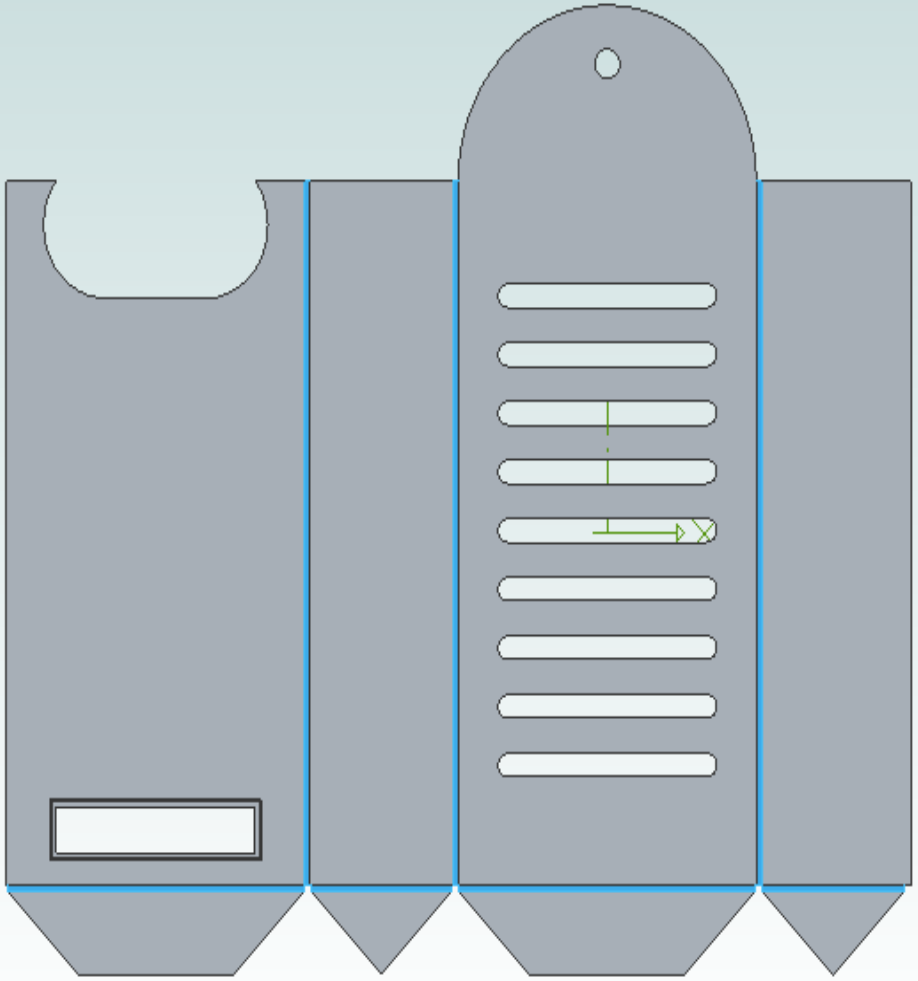
Alibre Design Exercise Manual

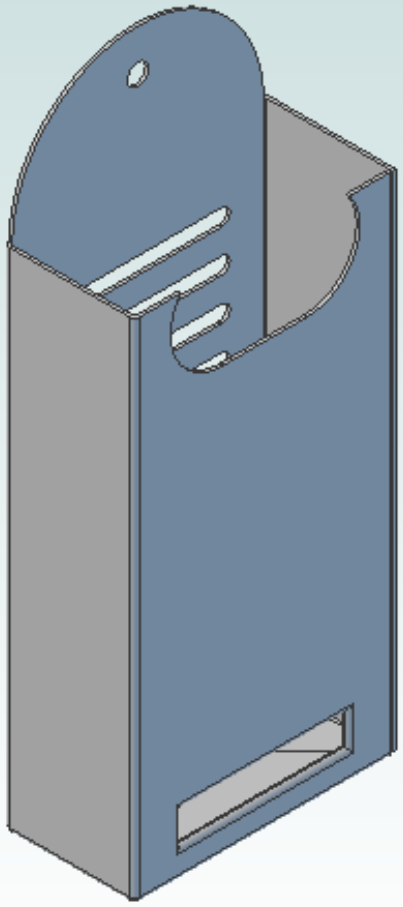
Introduction to Sheet Metal Design

Copyrights

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

www.alibre.com



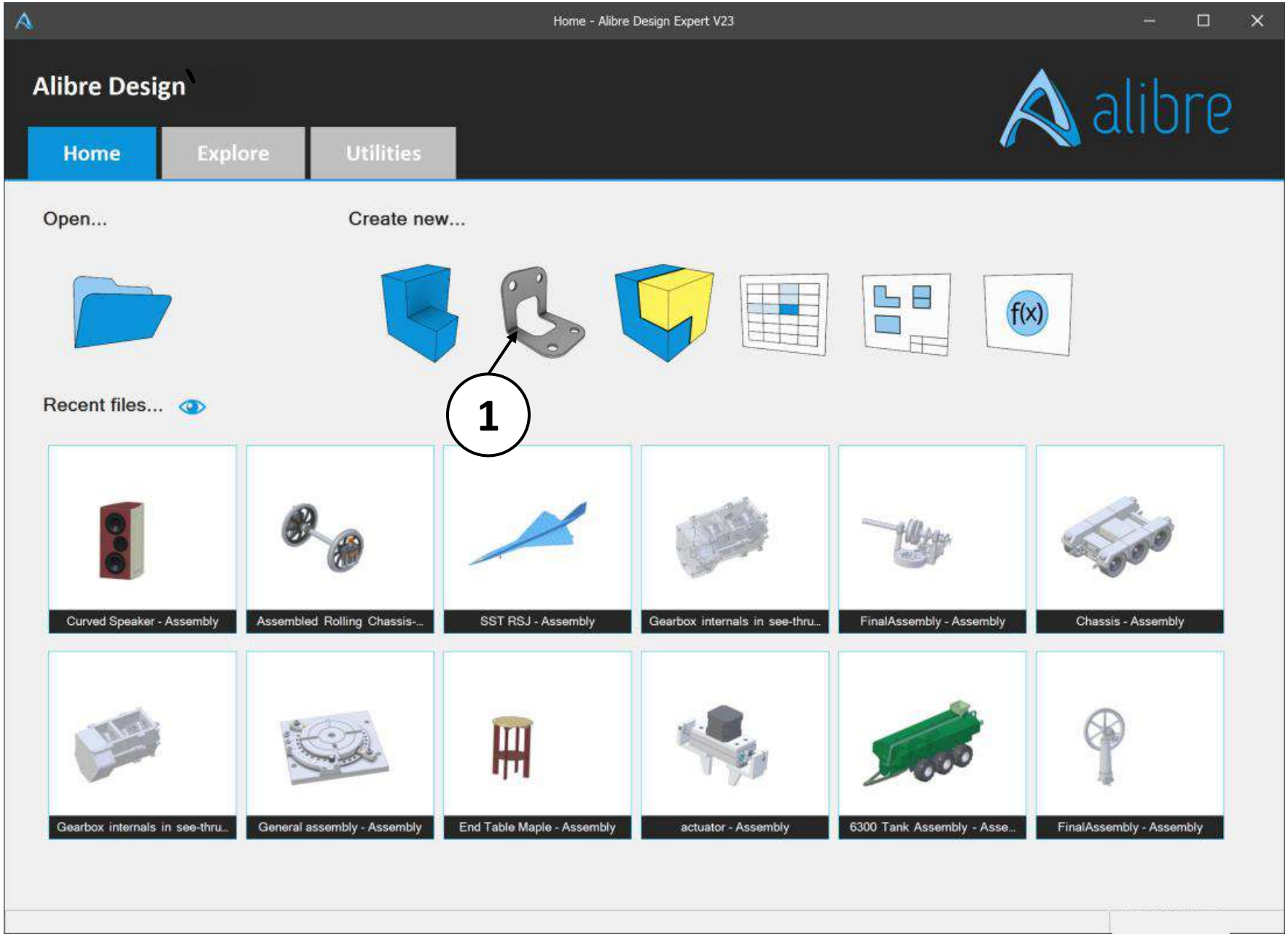


Creating a Tool Holder

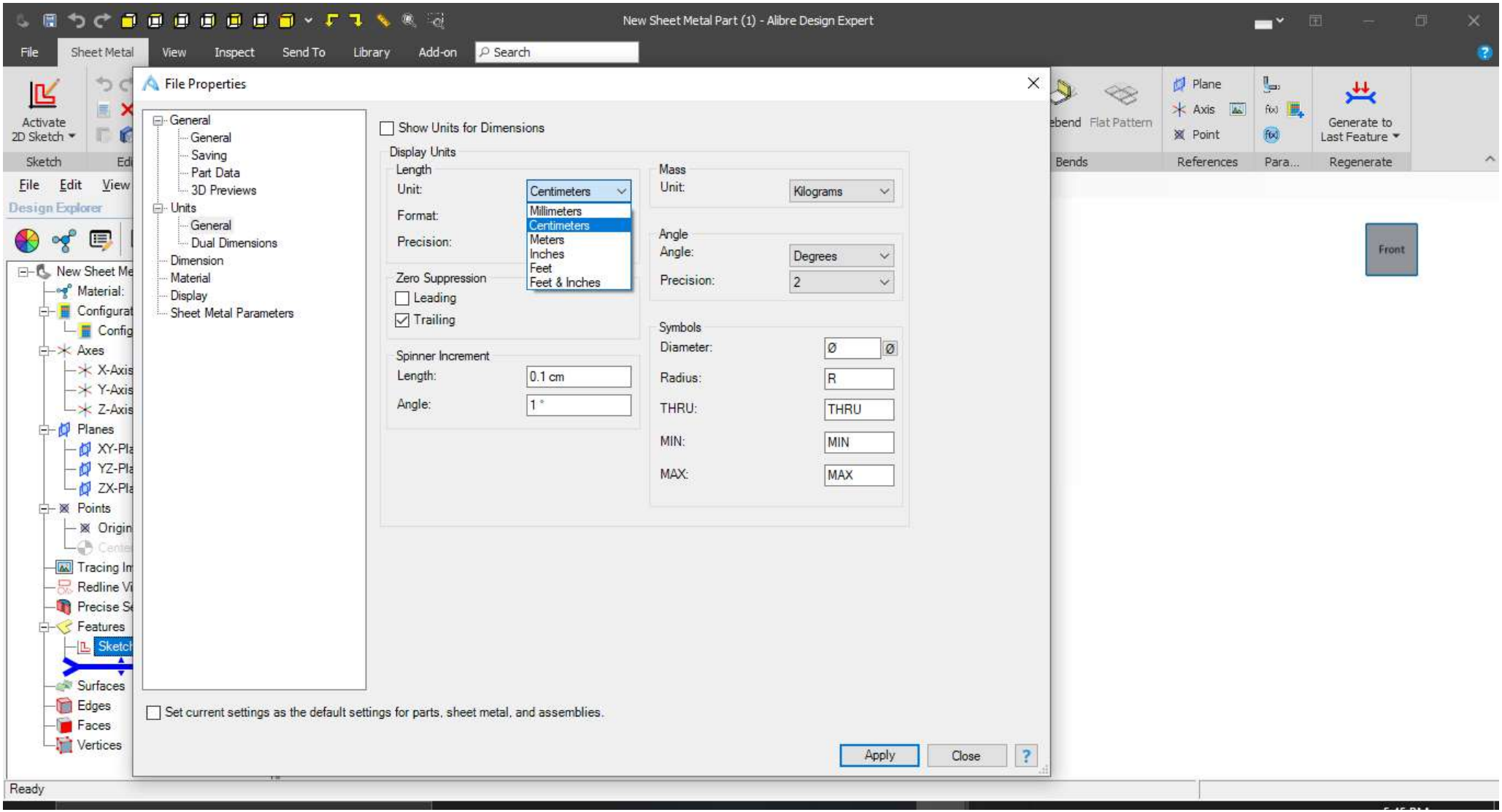
This lesson will take approximately 30 minutes to complete.

You will learn:

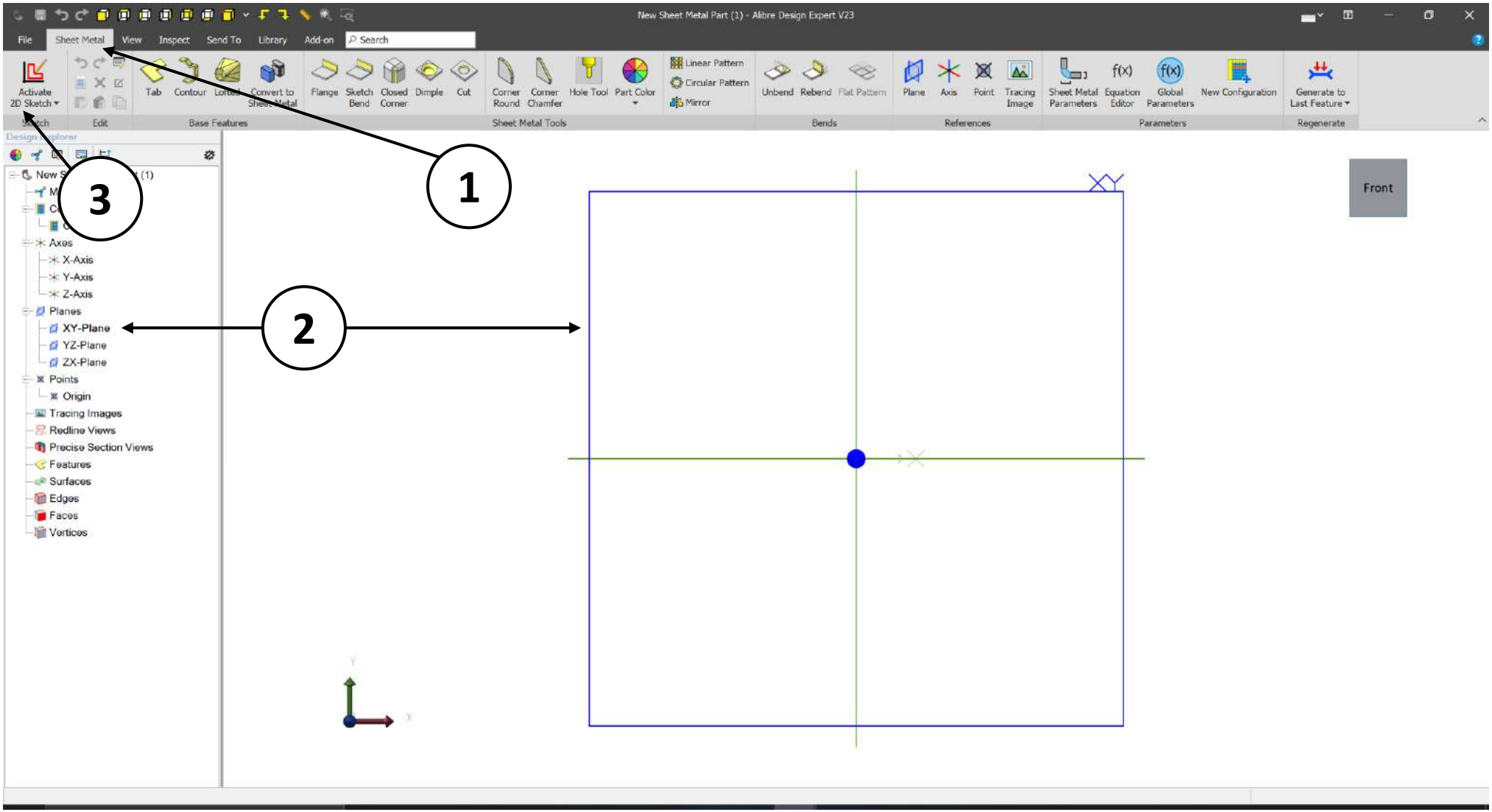
- How to create, edit, and dimension sketch figures.
- How to produce base Sheetmetal features like Tabs and Flanges.
- How to add standard versus miter flanges.
- How to use the Linear Feature Pattern tool.
- How to use the Cut and Dimple Sheetmetal tools.
- How to add Color Properties to your part
- How to Unbend, Rebend, and Flat Pattern your Sheetmetal part.



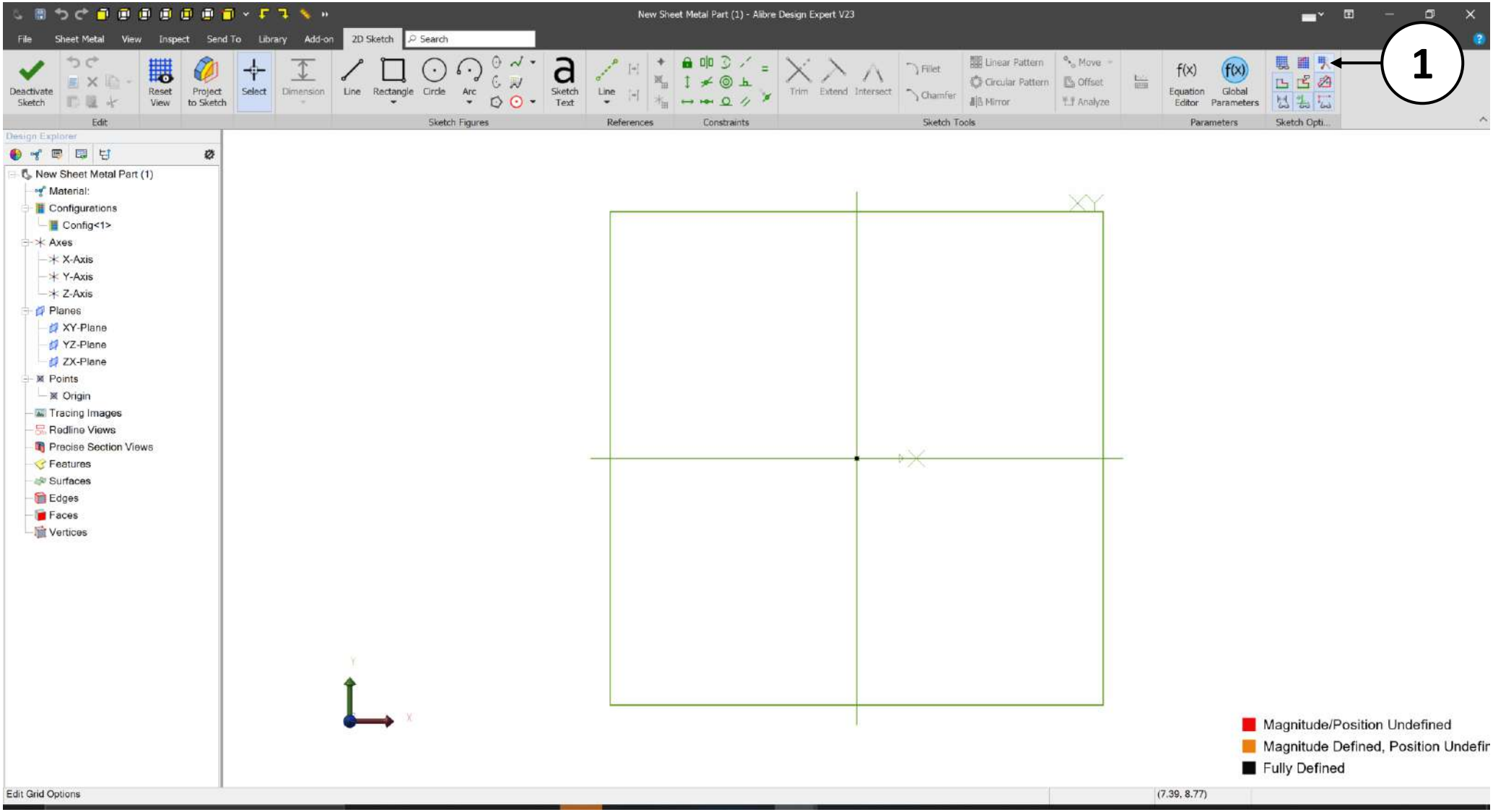
1. Open a new **Sheet Metal Part** workspace from Alibre Design's **Home** window.



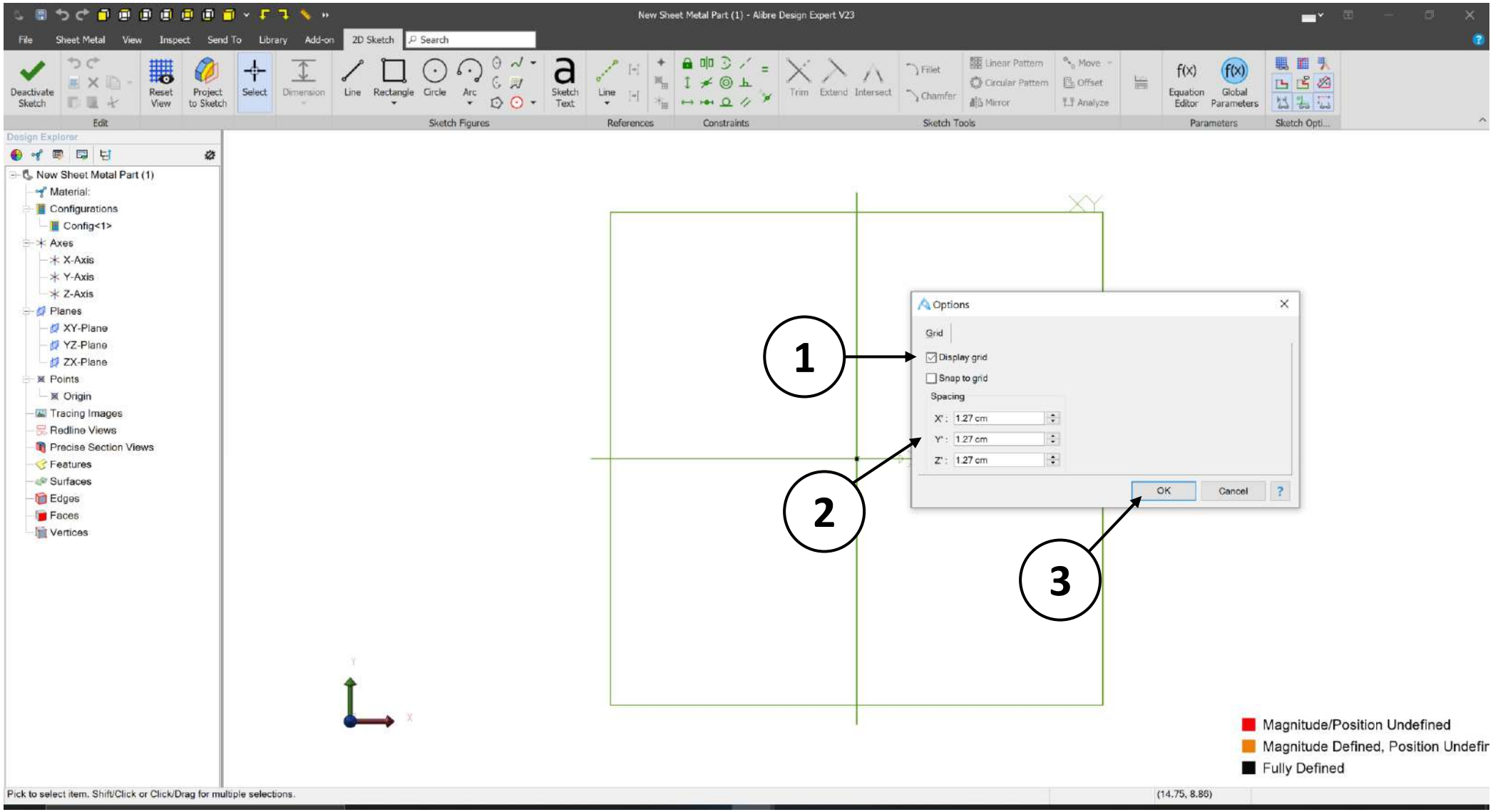
1. Click on the **File** tab.
2. Go to **Properties.**
3. Go to **Units>General** and select **Centimeters** from the dropdown.



