sketch** (Not shown).
2. Click on the **Cut** tool in the **Sheet Metal Tools** section on the Ribbon.
3. Select the **To Depth** option from the dropdown menu of **Type** in the **Cut** Dialog Box.
4. Click **OK.**



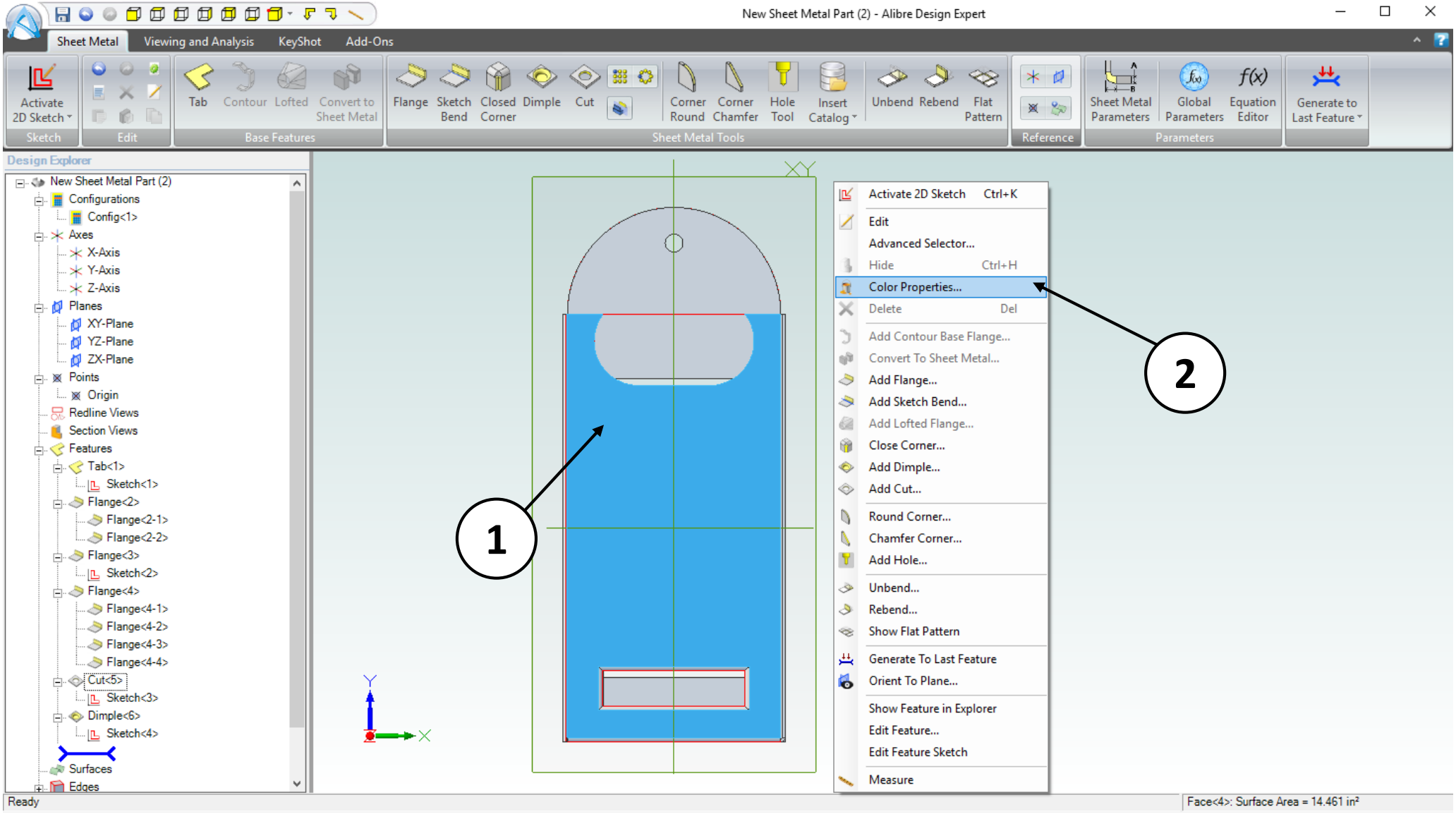
1. Confirm similar results for the **Cut** operation.
2. Right-click on the front face of the part and select the **Activate 2D Sketch** from the Menu or Press **Ctrl+K** on Keyboard.