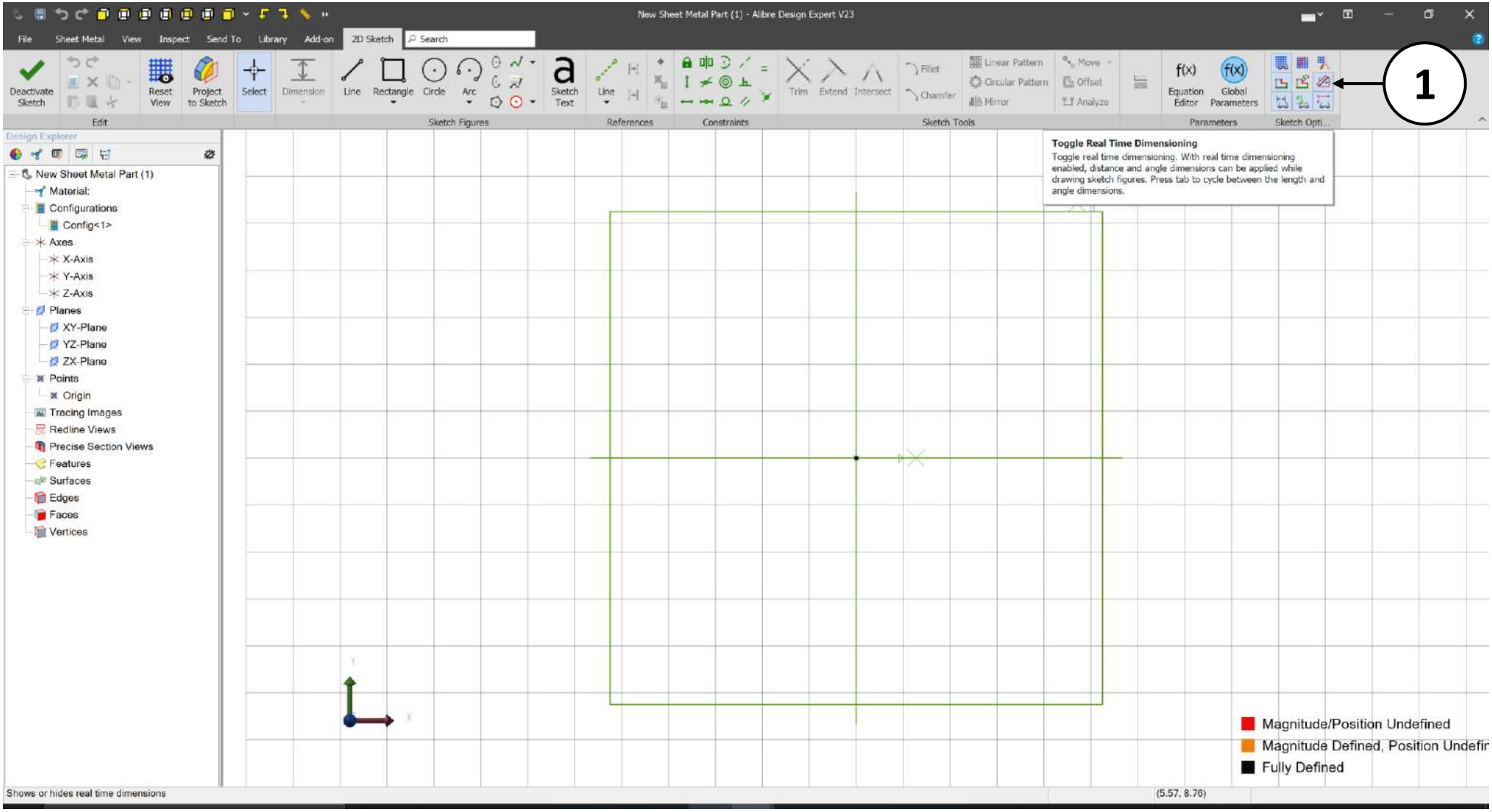
1. Click on the **Sheet Metal** tab in the ribbon, if necessary.
2. Click on the **Activate 2D Sketch** icon.
3. Click on the **XY Plane** in the **Design Explorer** or the **Work Area**.



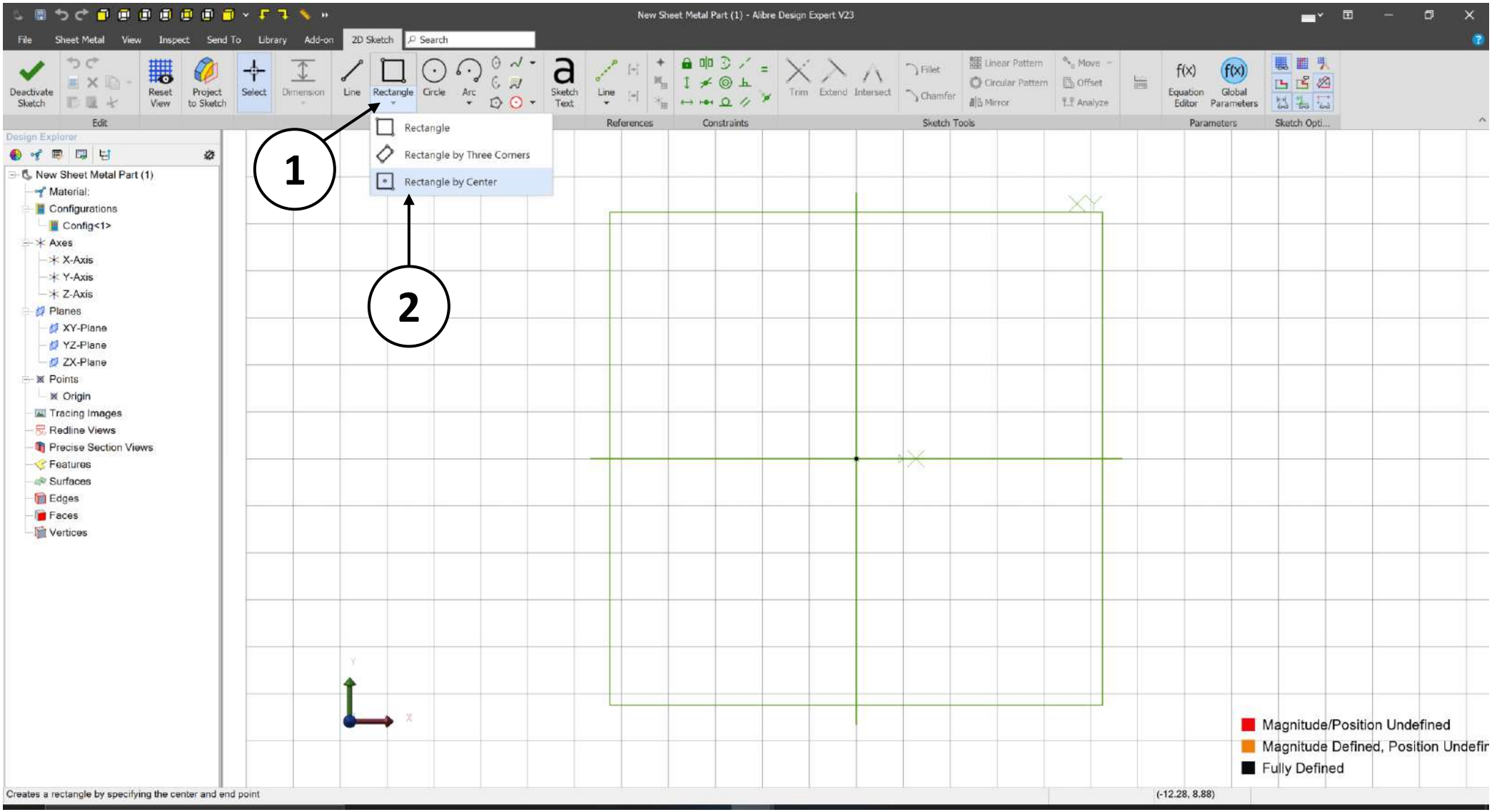
1. Select the **Grid** options.



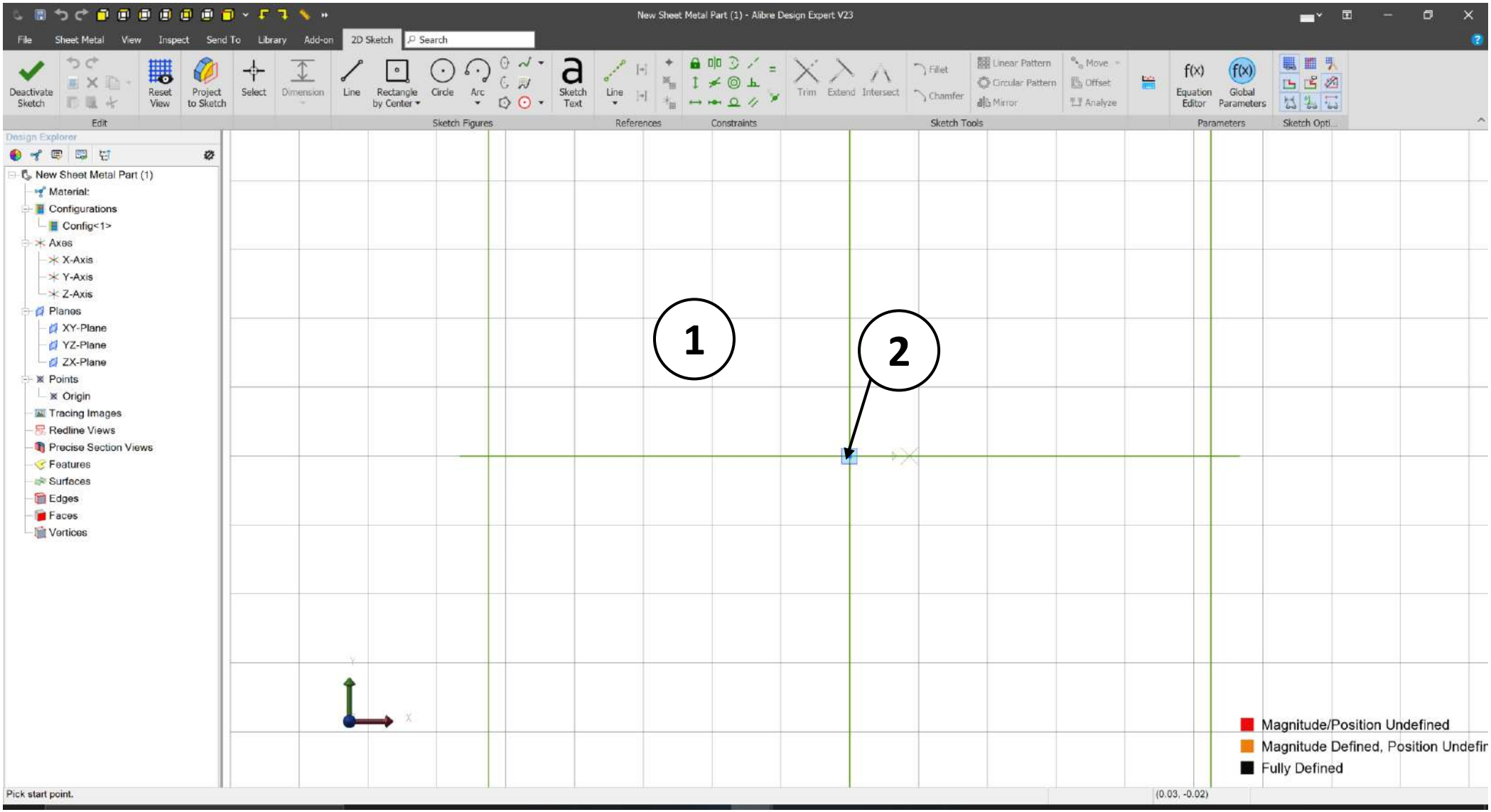
1. Enable the **Display Grid** option if it is unchecked.
2. If necessary, change the values to 1.27cm for X, Y, and Z.
3. Press the **OK** button.



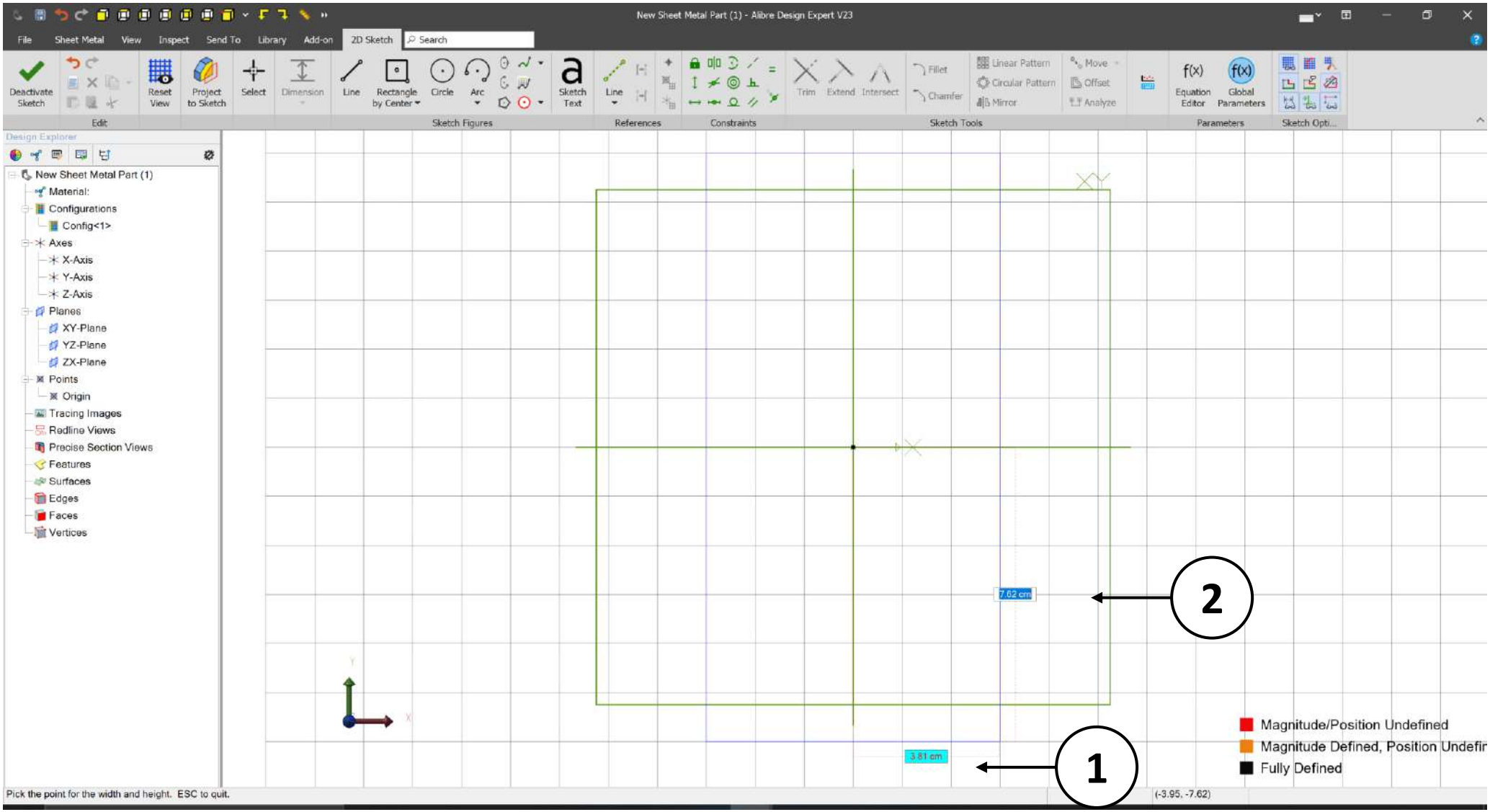
1. Make sure **Real Time Dimensioning** is turned on.



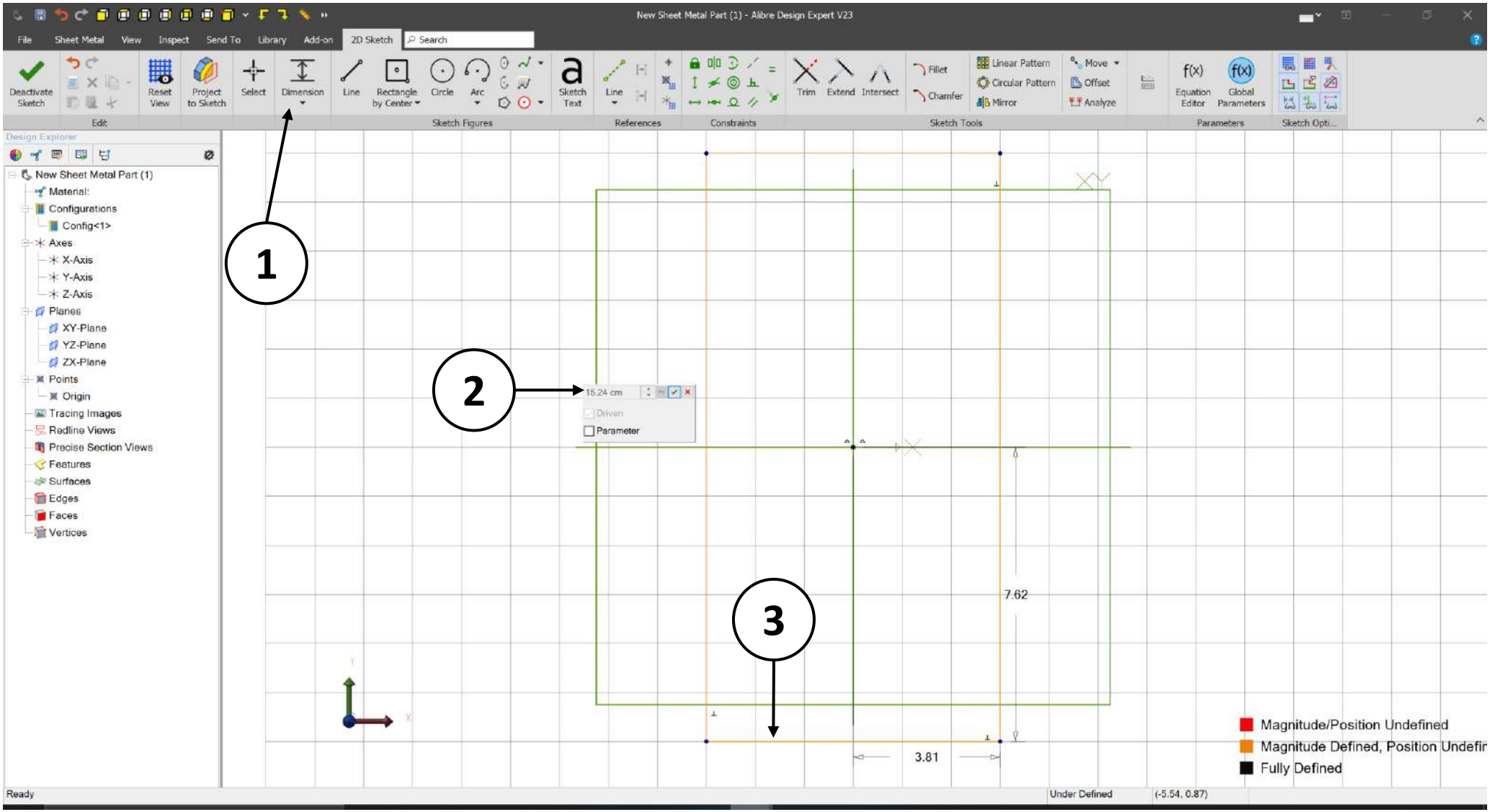
1. Click on the dropdown menu of **Rectangle** tool on the **Sketch Figures** tab in the ribbon.
2. Select the **Rectangle by Center** from the List.



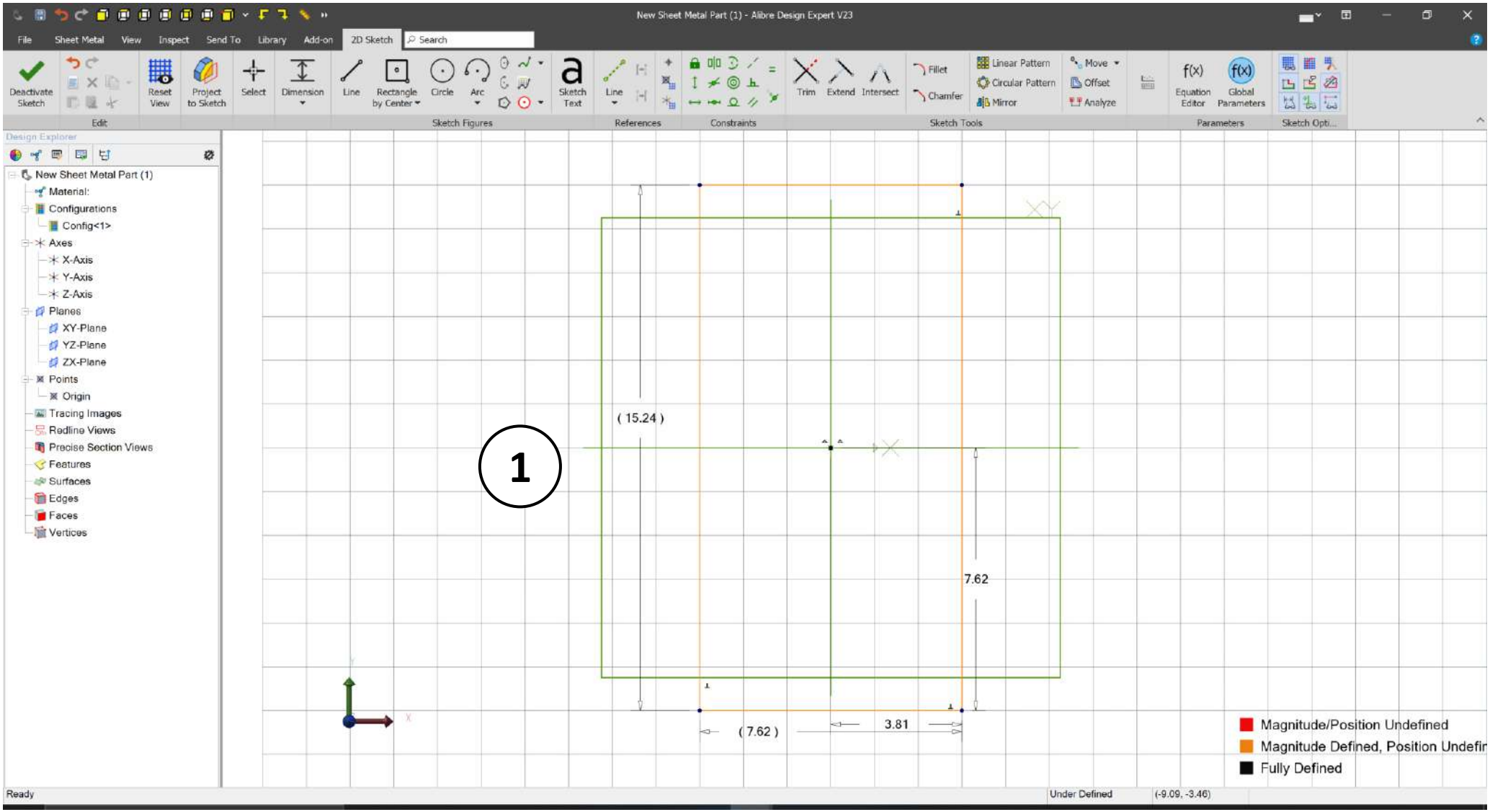
1. Position the mouse cursor near the **Origin** point. Roll the mouse wheel forward to zoom in. The view of the work area should be like the screen capture.
2. Position the mouse cursor over the **Origin** point. The point's color and cursor tool tip changes confirm that the mouse cursor is positioned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



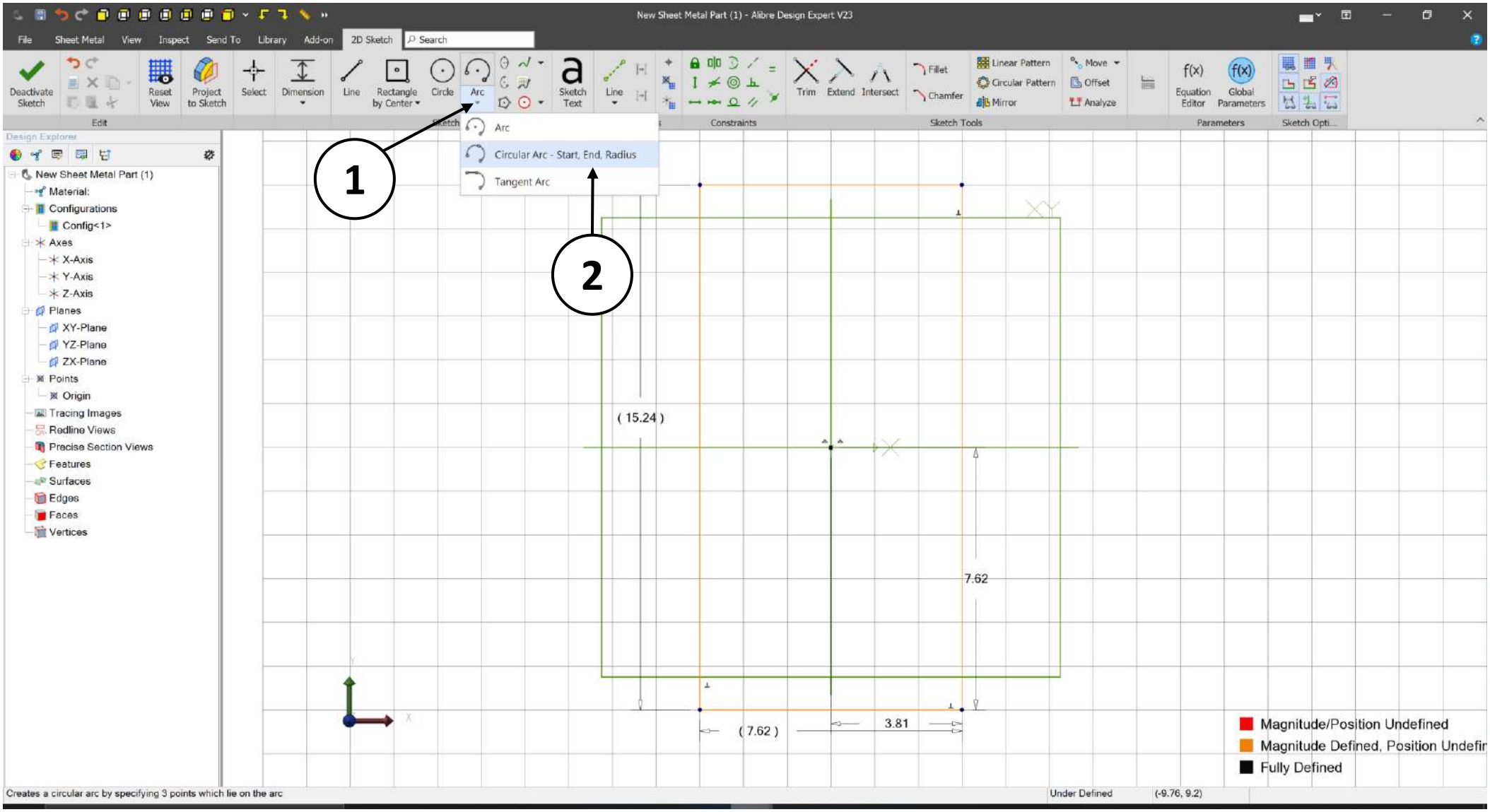
1. Move the mouse cursor horizontally to the right, input a dimension of **3.81cm** for the horizontal distance from the center of the rectangle to its edge, and then hit **Tab**.
2. Enter the value of **7.62cm** vertical distance from the center of the rectangle to the bottom and press **Enter**.



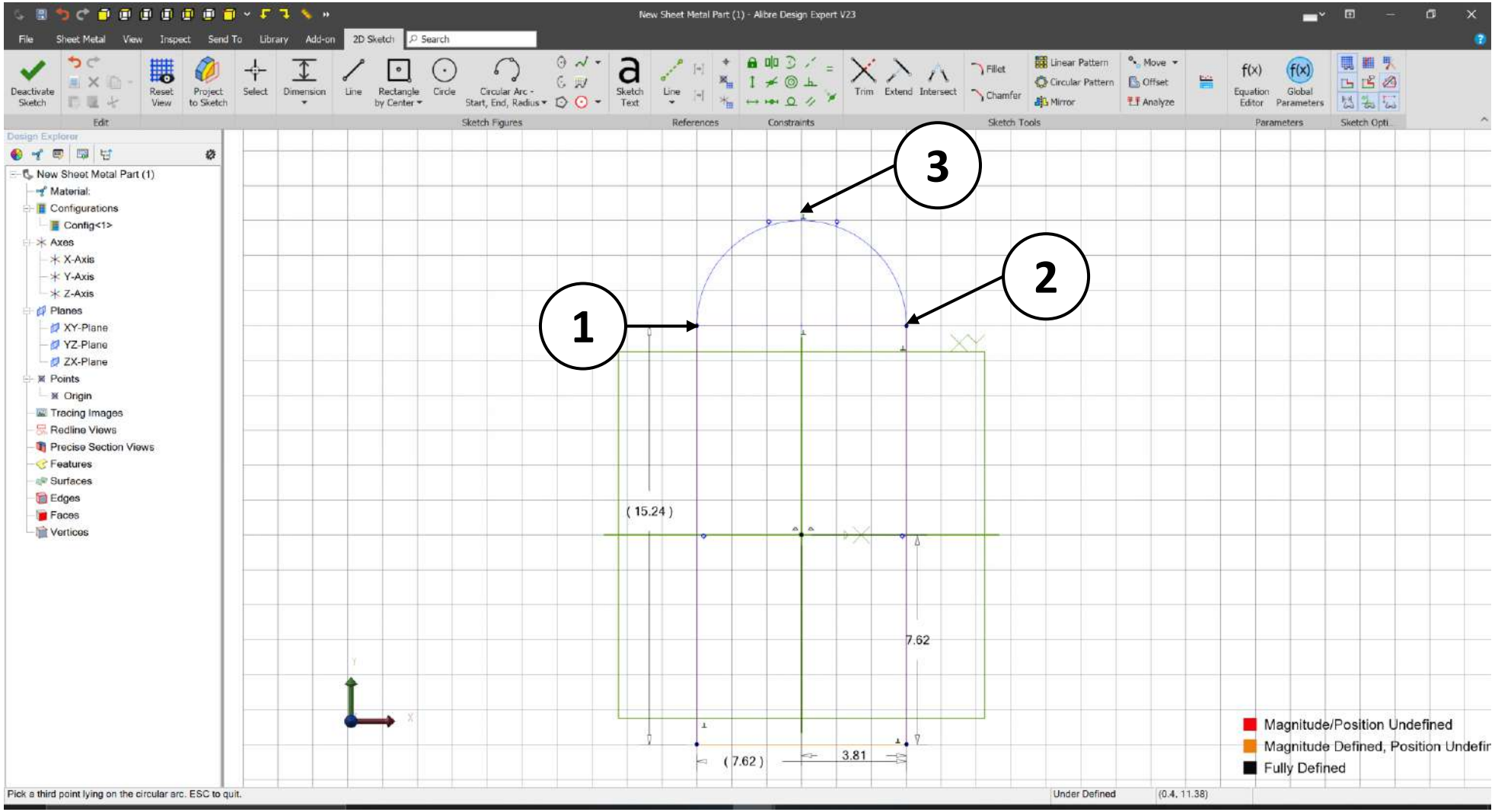
1. Click on **Dimension**.
2. Select the left or right vertical line of Rectangle, then click on the workspace location where you want to place the dimension and confirm the value of **15.24cm** and press **Enter**.
3. Select the bottom line of the Rectangle and click on the workspace location to place the dimension and confirm the value is **7.62cm** and click the **Green Checkmark** or press **Enter**.



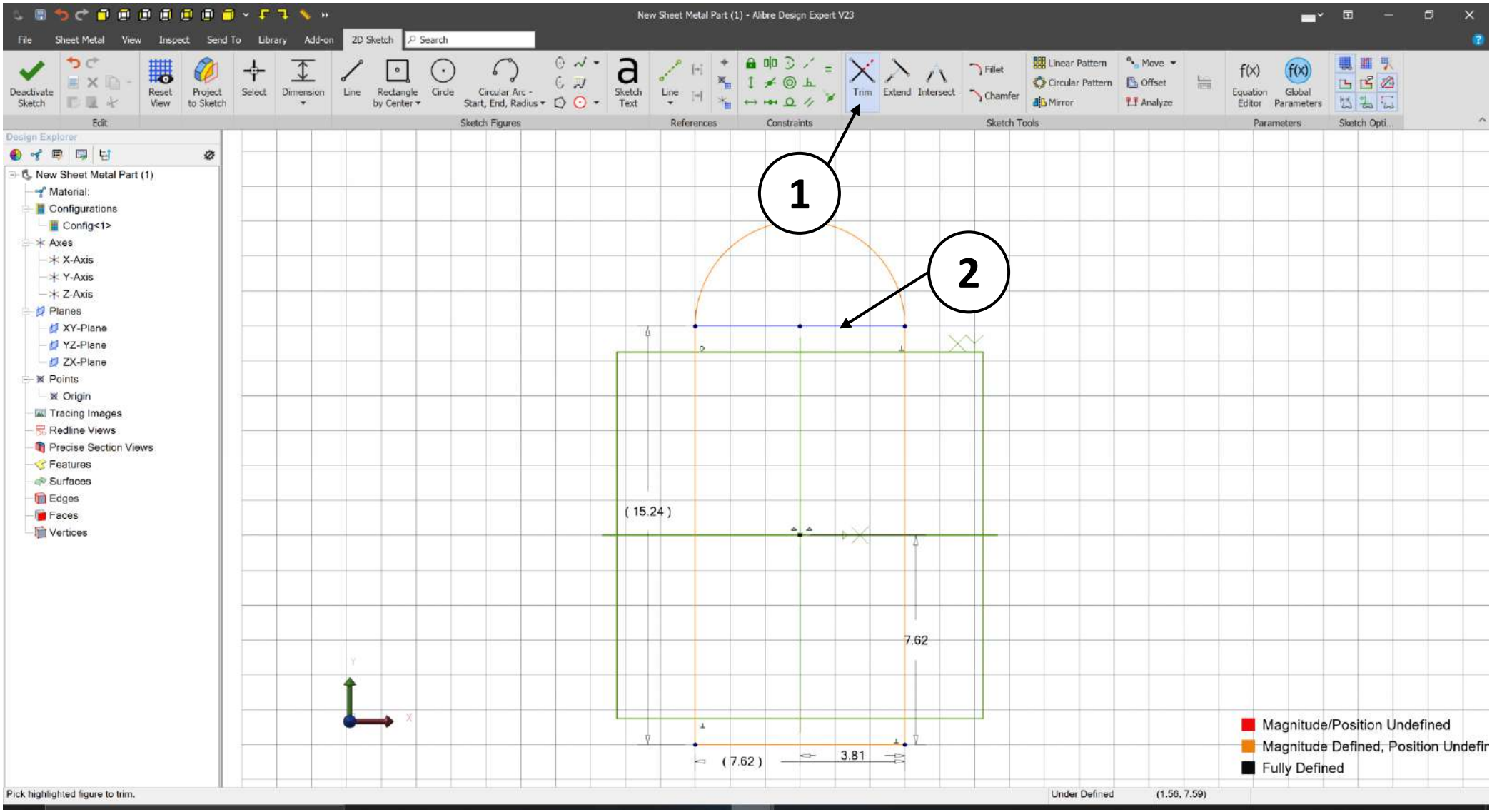
1. Completed rectangle view of the work area should be like the image above.



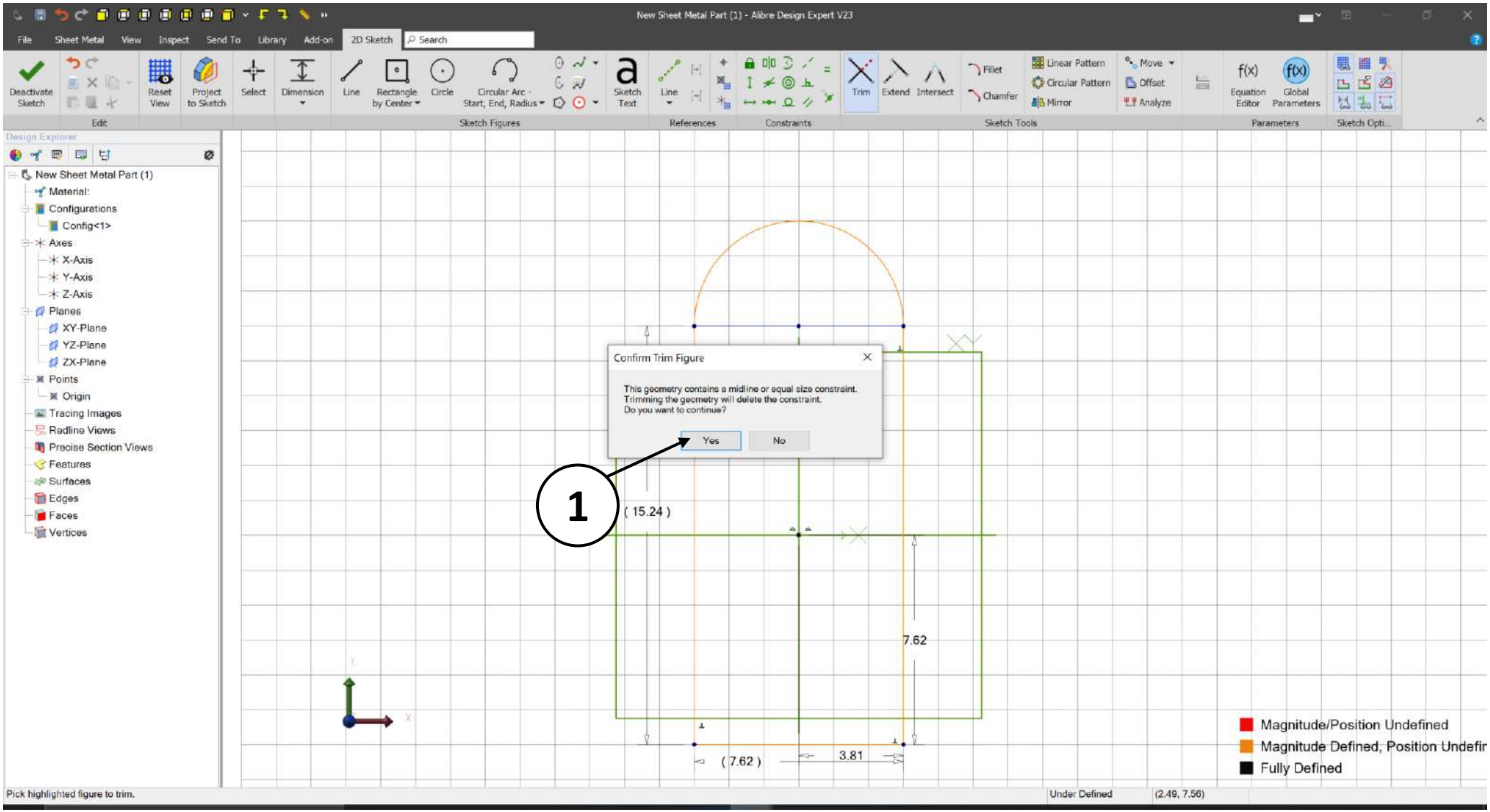
1. Click on the dropdown menu of **Circular Arc** tool on the **Sketch Figures** tab in the ribbon.
2. Select the **Circular Arc-Start, End, Radius** option from the List.



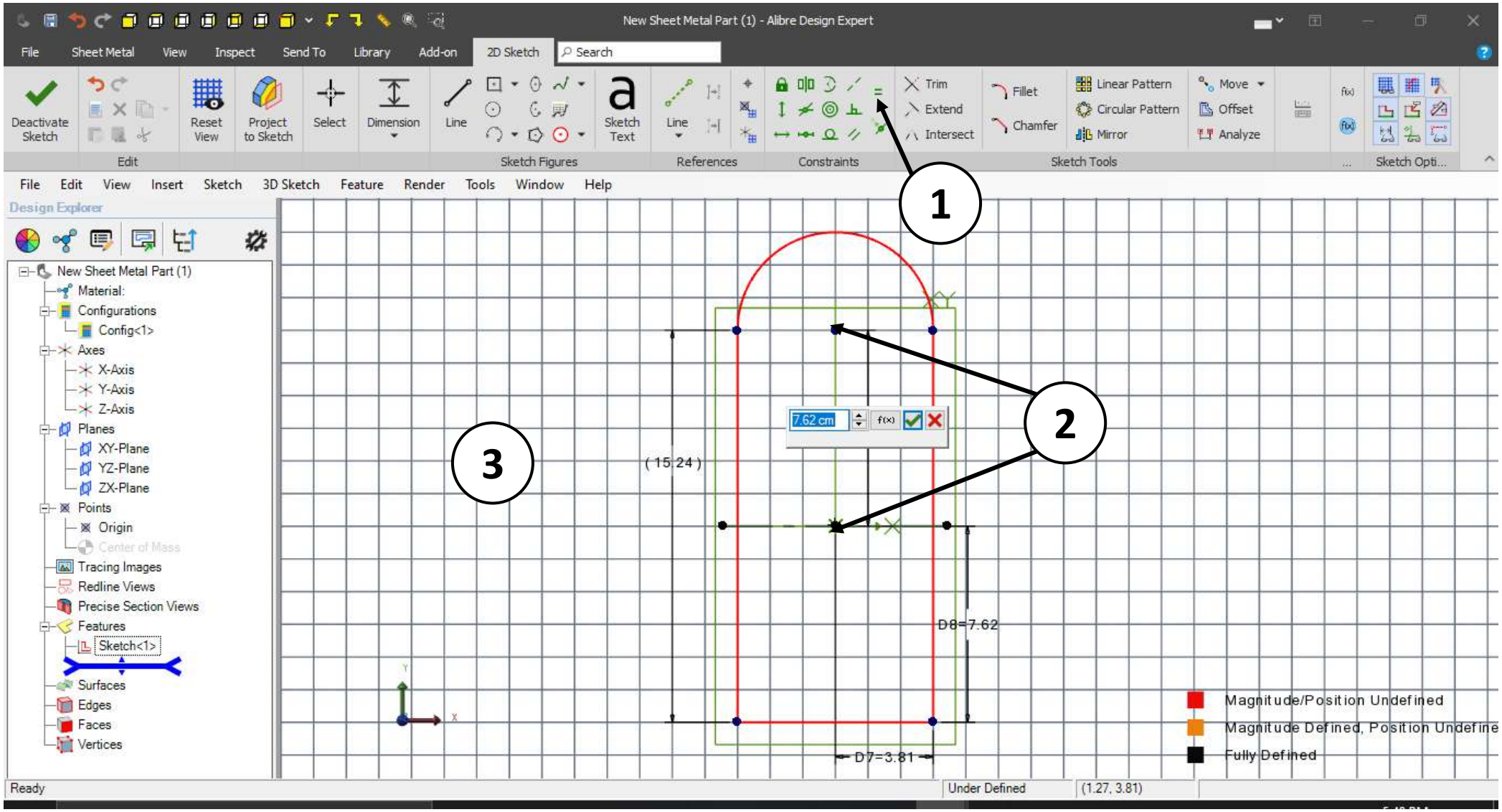
1. Move the mouse cursor over the top-left sketch node on the rectangular sketch **(3.81cm to the left of the origin & 7.62cm above the Origin).** *Click and release* to create the Start point of the Arc.
2. Move the mouse cursor over the top-right sketch node on the rectangular sketch. *Click and release* 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.