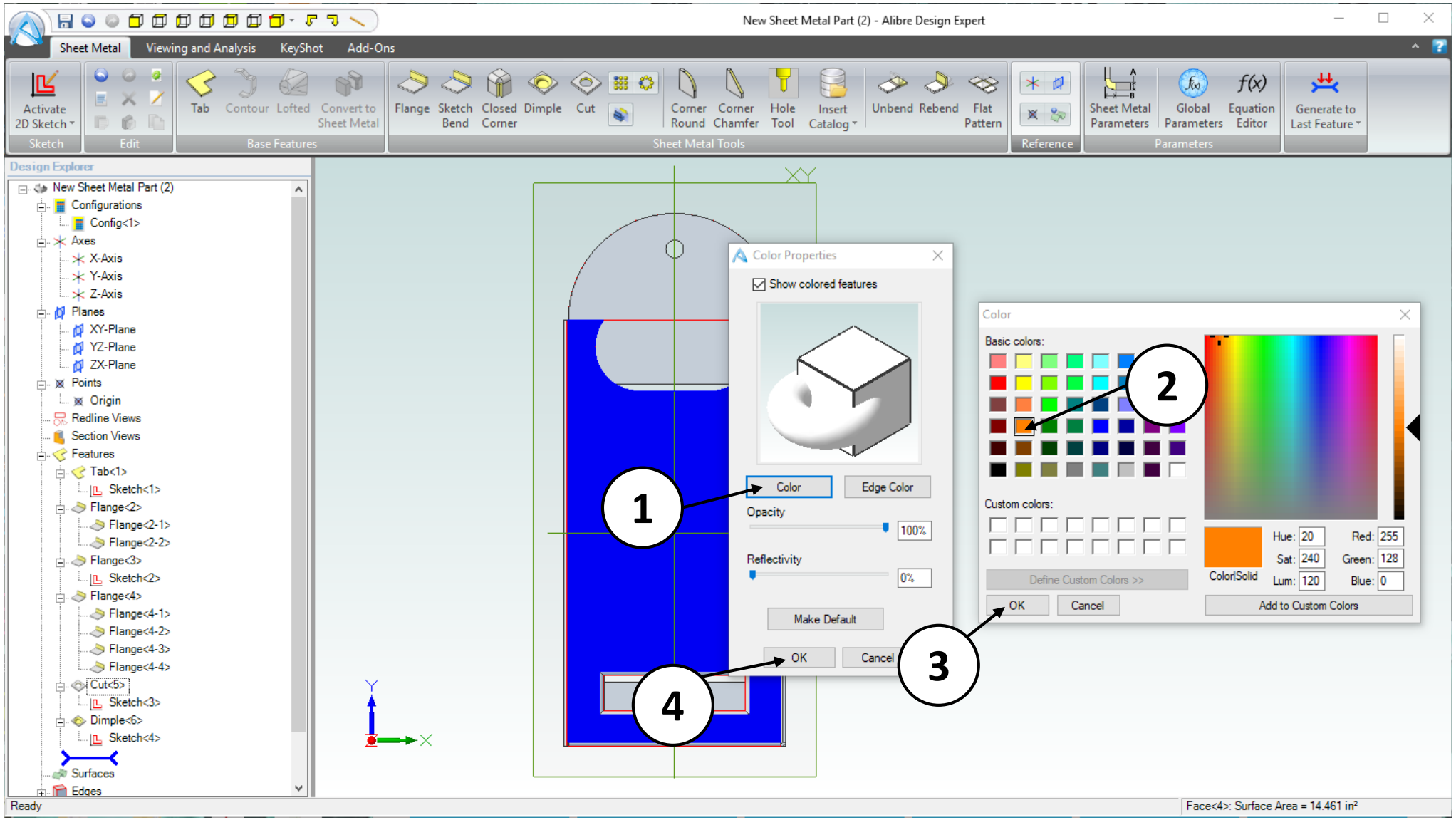
1. Select the **Rectangle** tool in the **Sketch Figures** section on the Ribbon.
2. Sketch a rectangle with a **Length** of **2.000”** and a **Width 0.250“**, whose top horizontal sketch line is one inch above the bottom of the part**.**
3. Click **Deactivate Sketch.**



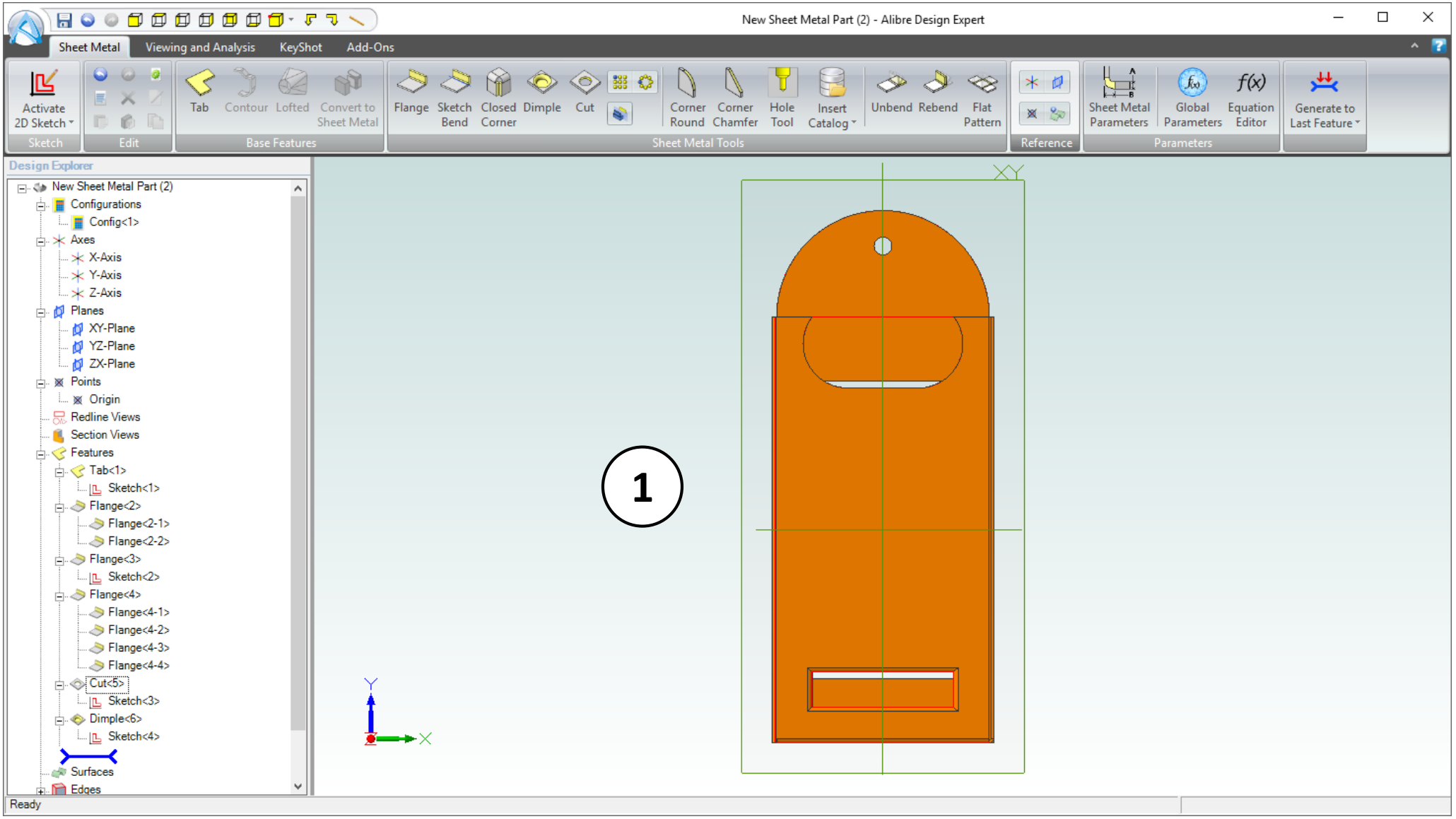
1. Click on the **Dimple** tool in **Sheet Metal Tools** section in the Ribbon.
2. Confirm that the sketch you just created appears in the Sketch field of the Dimple dialog.
3. Input a **1 degree** draft angle in the **Draft angle** field of the dimple dialog
4. Enable the Checkbox named **Cut Out Material**.
5. Click **OK.**



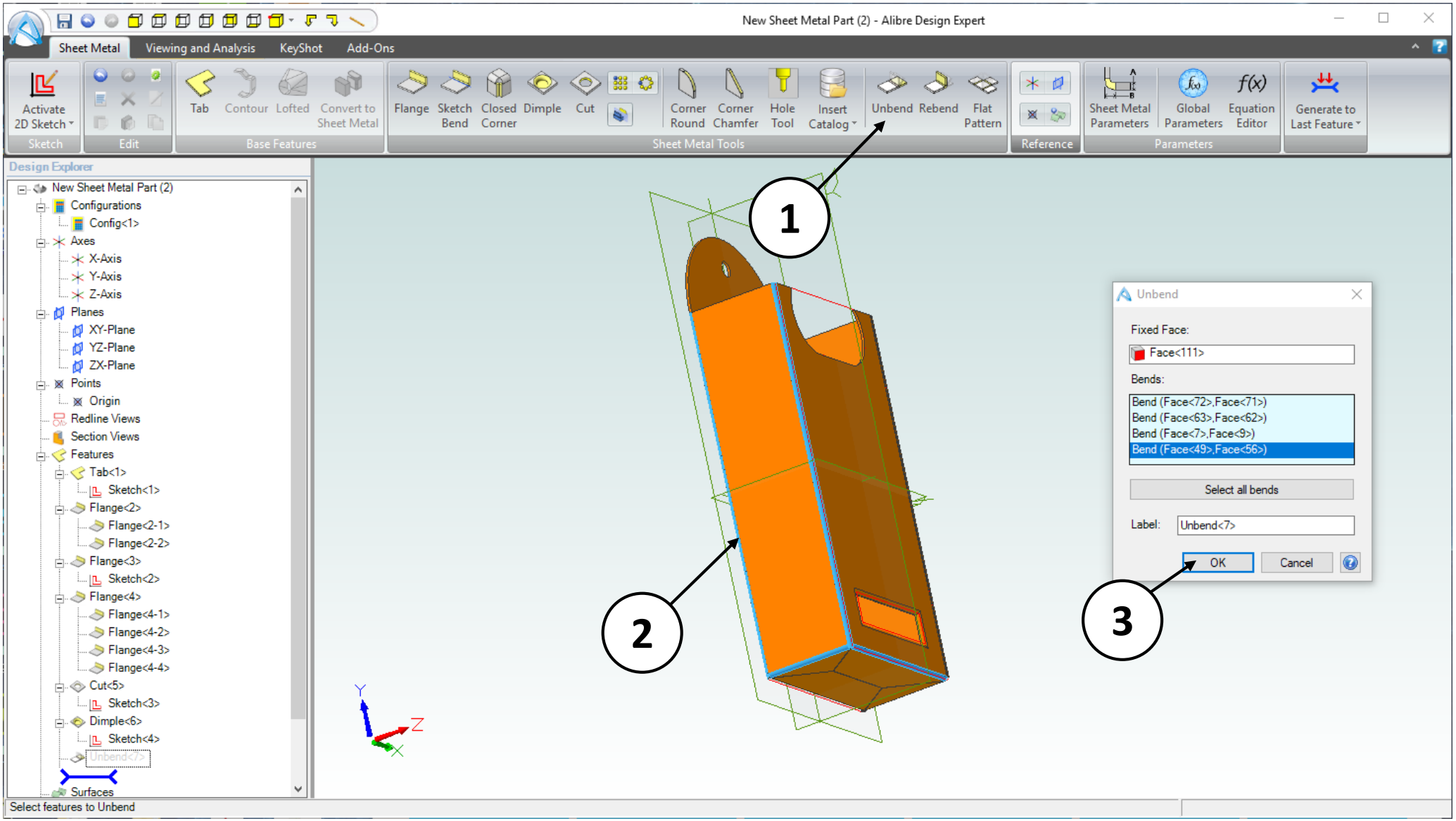
1. Right click anywhere on the Sheet Metal part.
2. From the pop-up menu select **Color Properties**.