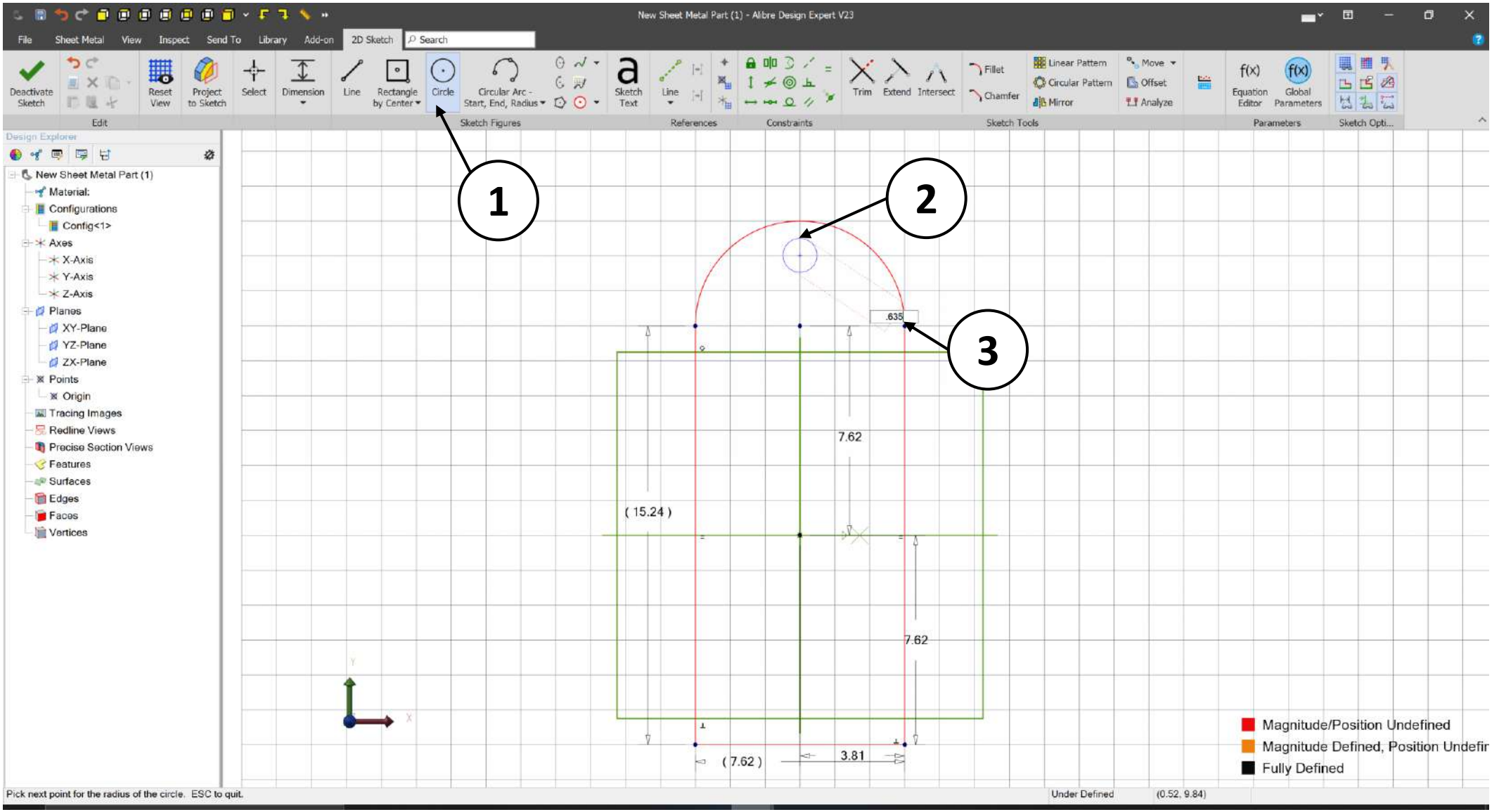
1. Click on the **Trim** tool on the **Sketch Tools** tab in the ribbon.
2. Click on the top horizontal sketch line to execute the **Trim**.



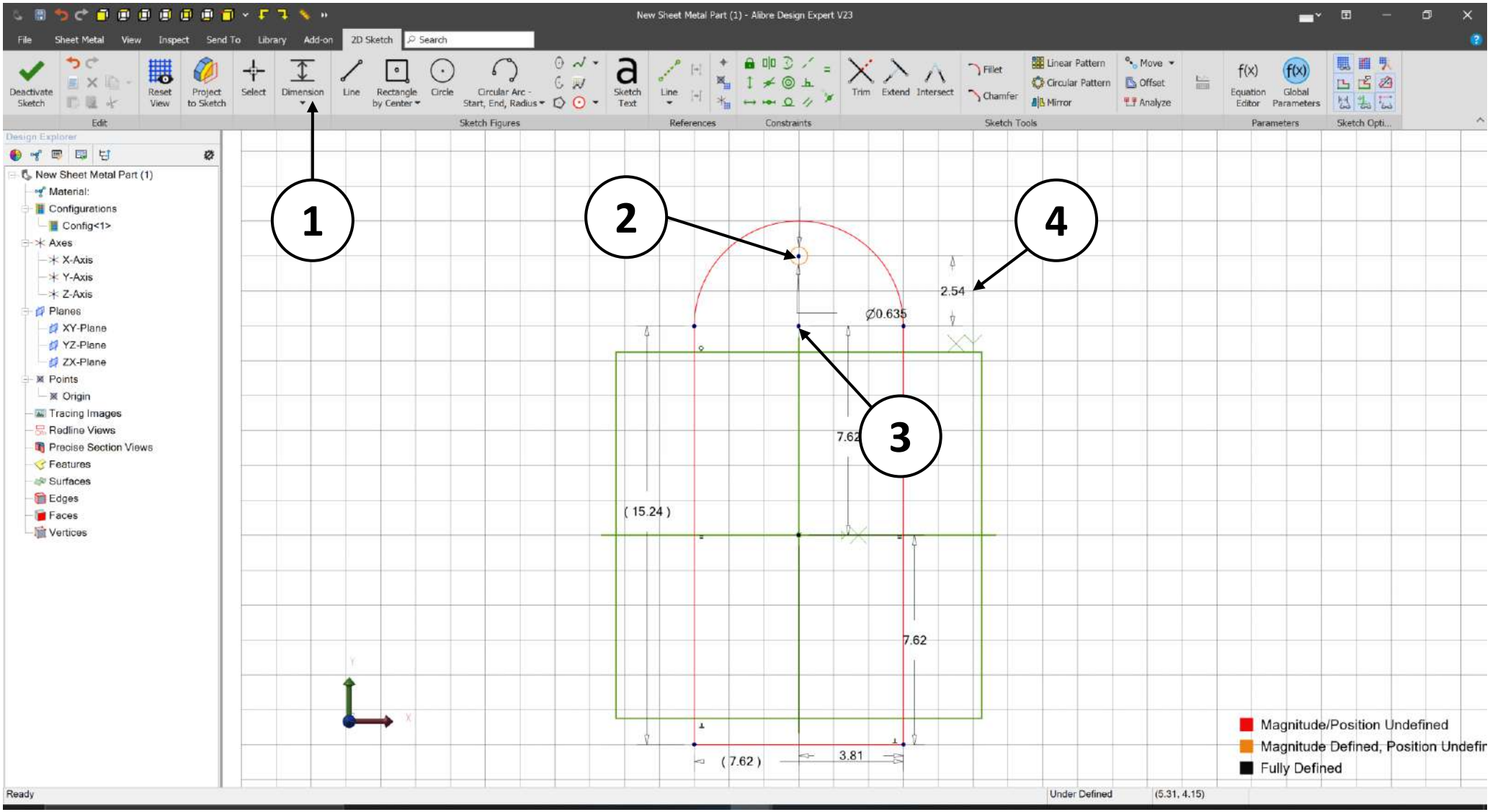
1. Click **Yes** on the pop-up window to complete the process.



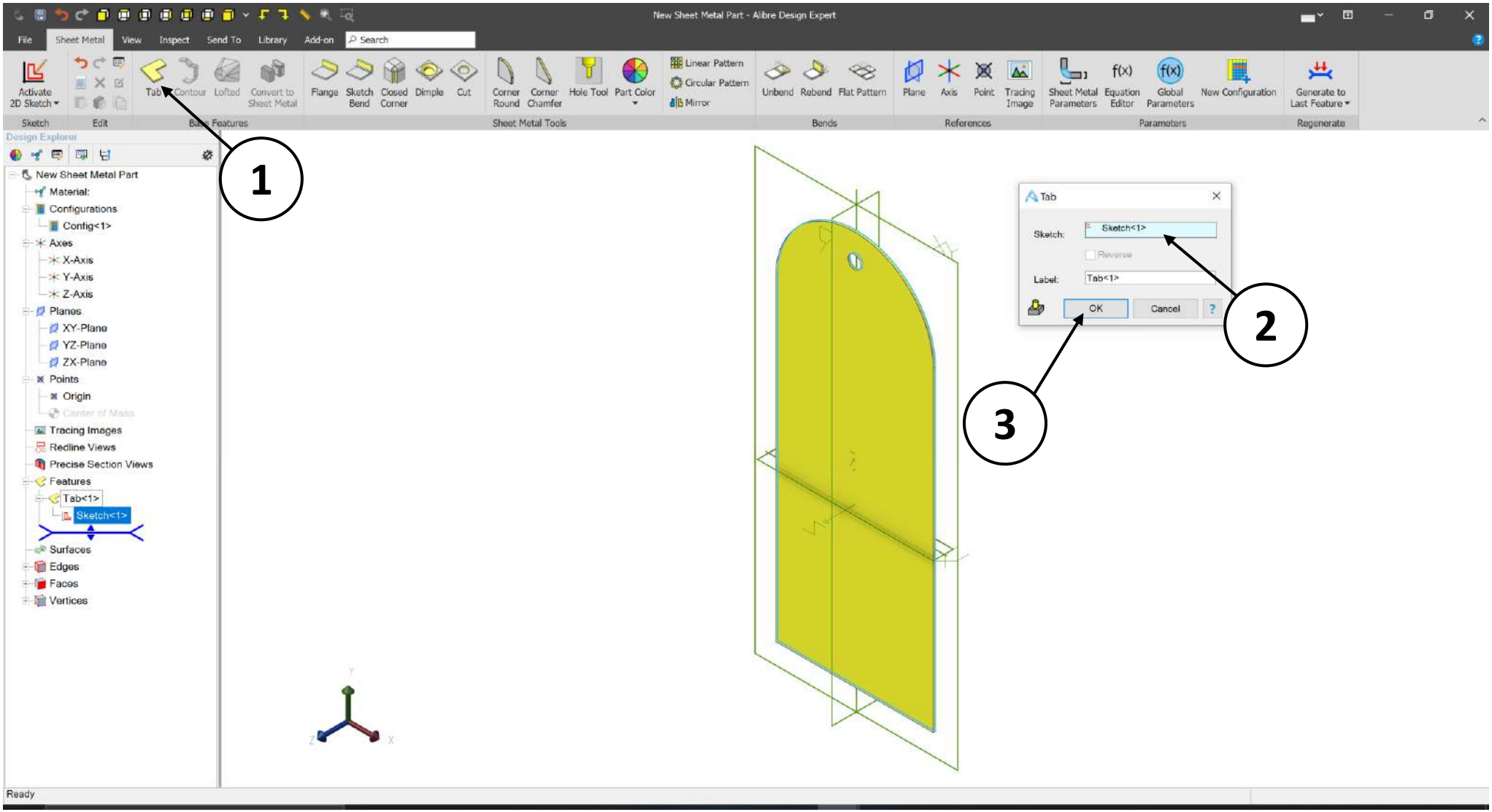
1. Use the **Equal Constraint** for the two long sides of the rectangle so they will remain the same length.
2. Use the Dimension tool to apply a **7.62 cm** dimension between the **Center Point** of the arc and the **Origin**.
3. Completed Sketch view of the work area should be similar to the screen capture.



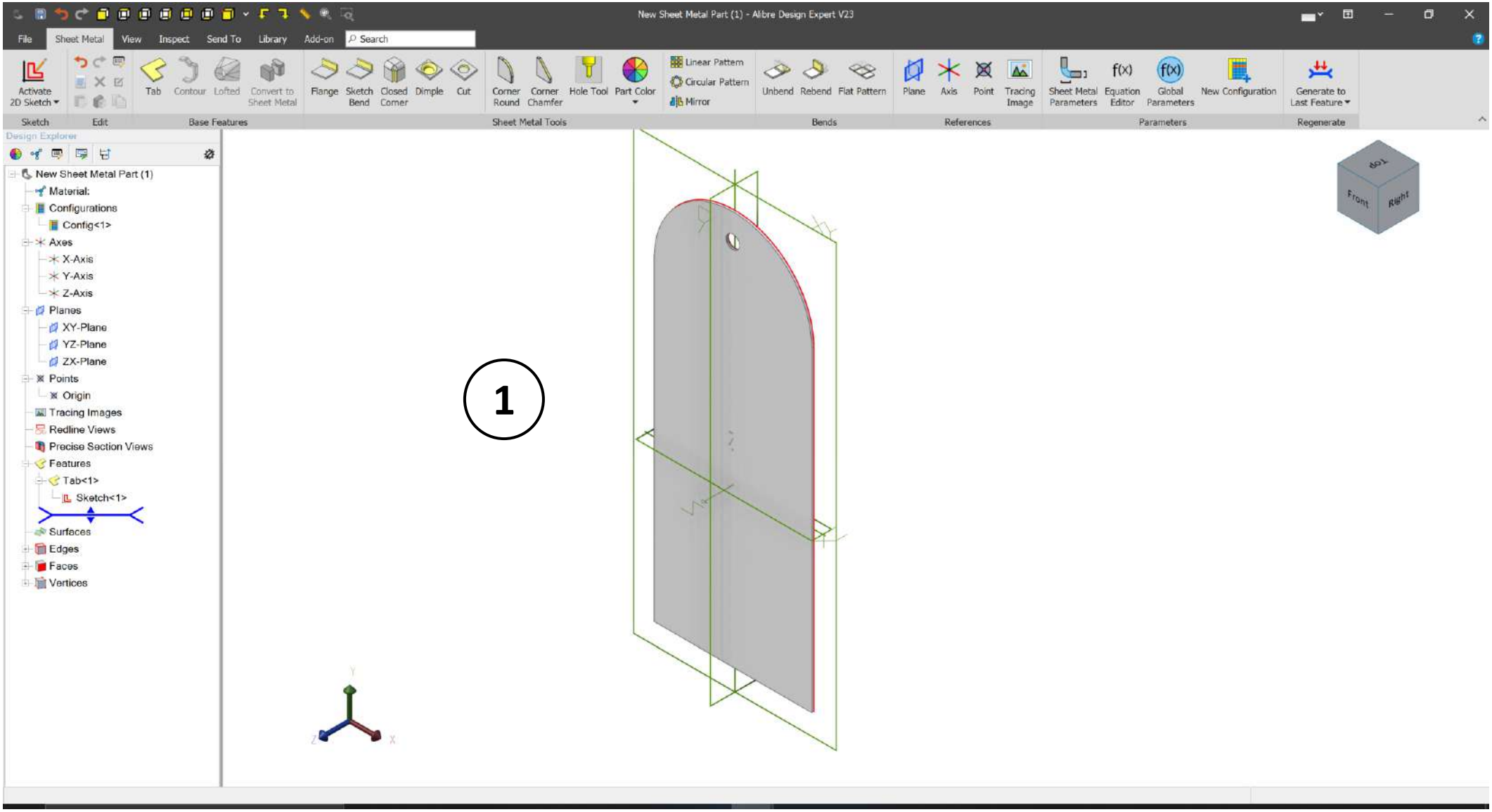
1. Click on the **Circle** tool on the **Sketch Figures** tab in the ribbon.
2. Move the mouse cursor close to the position shown **(10.06cm over the Origin).** *Click and release* to create the Center point of the **Circle**.
3. Enter a value of **.635cm** for the circle's diameter, and then hit **Enter** on your keyboard to apply the value.



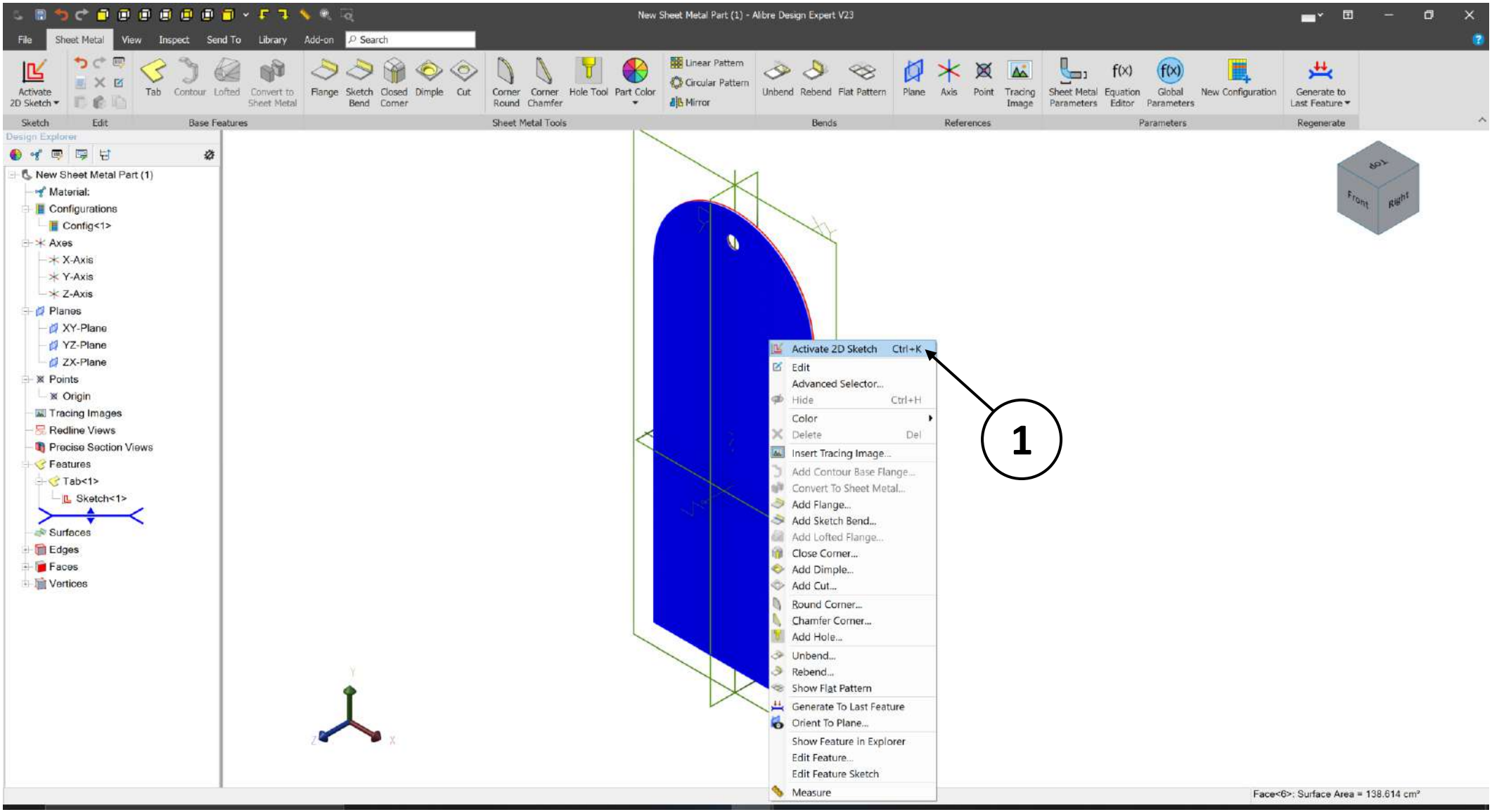
1. Click on **Dimension**.
2. Click on the **Center Point** of the circular sketch you just created.
3. Click on the **node** point referenced by Callout “3” in the image above.
4. Enter a dimension of **2.54cm**, hit **Enter**, and click **Deactivate Sketch**.