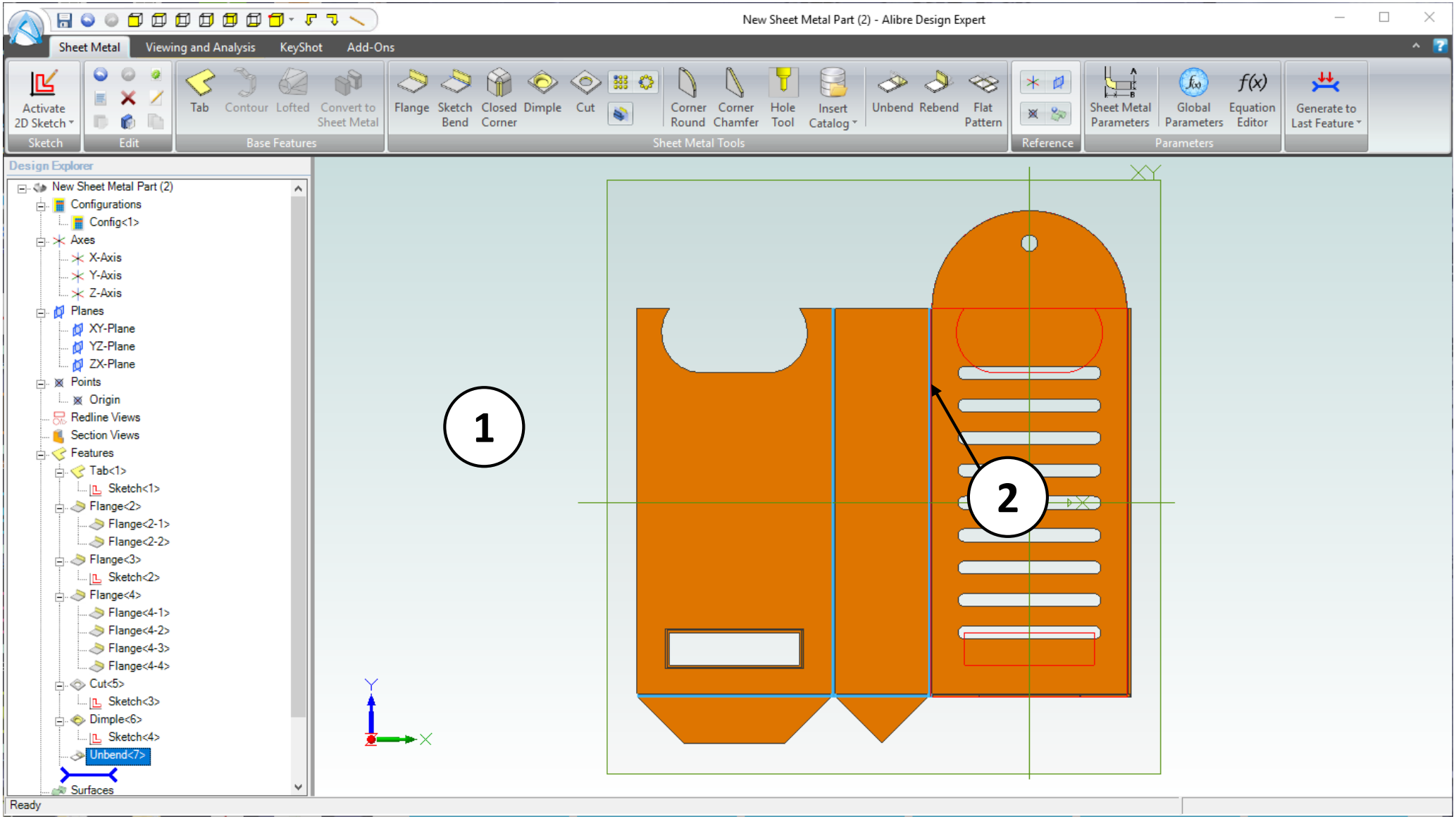
1. Click on **Color** in the **Color Properties** dialog.
2. Select a bright orange color in the **Color** dialog box by clicking on one of the colored tiles in the **Basic colors** area**.**
3. Click **OK** in the **Color** dialog box**.**
4. Click **OK** in the **Color Properties** dialog box.



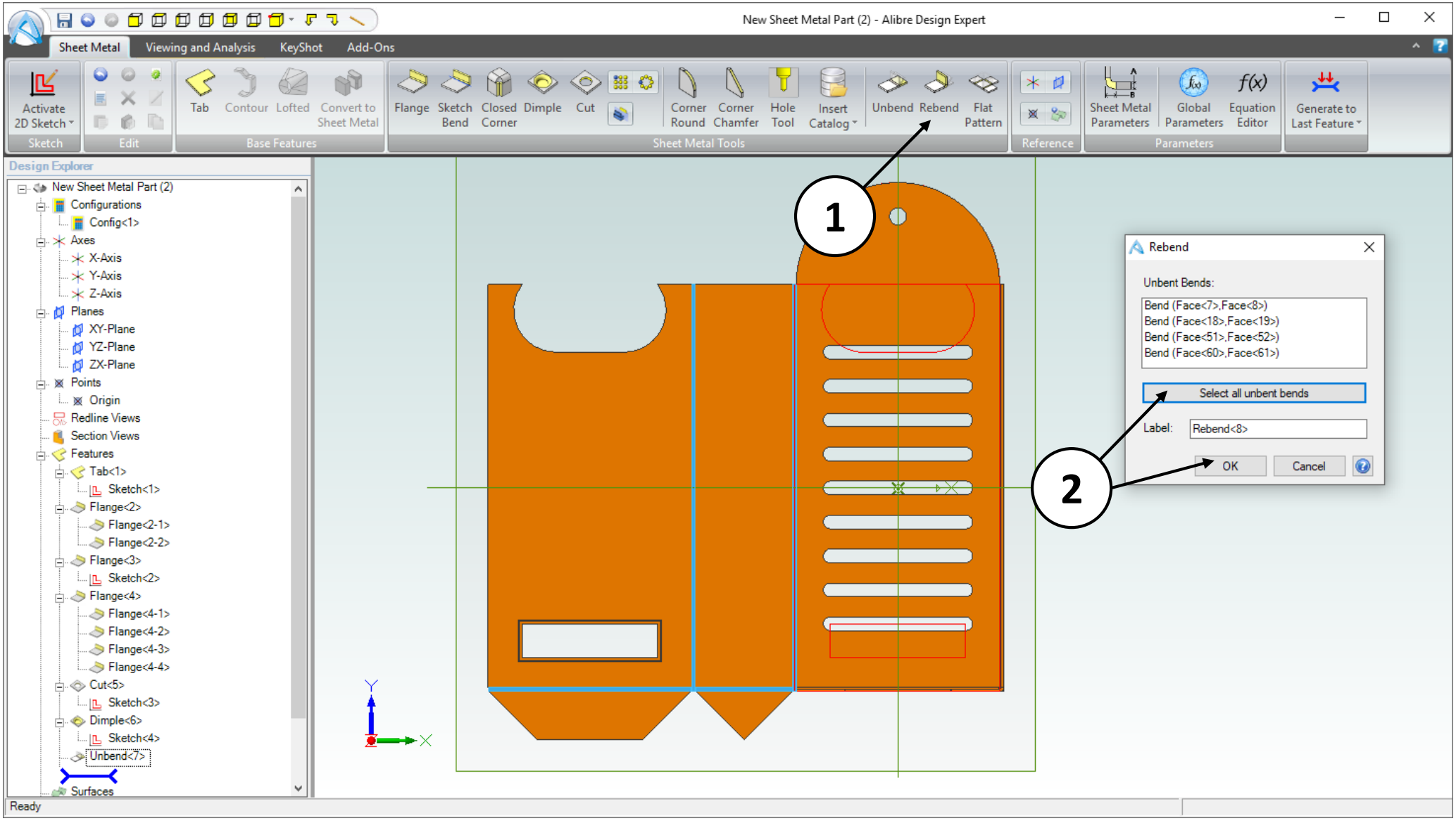
1. Confirm similar results for the **Color Properties** operation.



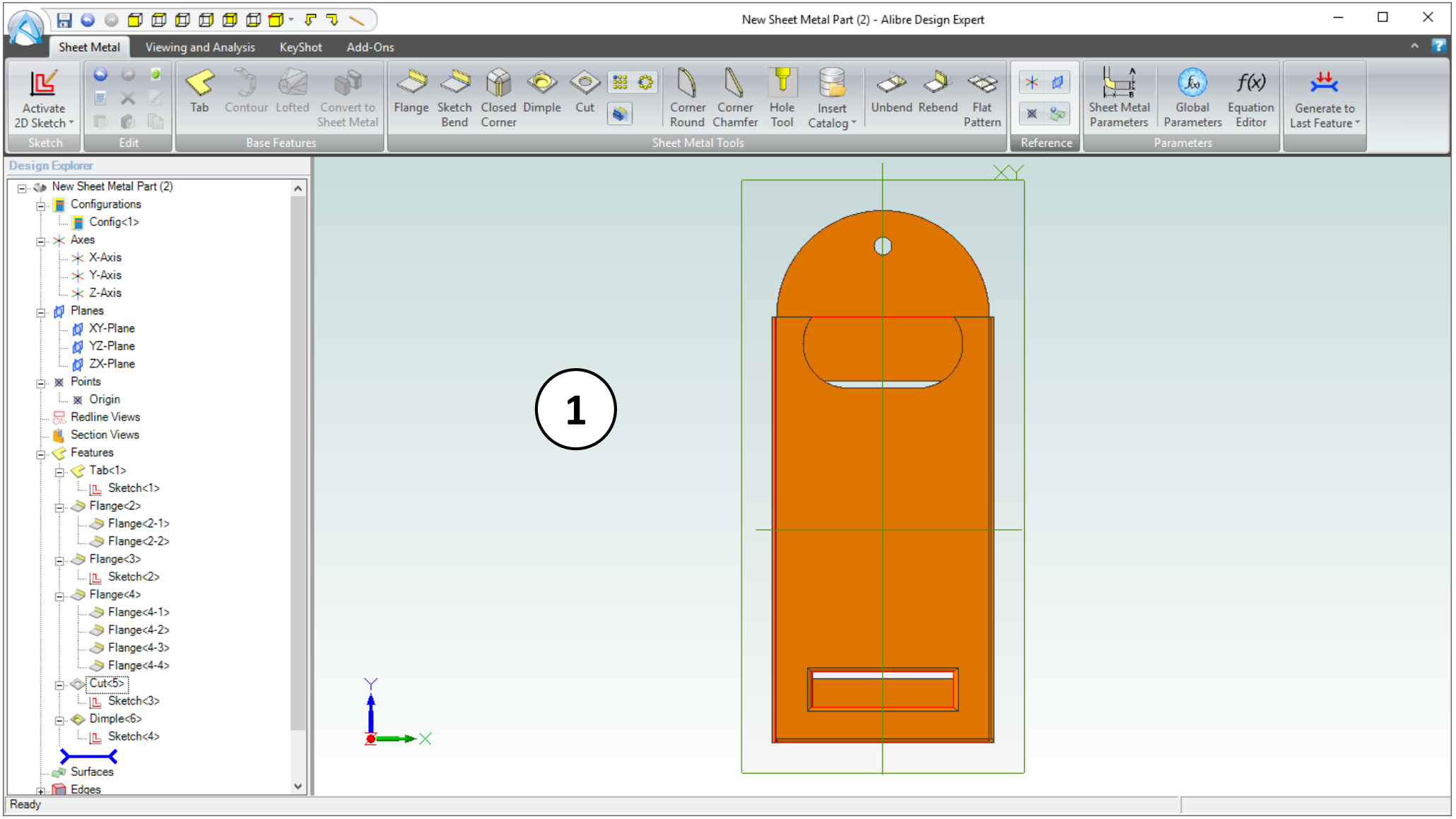
1. Click the **Unbend** tool, and select the back face of the part as the Fixed Face.
2. Click on the **Select All Bends** option.