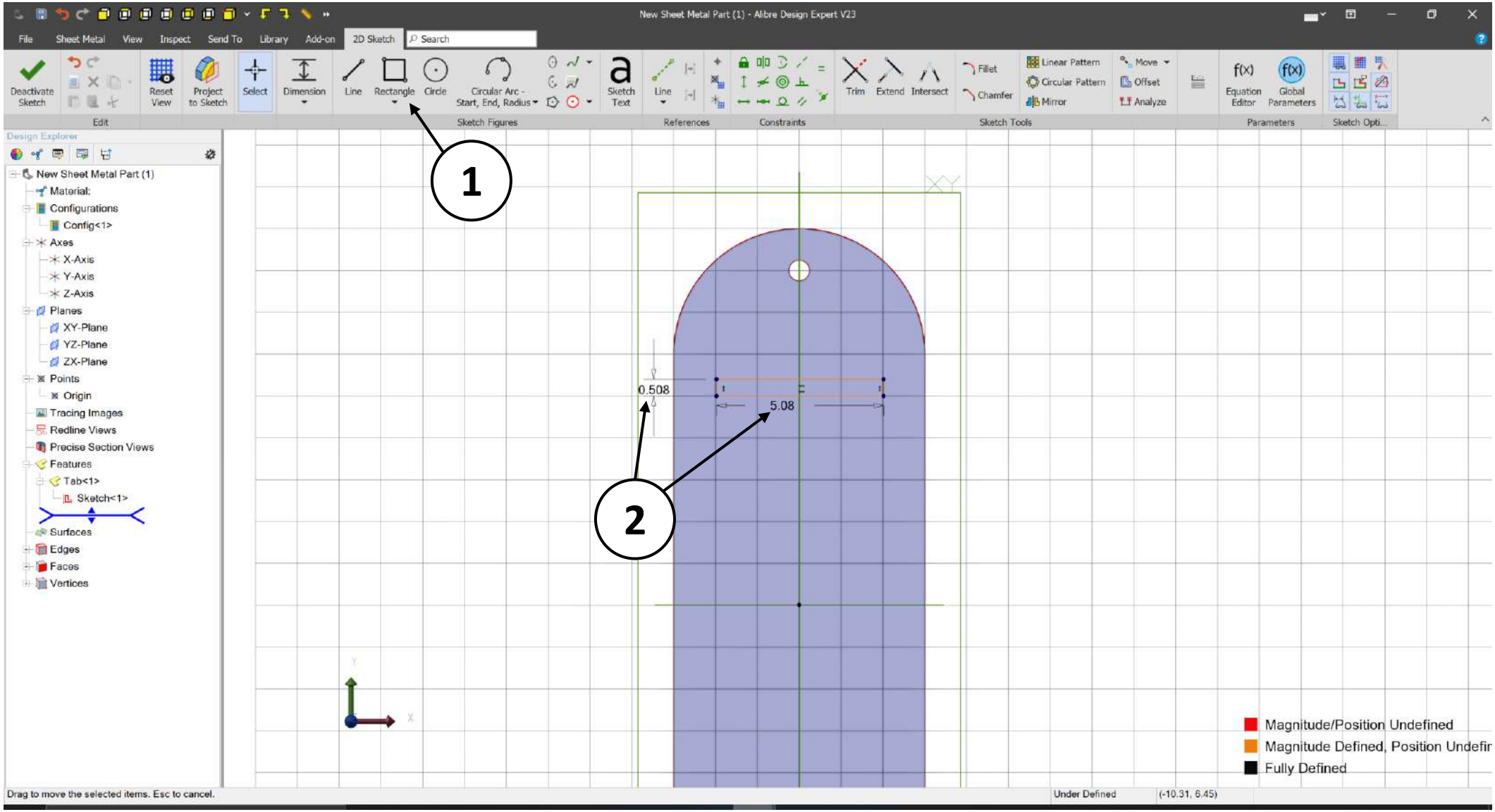
1. Click on the **Tab** in **Base Features** section in the ribbon.
2. Confirm that **Sketch<1>** appears in the **Tab;** entry box.
3. Click **OK.**



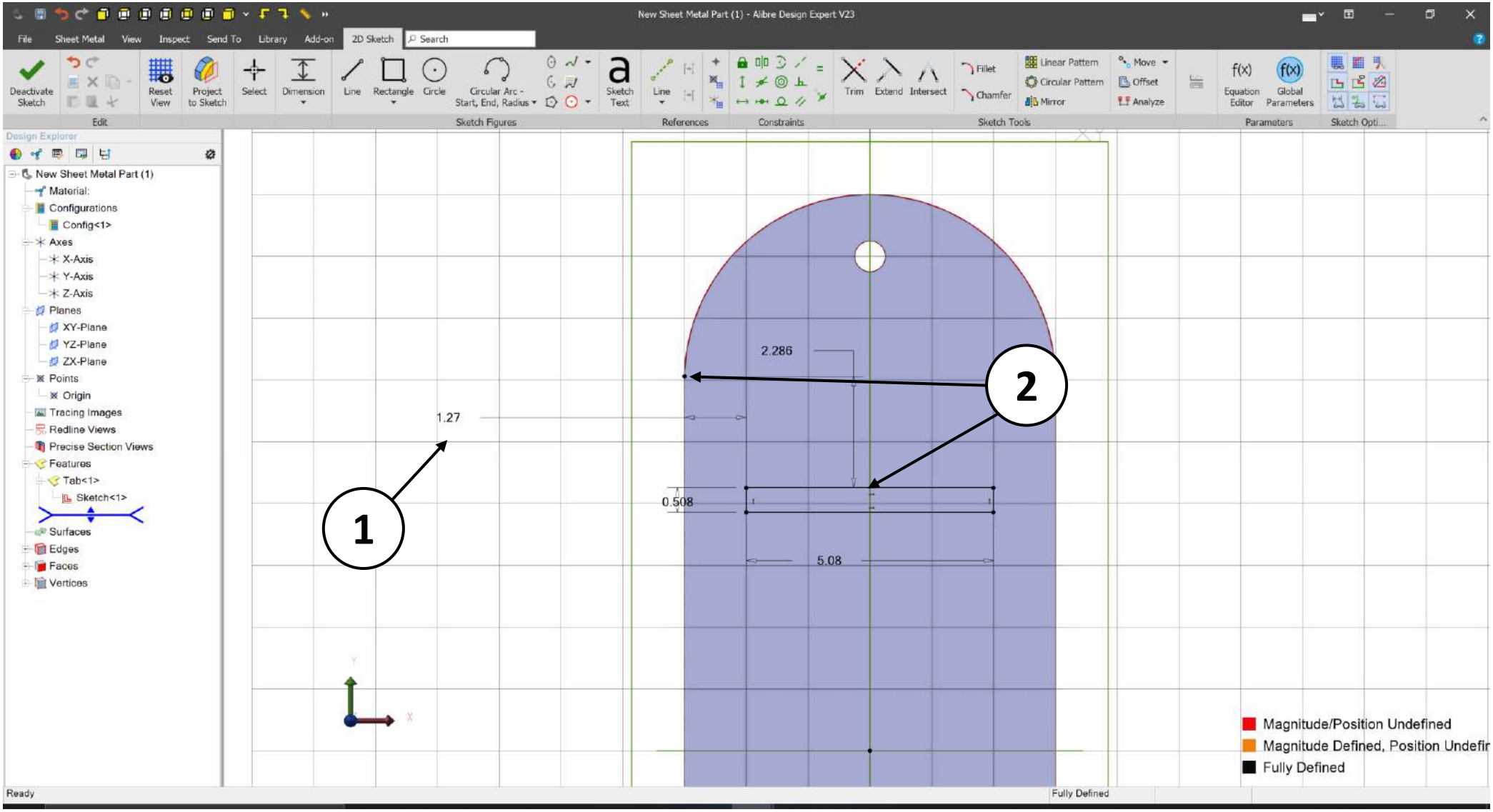
1. Confirm similar results for the **Tab** operation.



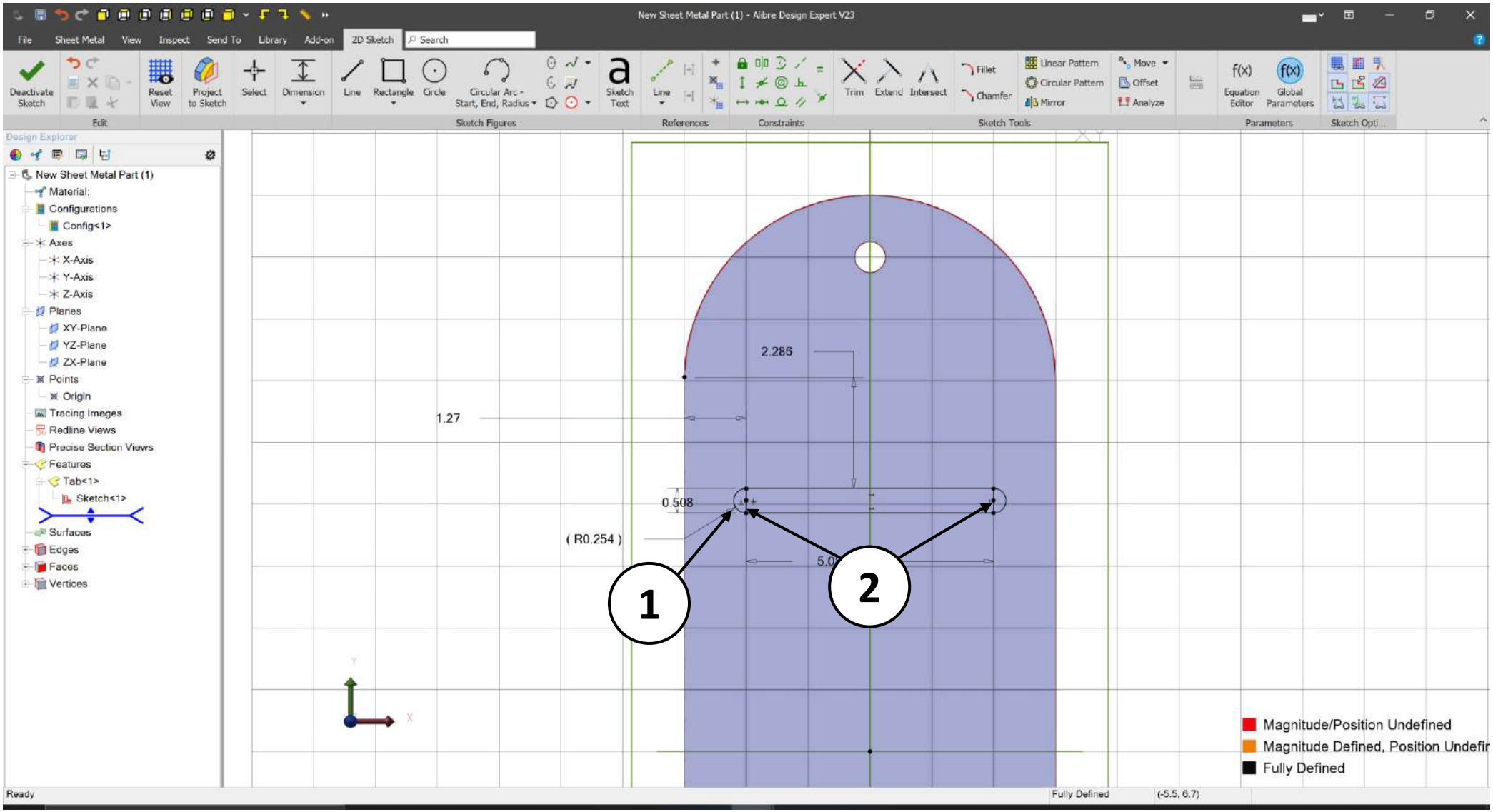
1. Right-click on the Front Face of the Tab and click **Activate 2D Sketch**. Alternatively, you can click on the front face and press **Ctrl+K.**



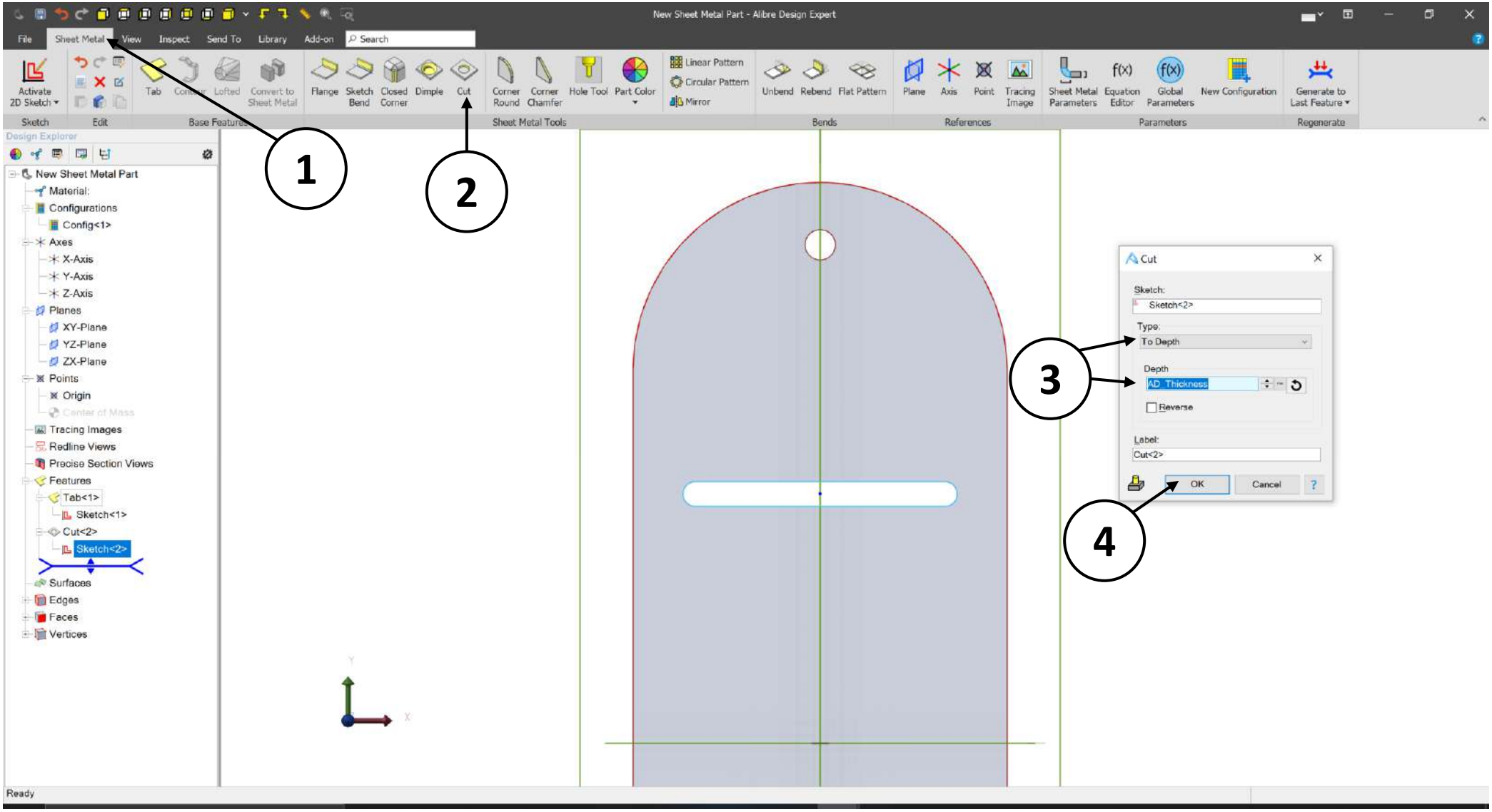
1. Use the **Rectangle** sketch tool to create a rectangular sketch similar to what is shown above.
2. Use the **Dimension** tool and apply a width of **0.508cm** and a length of **5.08cm** to the rectangular sketch.



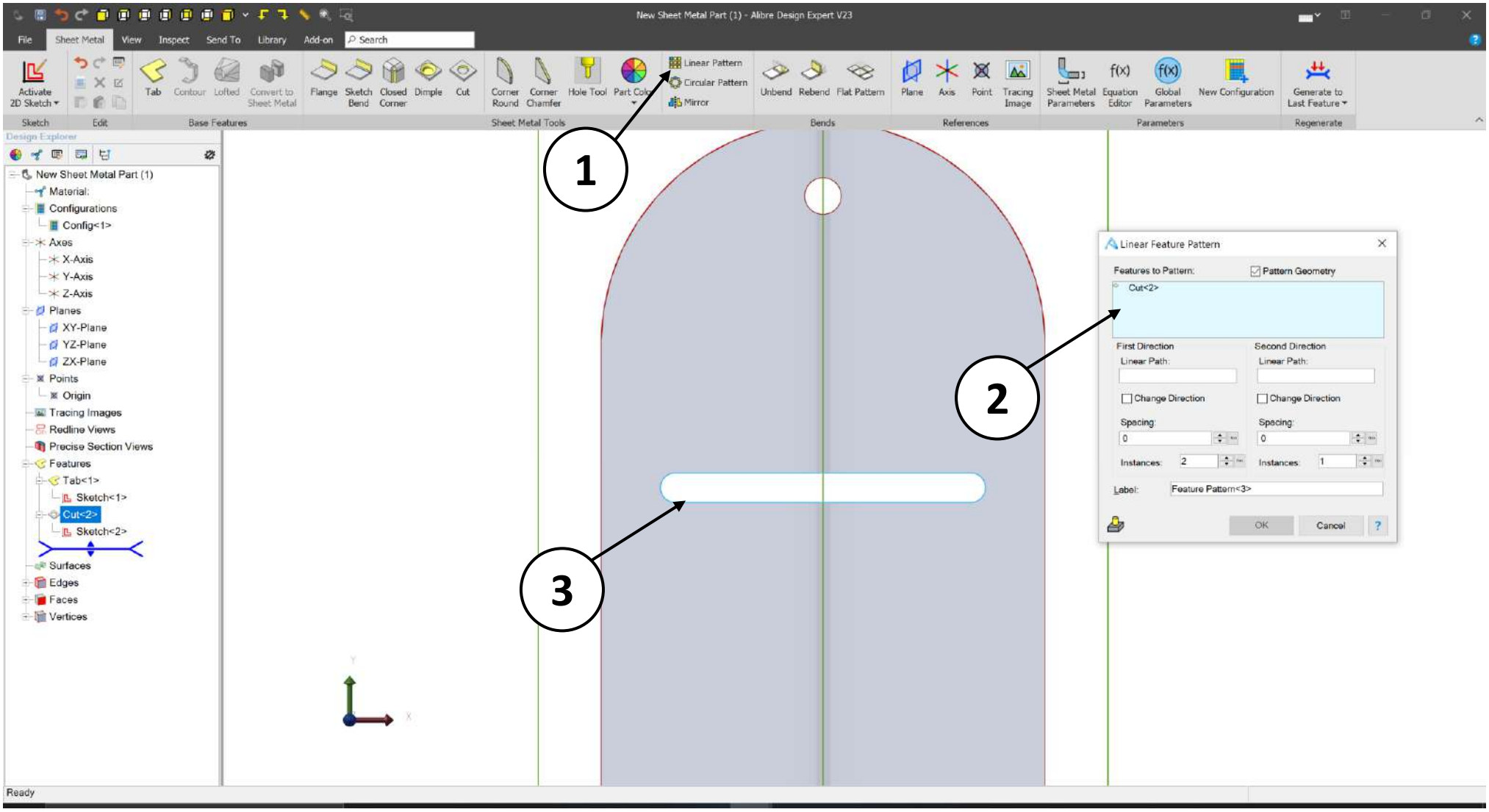
1. Zoom in if needed (not shown).
2. Use the **Dimension** tool to center the rectangular sketch. Use a value of **1.27cm** for the distance from the left edge of the tab. You will see a reference line appear.
3. Use the **Dimension** tool and apply a **2.286cm** distance from the top horizontal line of the rectangular sketch to the top node point of the reference line.



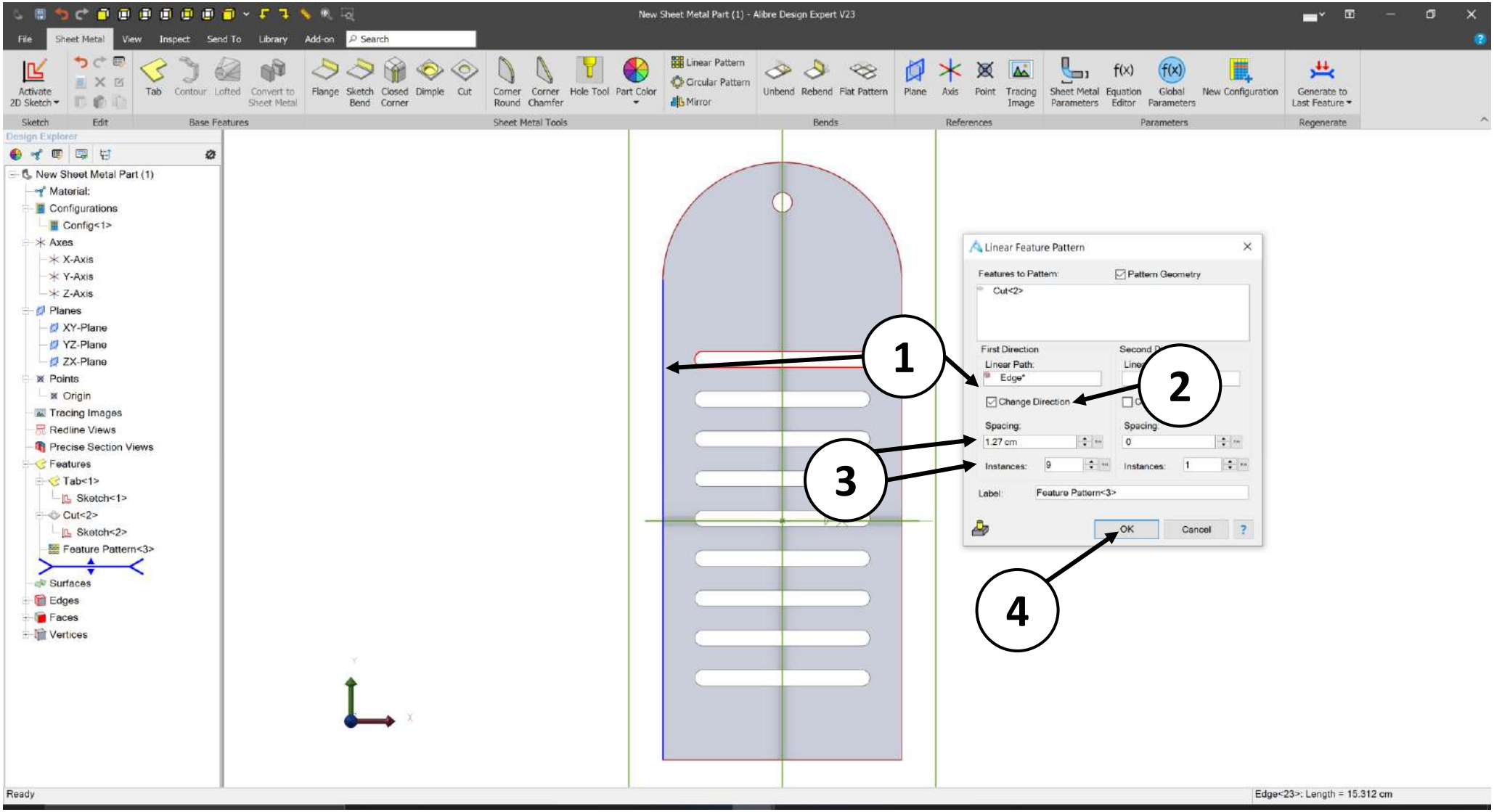
1. Use the Circular **Arc** – Start, End, Radius sketch tool to create two arcs at each end of the rectangle with a radius of **.254cm**
2. Use the **Trim** tool to remove the vertical sketch lines of the rectangle, creating an obround shape as shown above.