3. Click **OK**



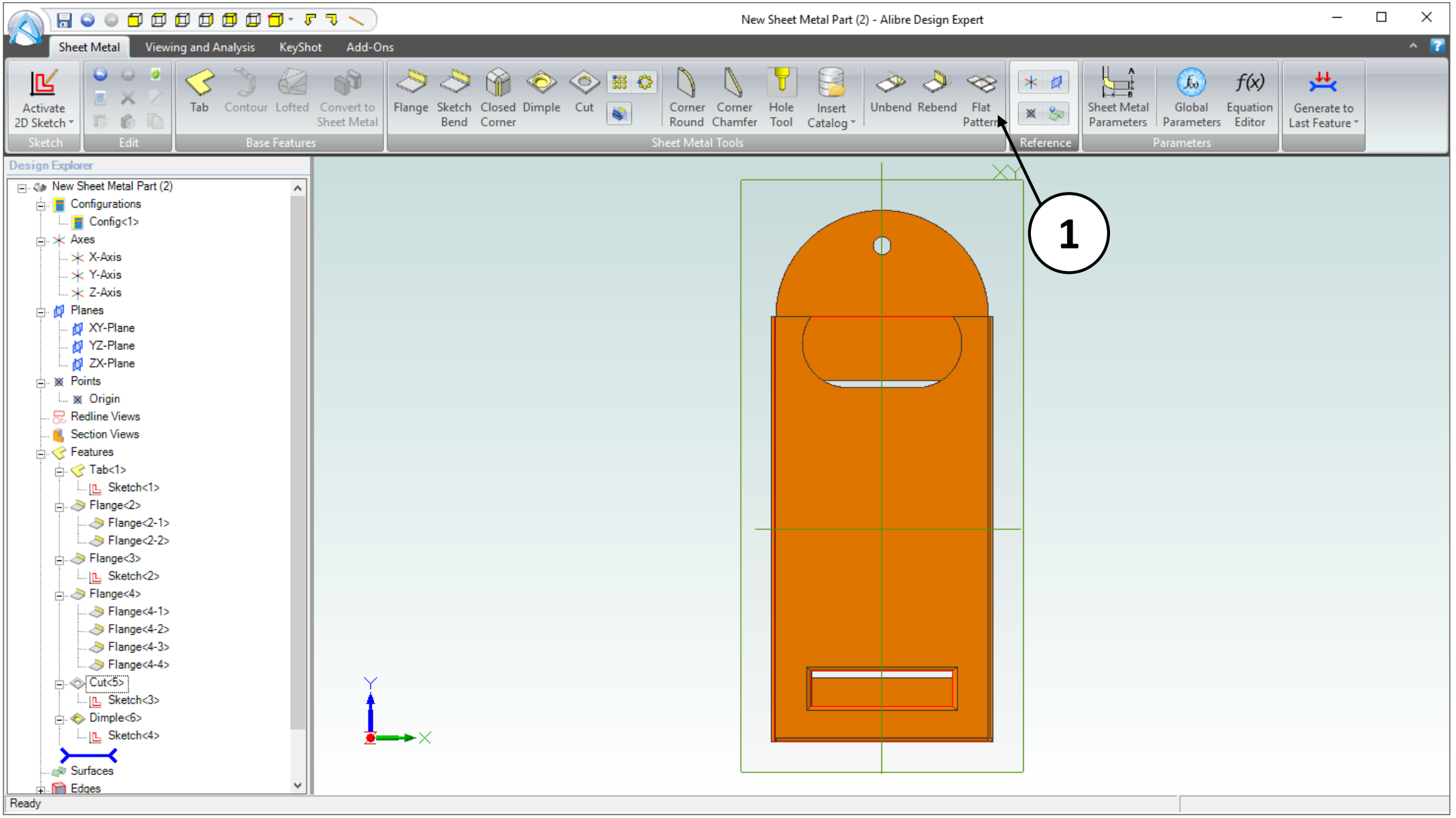
1. Confirm the results of the **Unbend** tool.
2. Blue color lines show the Bending Pattern.
   *Notice that the Pattern represents the bends as it would on the physical object.