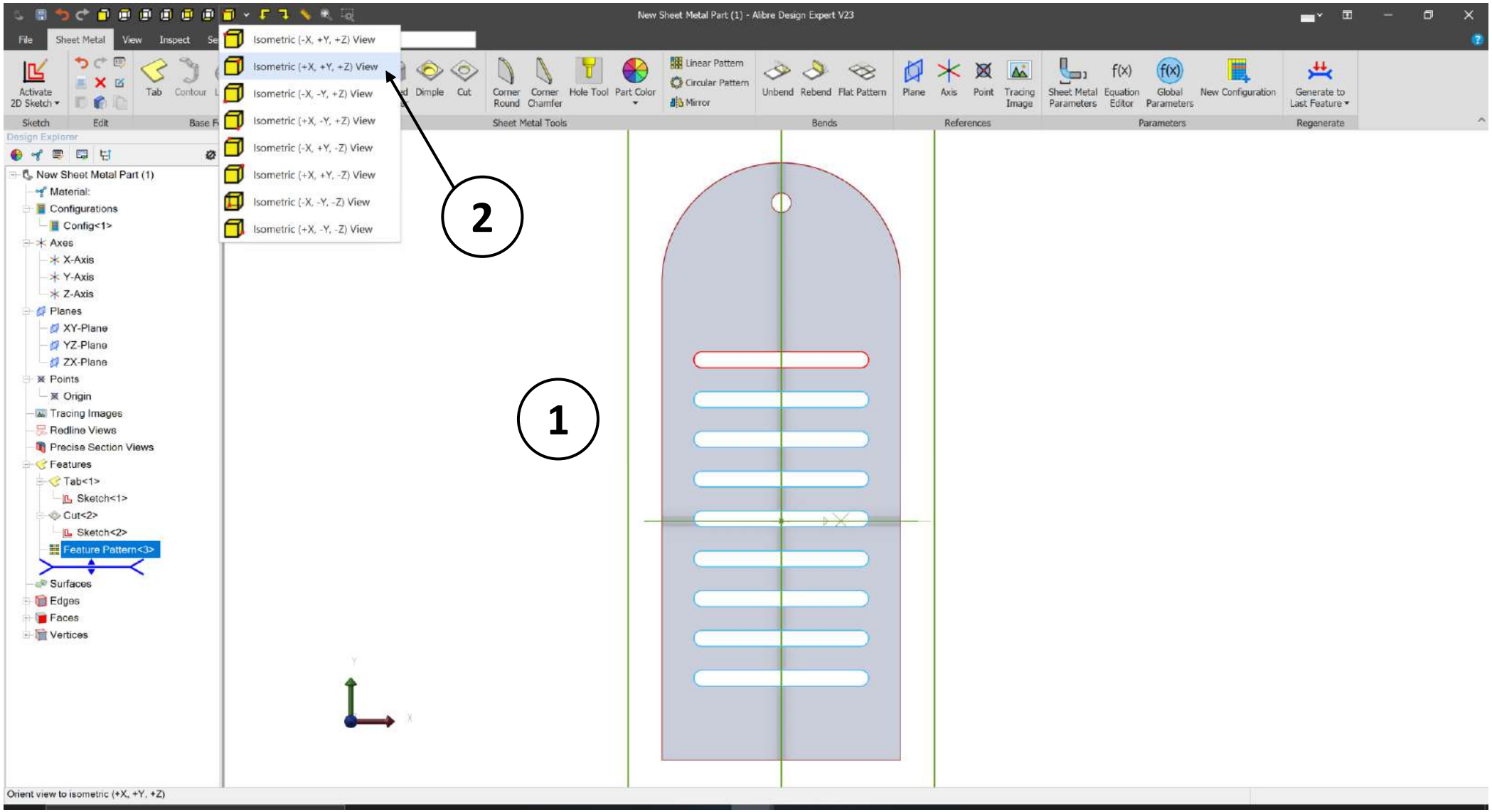
1. Click on the **Sheet Metal** tab.
2. Click on the **Cut** tool in the **Sheet Metal Tools** section of the Ribbon,
3. In the dialog, select **Sketch<2>,** make sure the cut type is **To Depth** and the depth is set to **AD_Thickness.**
4. Click **Ok.**



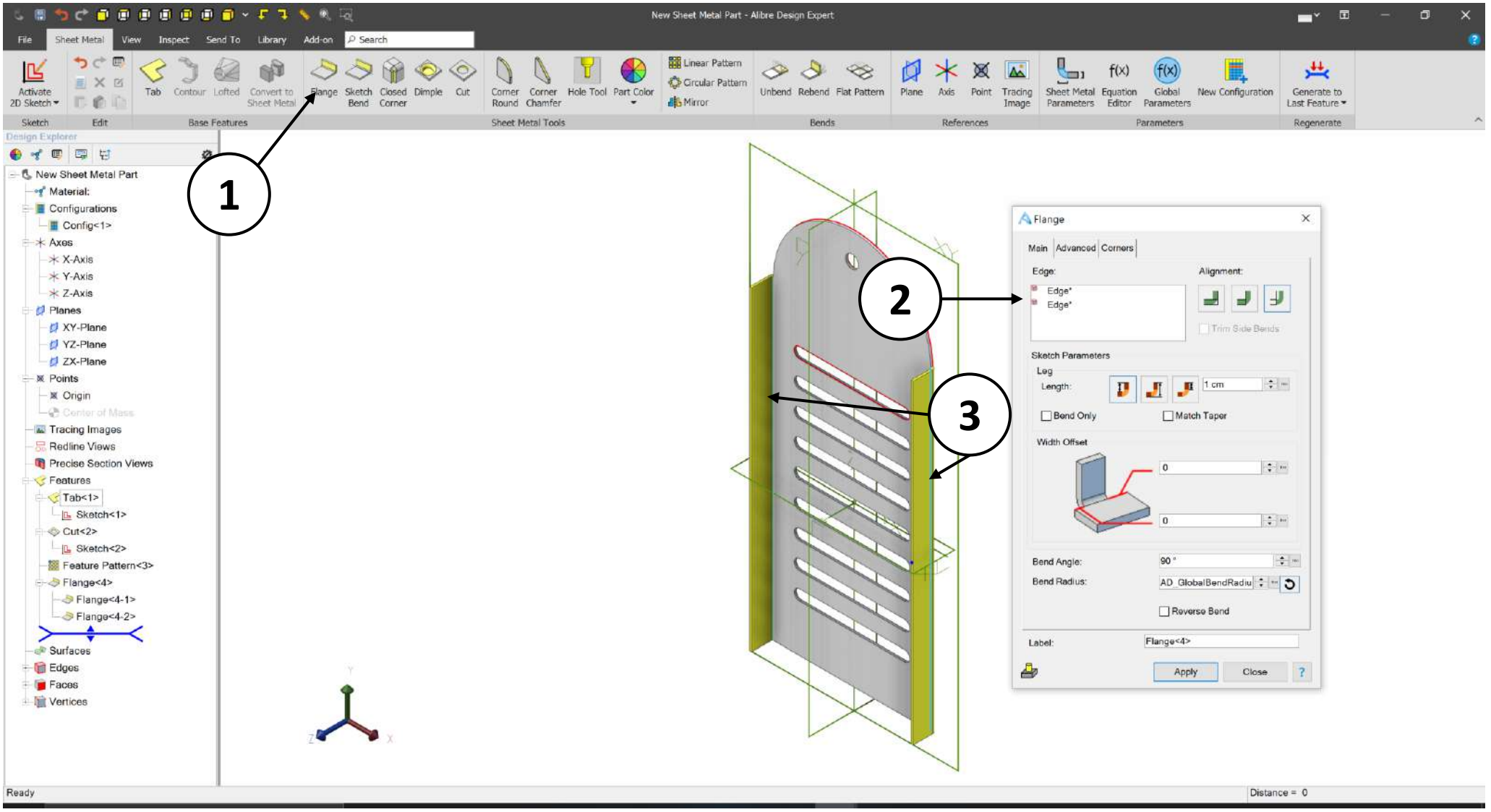
1. Click the **Linear Feature Pattern** tool.
2. In the **Linear Feature Pattern** dialog box, click on the **Features to pattern** field.
3. Select **Cut<2>.**



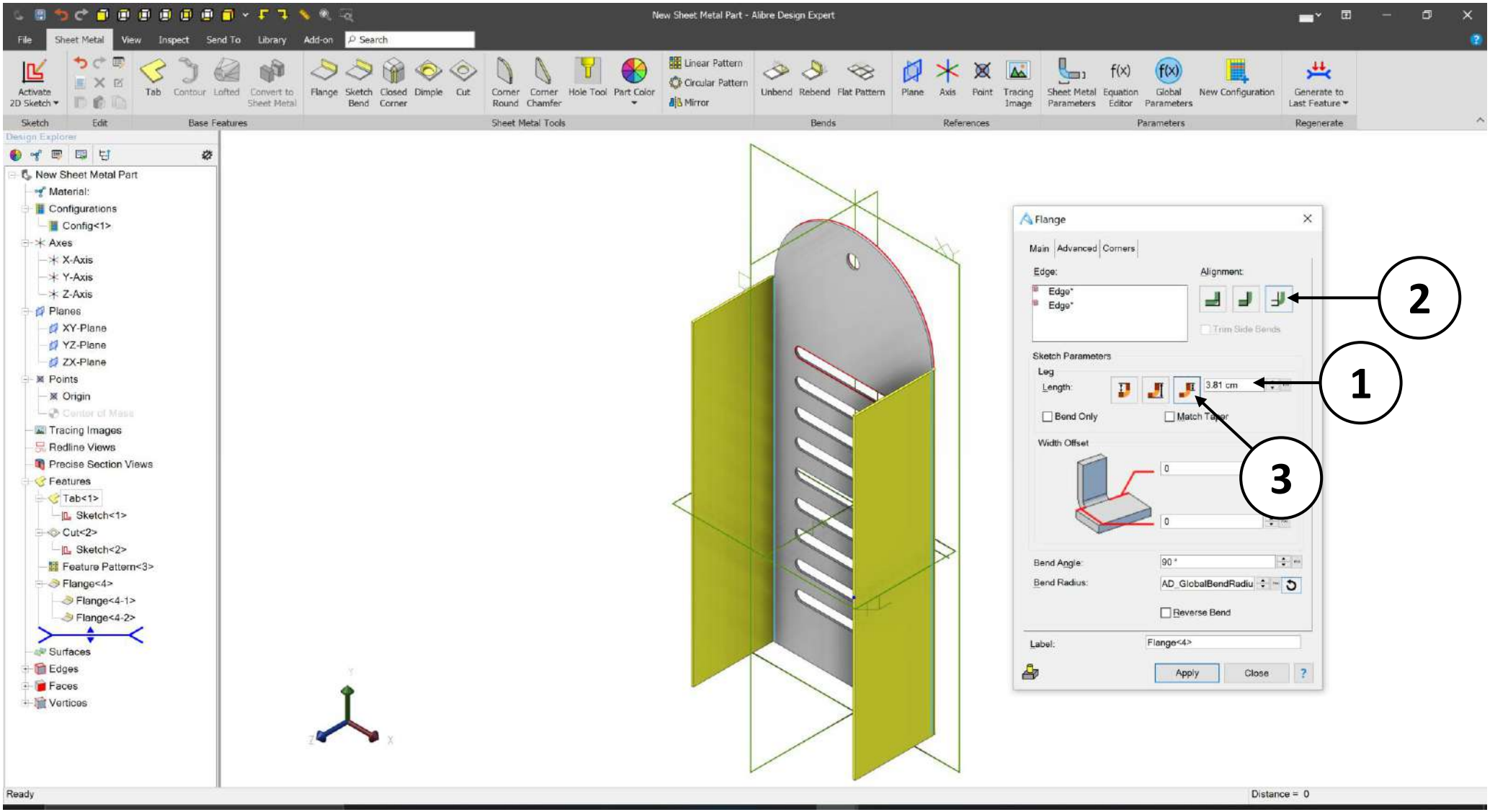
1. In the **Linear Feature Patterm** dialog box, click on the box named **Linear Path** and select the highlighted edge as the **Pattern Direction**.
2. Click the **Change Direction** checkbox.
3. Set the **Spacing** to **1.27 cm** and the **Instances** to **9**.
4. Click **Ok**.



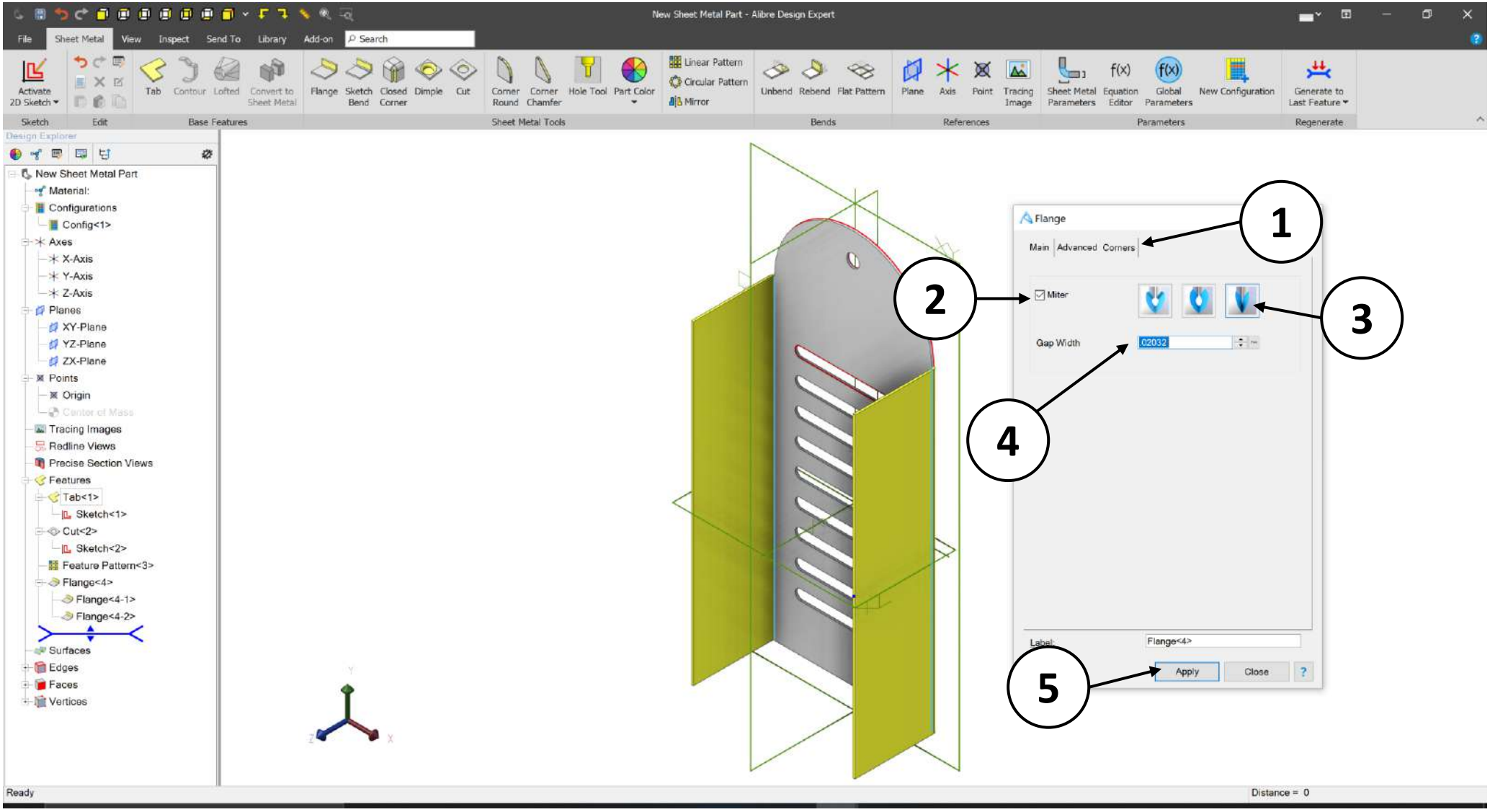
1. Confirm the results of the **Feature Pattern**.
2. From the **Quick Access** bar, click the dropdown for **Isometric Views** and select **Isometric (+X,+Y,+Z) View**.



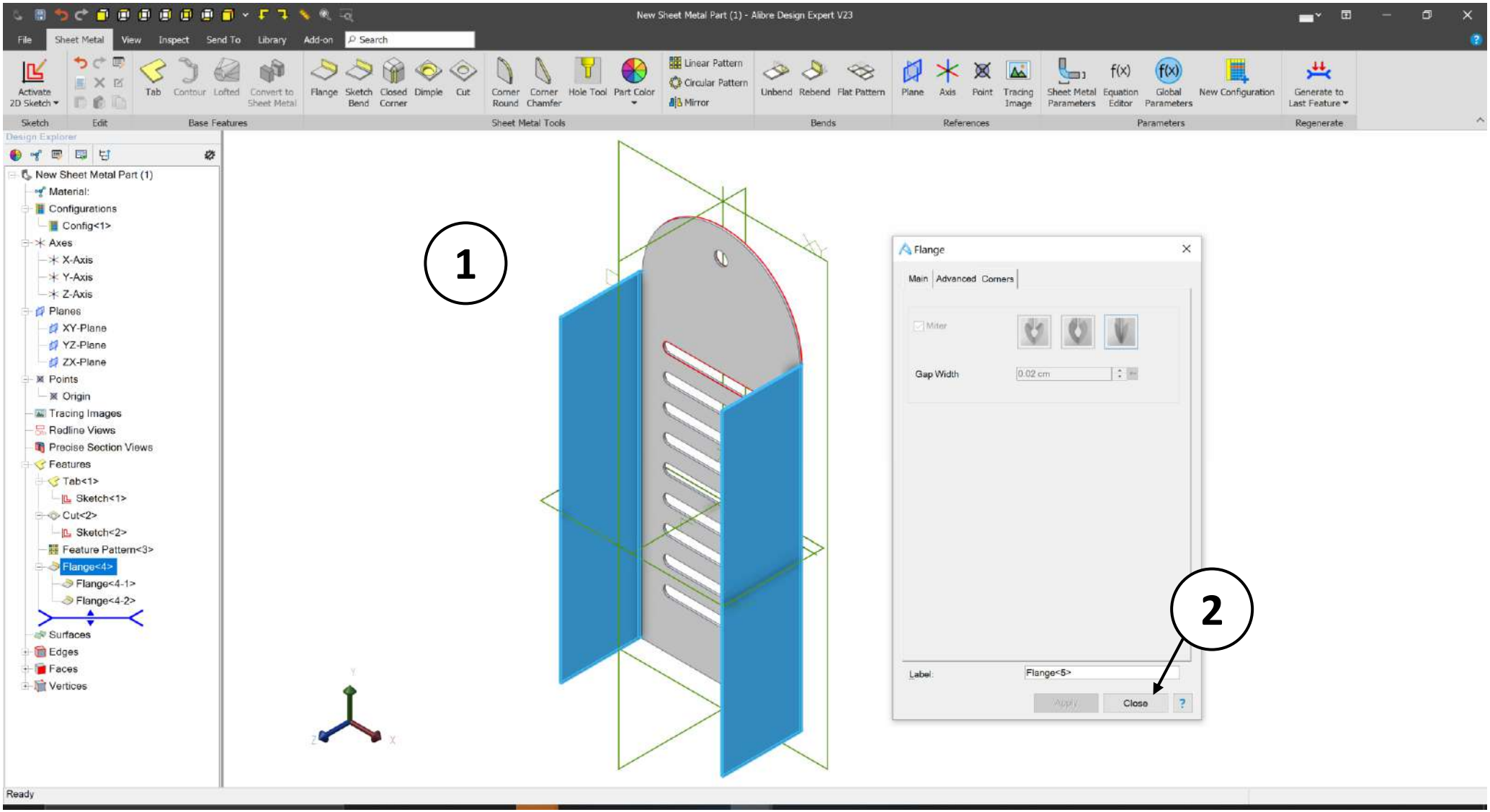
1. Click on the **Flange** in **Sheet Metal Tools** section in the ribbon.
2. Click the **Edge** box inside the Dialog.
3. Click on the edges shown to select them.



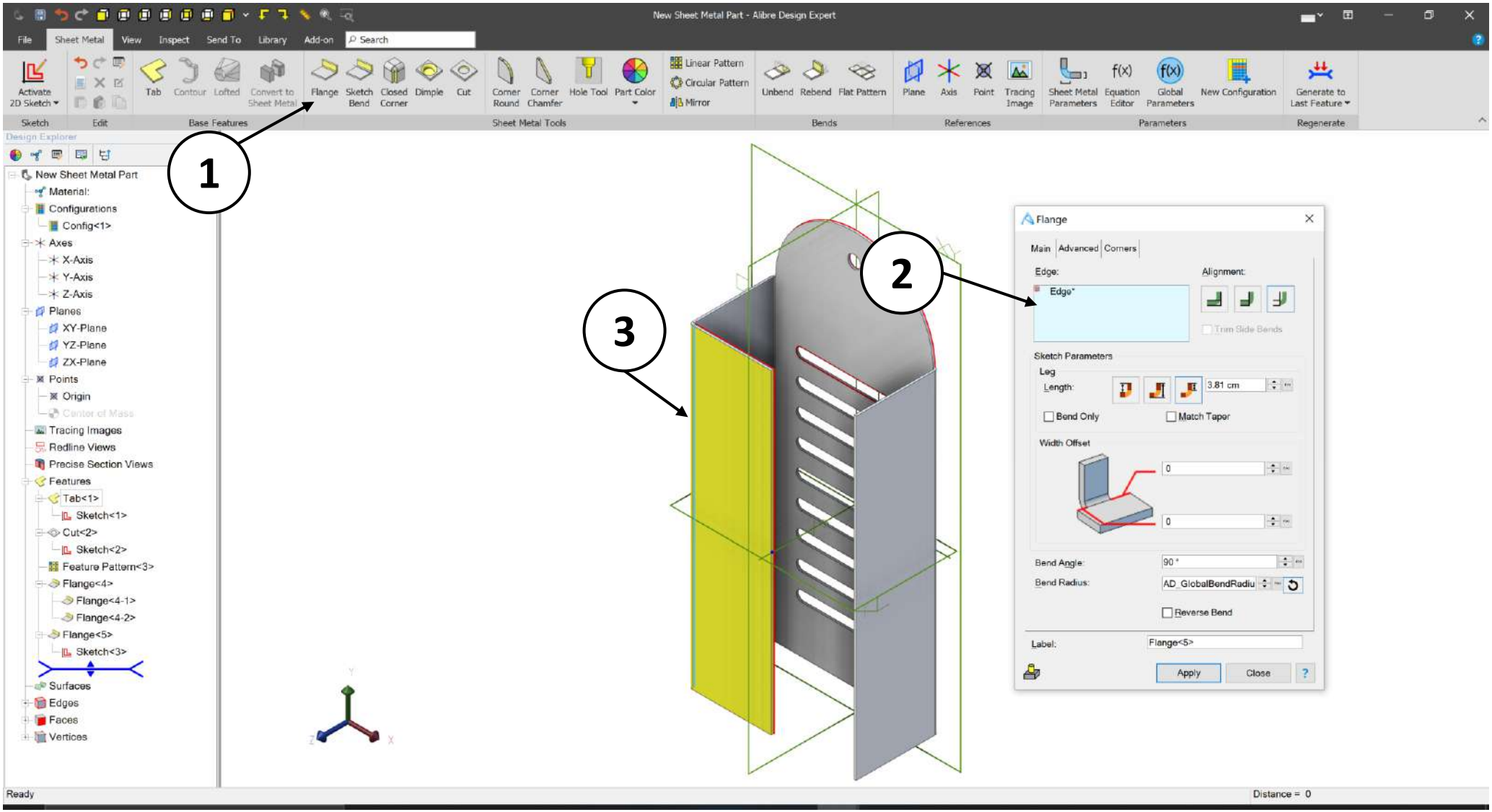
1. Set the length value for the **Flange** to **3.81 cm**.
2. Select the **Adjacent** option under **Alignment** in **Main** tab of **Flange** dialog box.
3. Select the **Tab** under the **Length** option in **Main** tab of **Flange** dialog box.