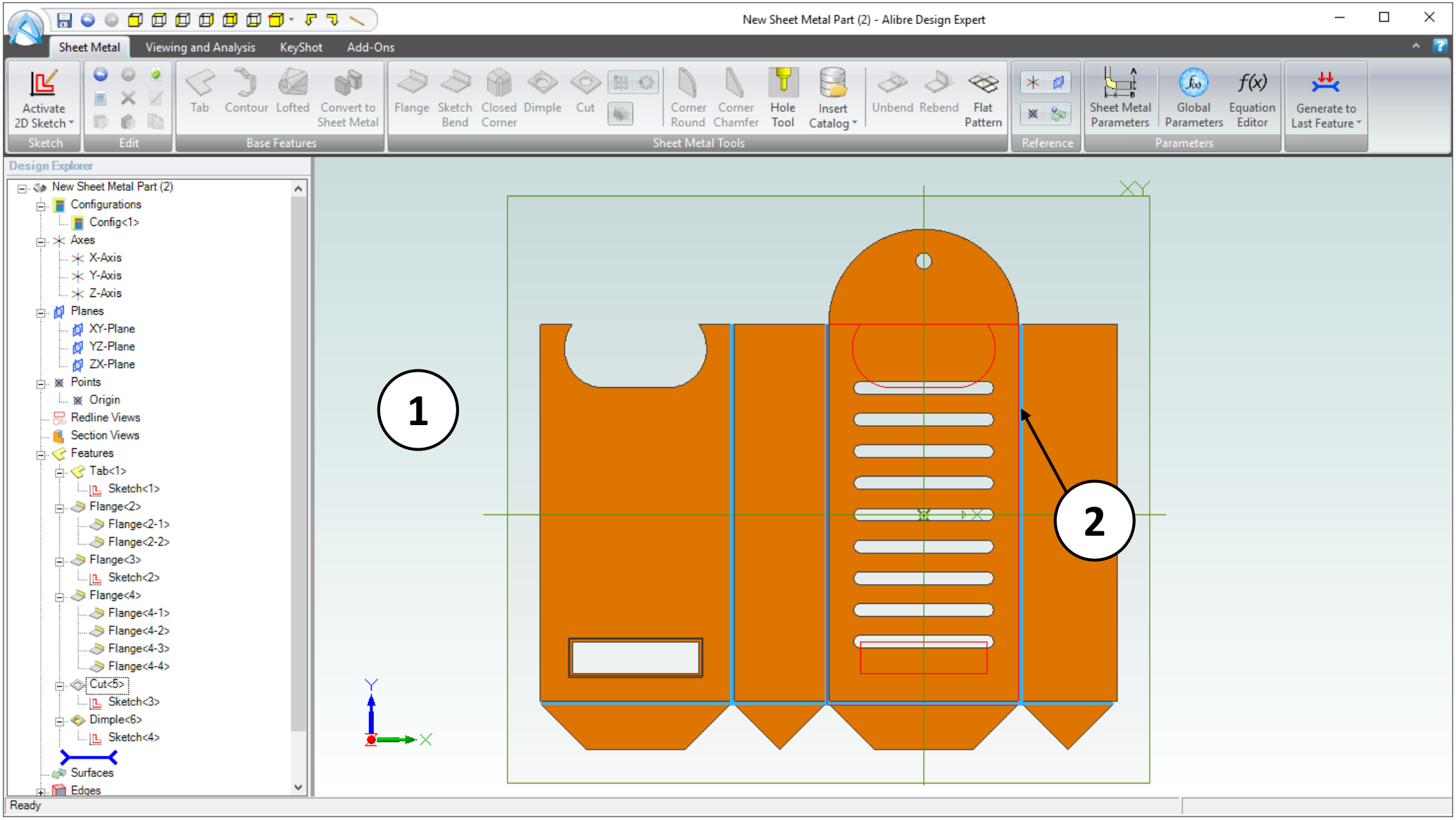
1. Click on the **Rebend** tool.
2. Click **Select all unbent bends**, and click **OK**



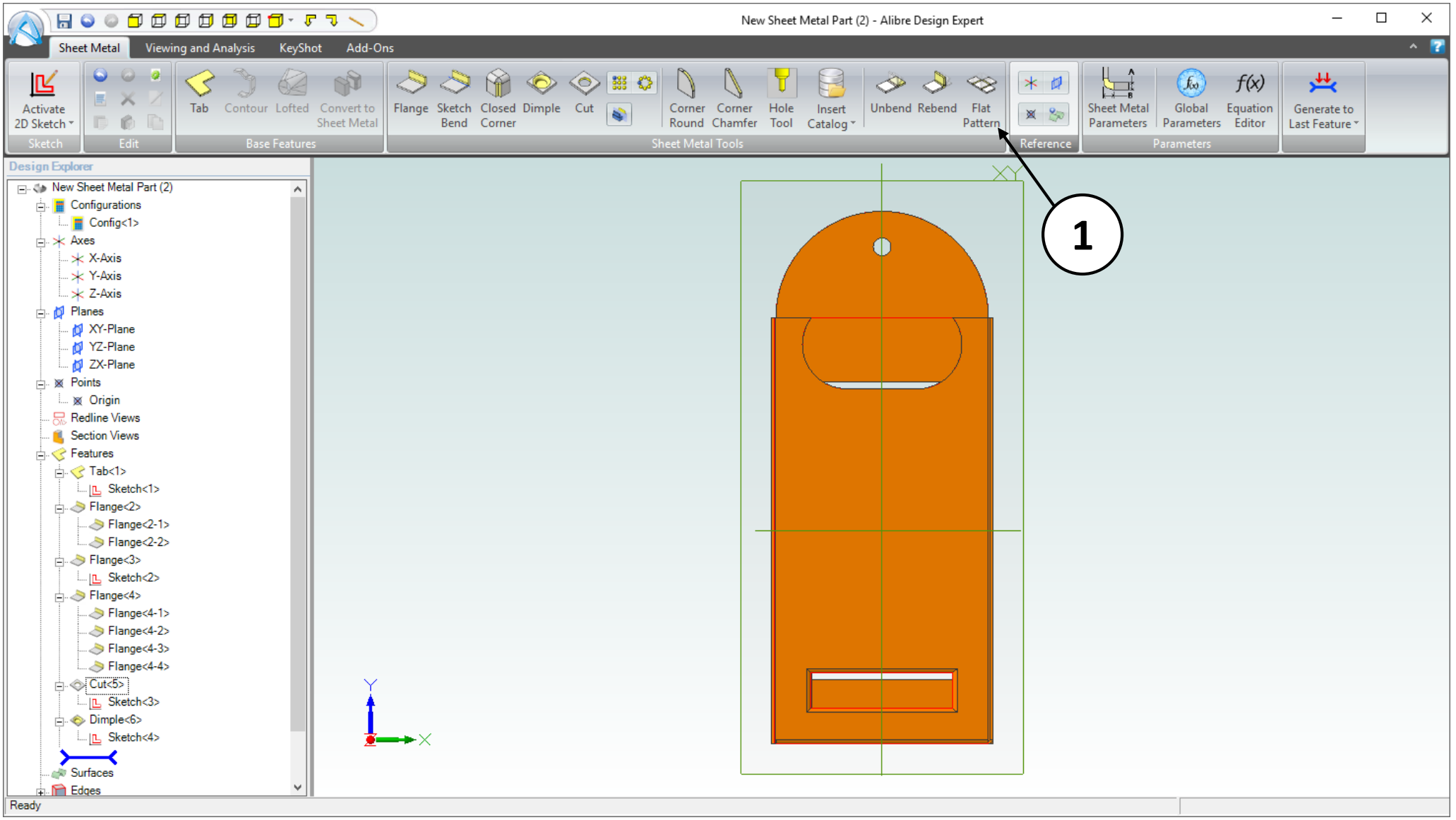
1. Confirm the results of the **Rebend**.



1. Click on the **Flat Pattern** in the **Sheet Metal Tools** tab in the Ribbon.



1. Confirm similar results for the **Flat Pattern** operation.
2. Blue color lines show the Bending Pattern.
   * Notice that the Pattern represents the bends as it would on the physical object.



1. Click **Flat Pattern** again to return the sheet metal part to its closed state.
2. **Save** this Sheet Metal file to your preferred location with the name: **Tool_Holder** (not shown).
3. **Close** the **Sheet Metal** workspace (not shown).
   This concludes the Tutorial.