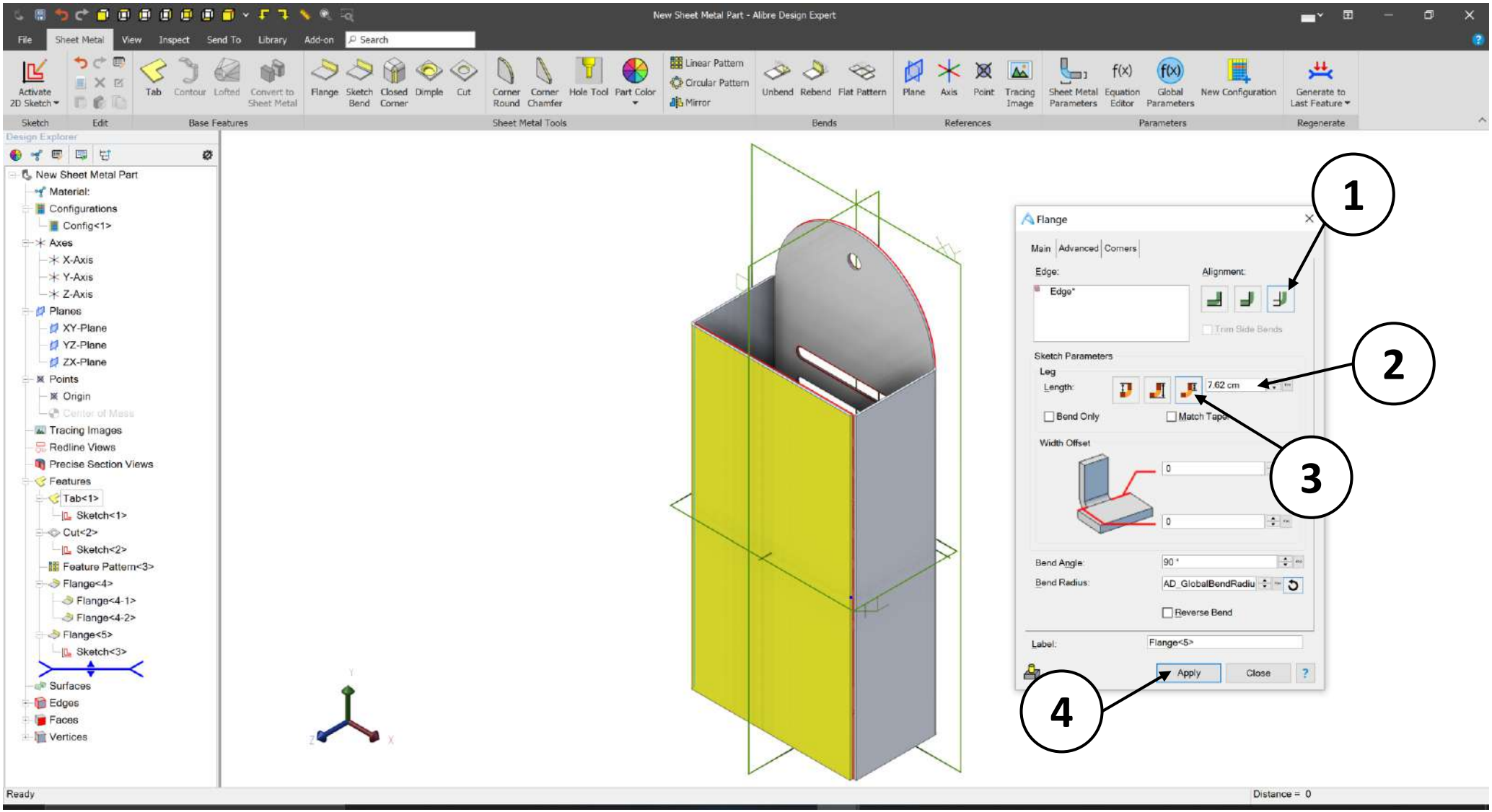
1. Click on the **Corners** tab in the **Flange** dialog box.
2. Click on the checkbox and Enable the **Miter**.
3. Select the **Corner Closed** under the **Miter** in **Corners** tab of **Flange** dialog box.
4. Input a value of **.02032** for the **Gap width**
5. Click **Apply.**



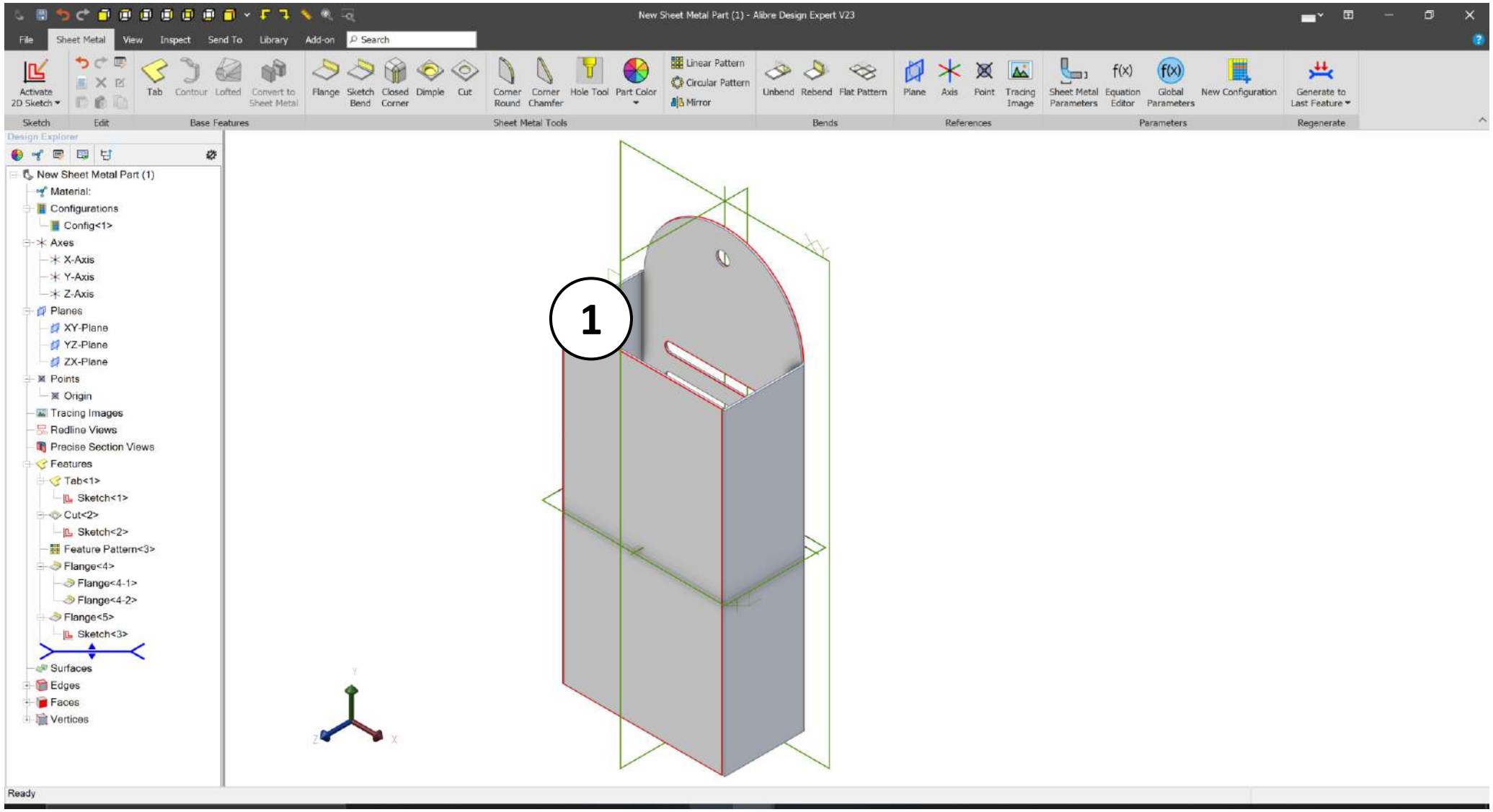
1. Confirm similar results for the **Flange** operation.
2. Click on the **Close**.



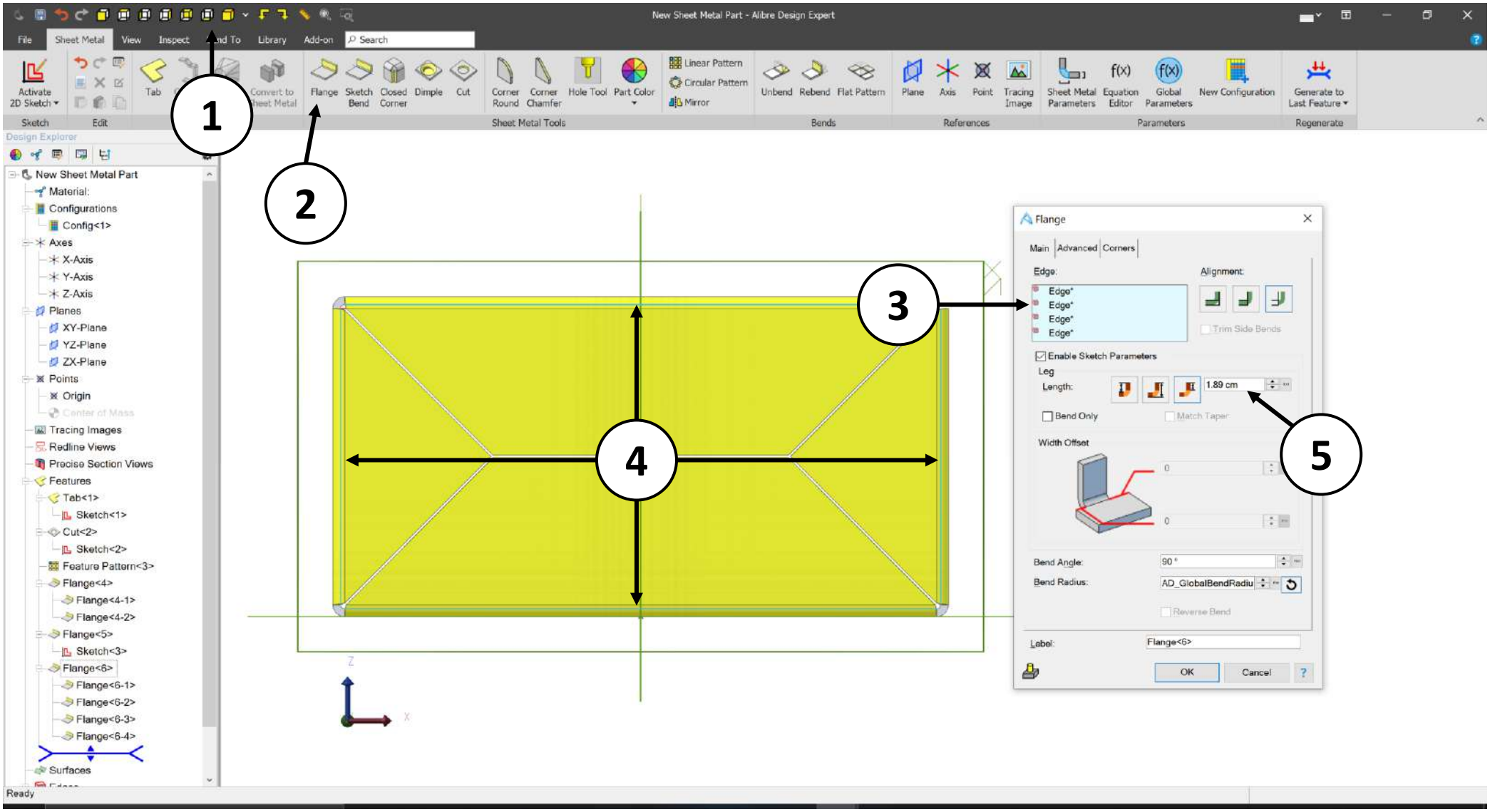
1. Click on **Flange** in **Sheet Metal Tools** section in the ribbon.
2. Click on the **Edge** box in the dialog.
3. Position the mouse cursor over the inside edge of the left flange, and then click to select it.



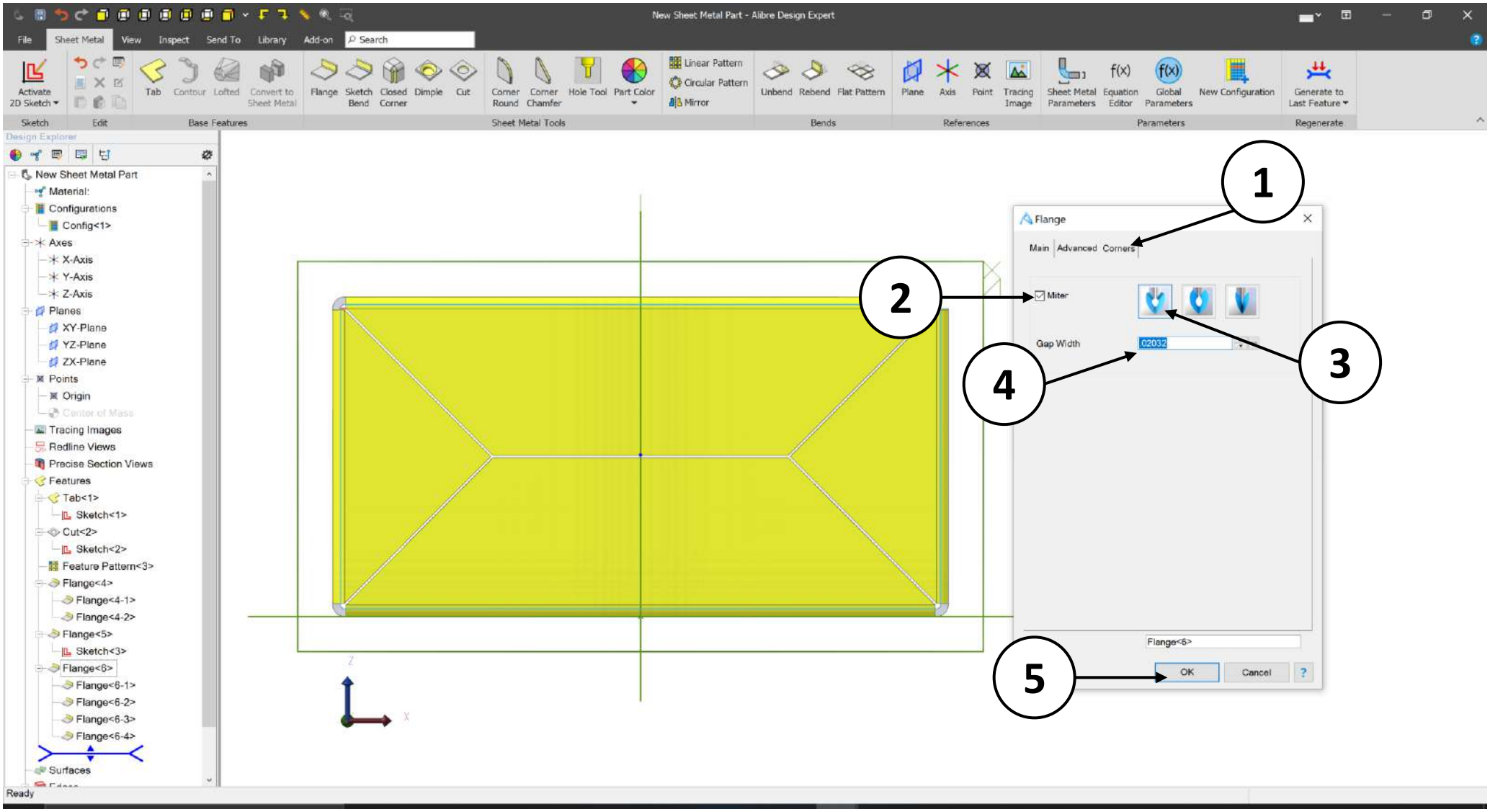
1. Select the **Inside** option under **Alignment** in **Main** tab of **Flange** dialog box.
2. Enter a value of **7.62 cm** for the leg length in the dialog.
3. Select the **Tab** under **Length** in the **Main** tab of **Flange** dialog box.
4. Click **Apply** then **Close.**



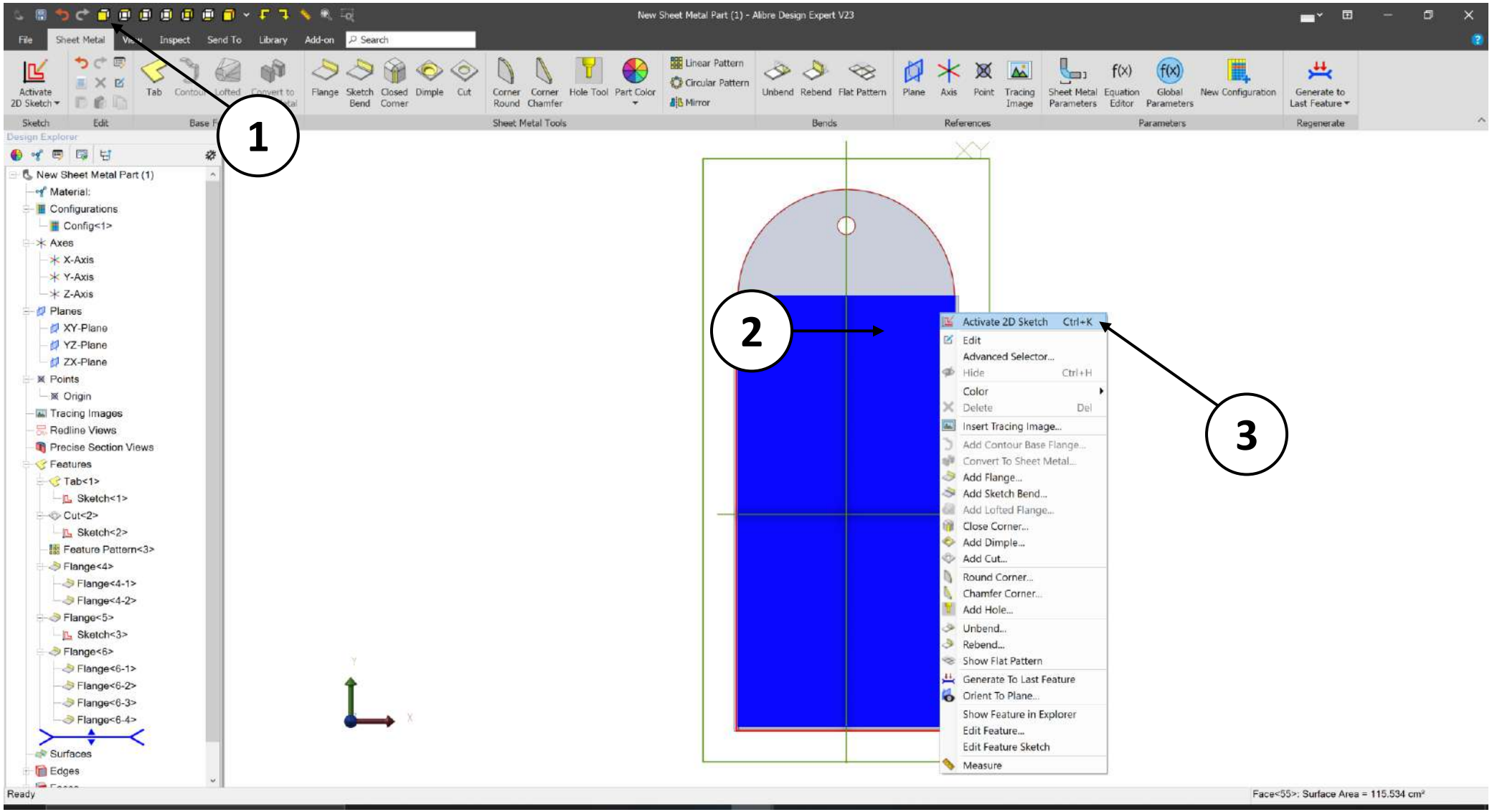
1. Confirm similar results for the **Flange** operation.



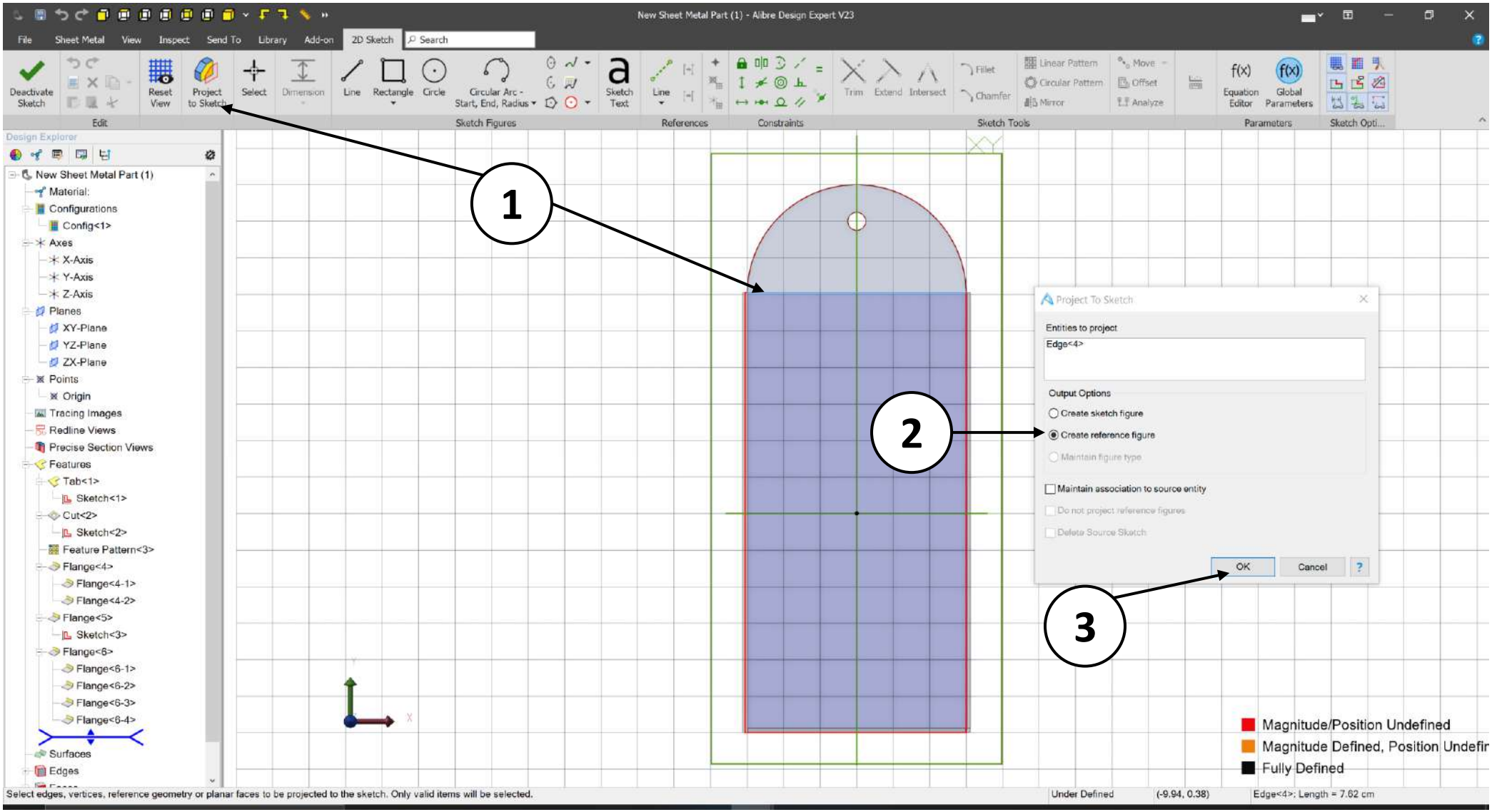
1. Select **Bottom View** icon from the **Quick Access** toolbar.
2. Click on **Flange** in **Sheet Metal Tools** section in the ribbon.
3. Click the **Edge** box in the dialog.
4. Click on all four inside edges of the bottom of the sheet metal part
5. Set the length value to **1.89 cm**



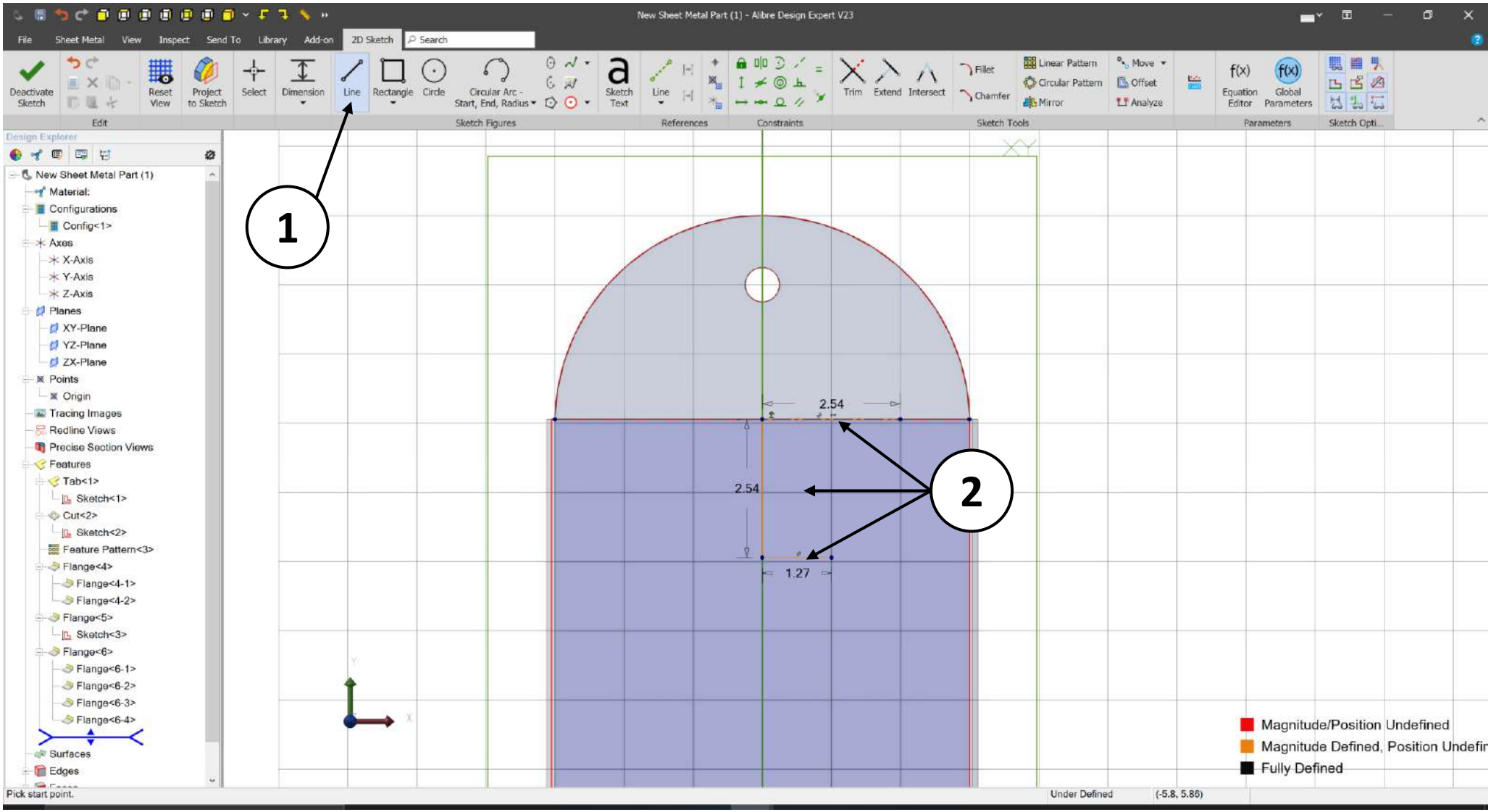
1. Click on the **Corners** tab in the **Flange** dialog box.
2. Click on the checkbox and Enable the **Miter**.
3. Select **Corner Relief** under **Miter** in the **Corners** tab of **Flange** dialog box.
4. Confirm the **Gap width** value in the **Flange** dialog box is **.02032.**
5. Click **OK.**



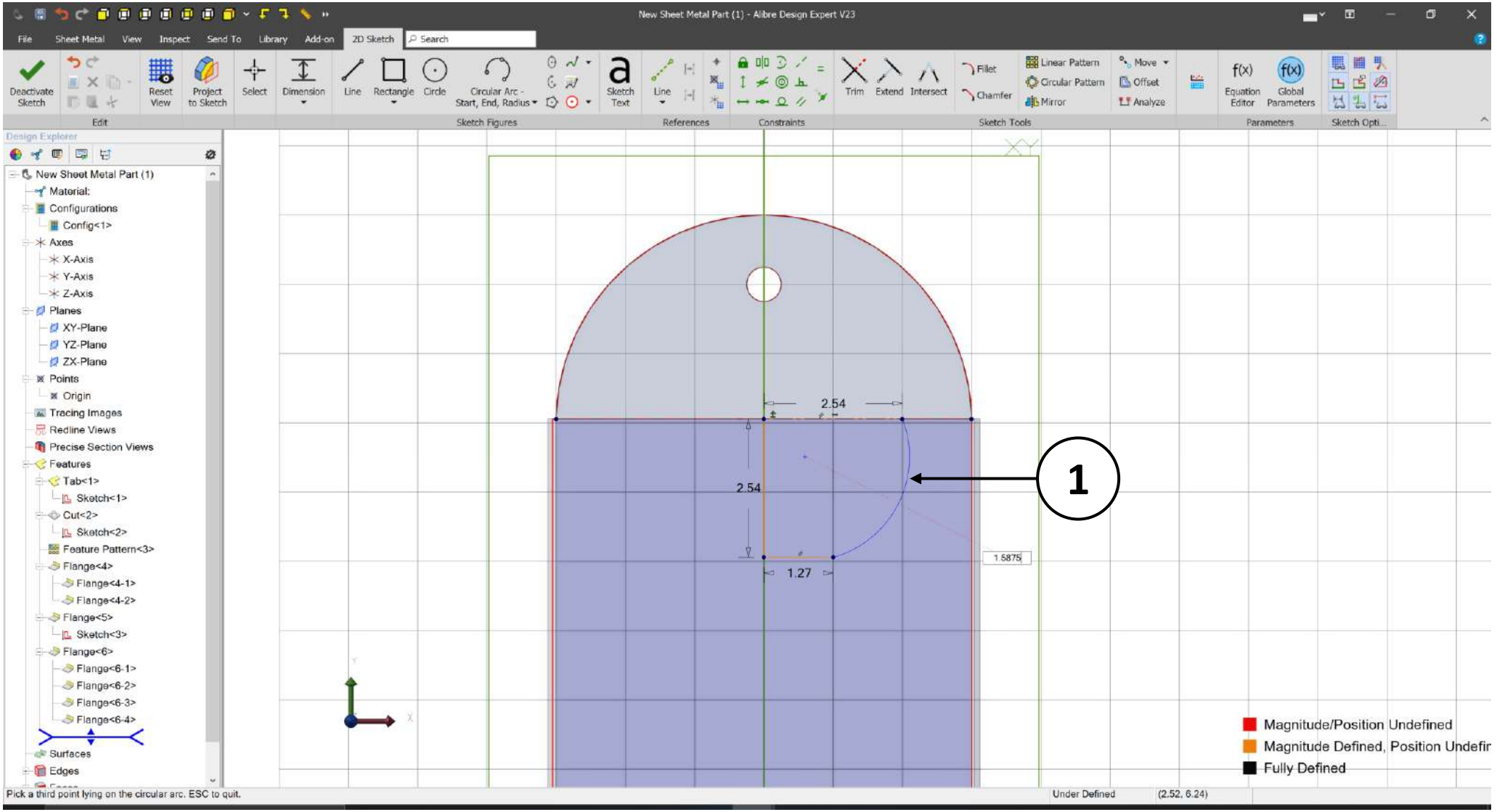
1. Click the **Front View** icon in the **Quick Access** toolbar.
2. Right-click on the front face of the sheet metal part.
3. Select the **Activate 2D Sketch** from the Menu or Press **Ctrl+K** on Keyboard.



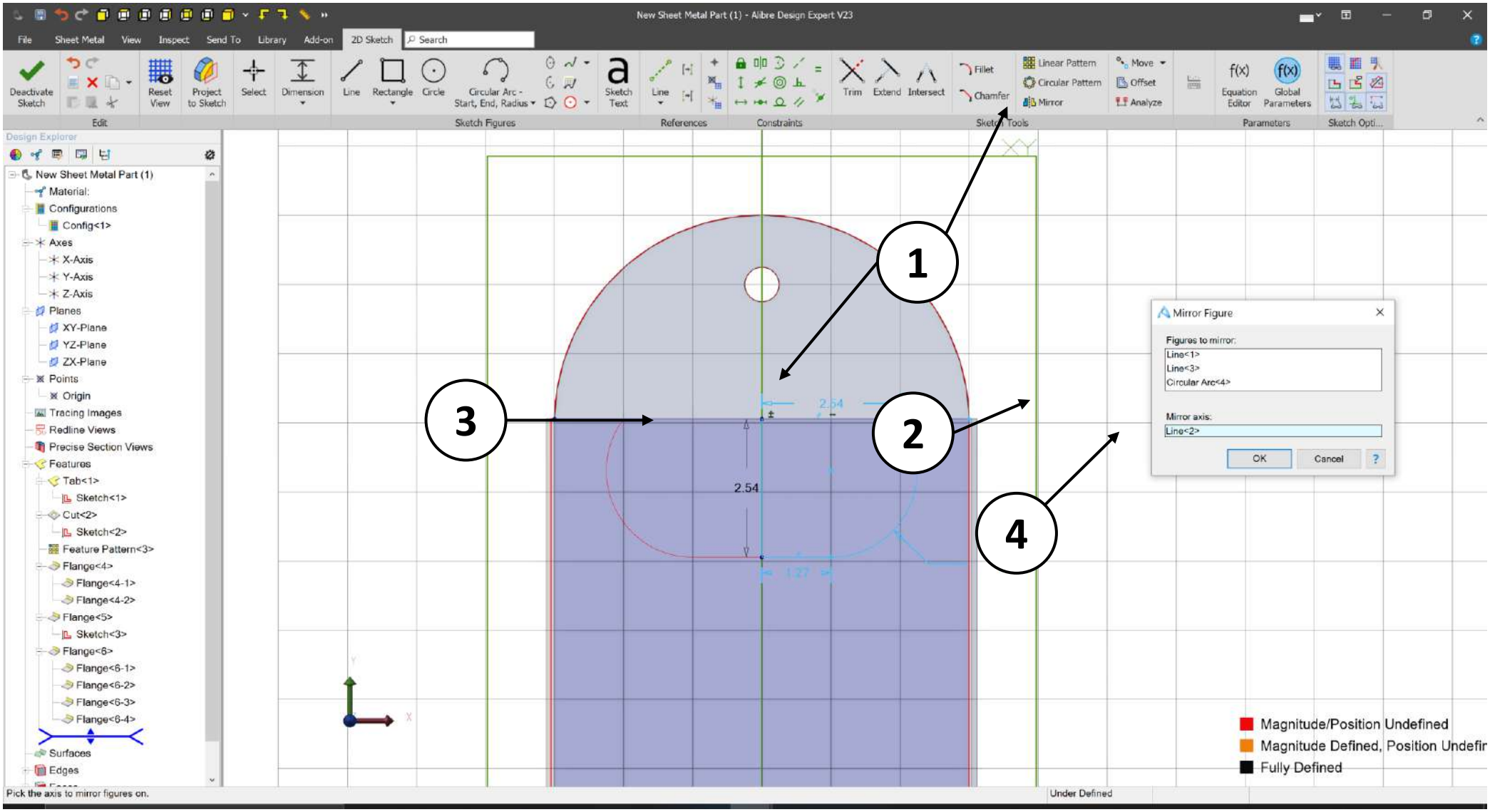
1. Click on the **Project to Sketch** tool and then click on the top edge of the front face of the model.
2. Choose the **Create reference figure** option in the dialog.
3. Click **OK.**



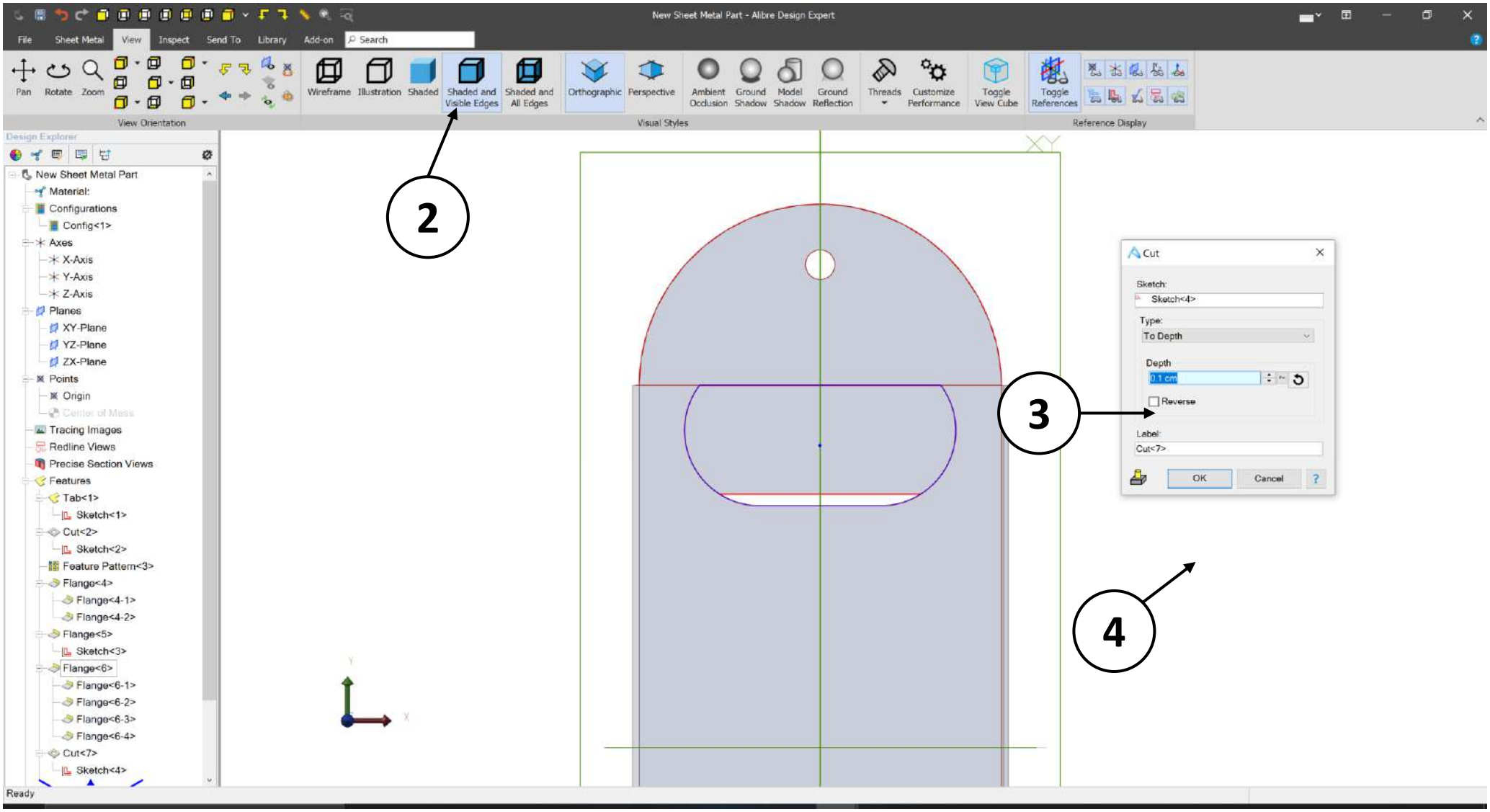
1. Click on the **Line** tool, then hover your cursor over the center point of the reference line.
2. Create the sketch lines as shown.



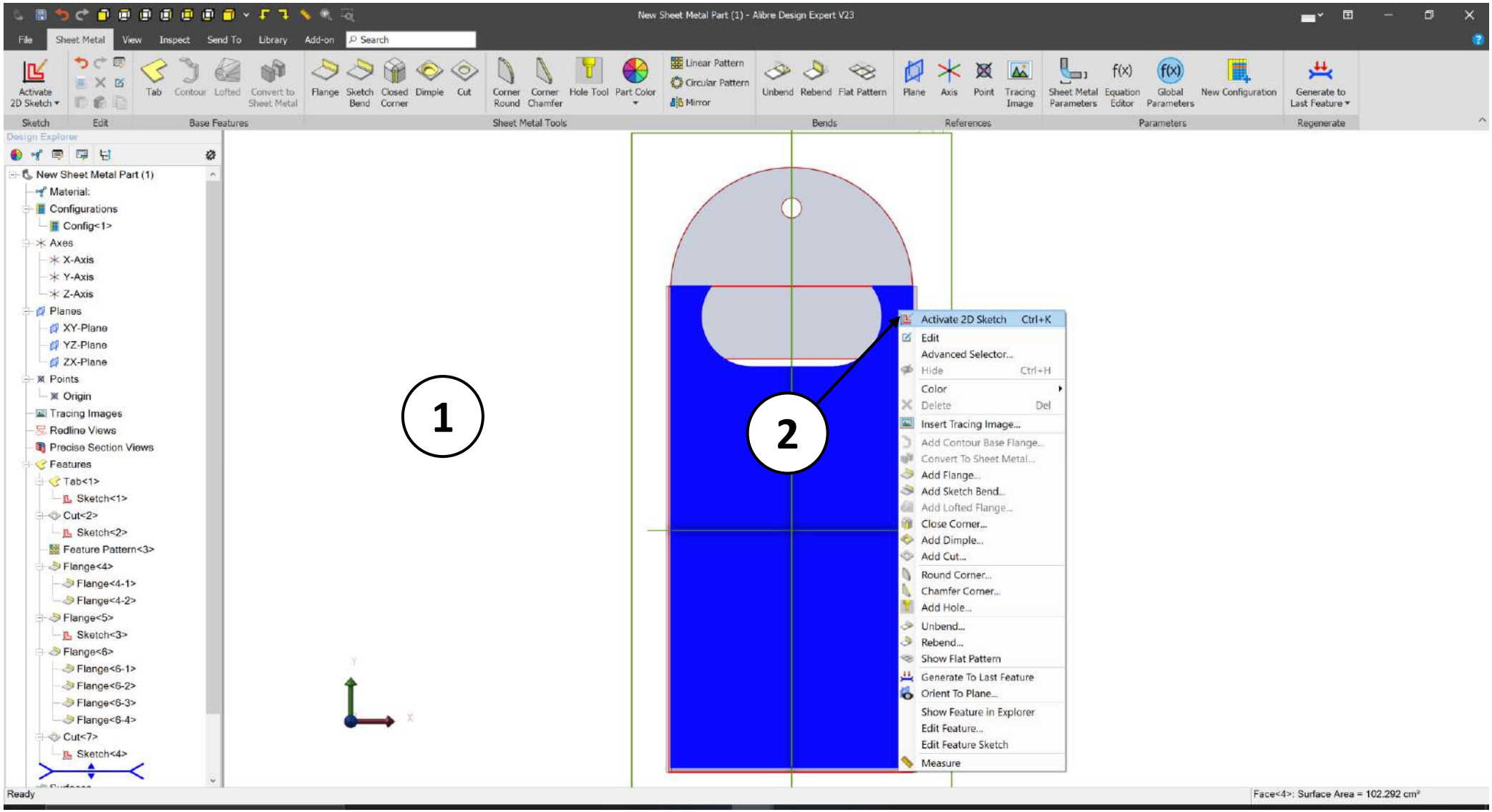
1. Use the **Circular Arc-Start, End, Radius** to create an arc with a radius of **1.5875 cm** to close the sketch.



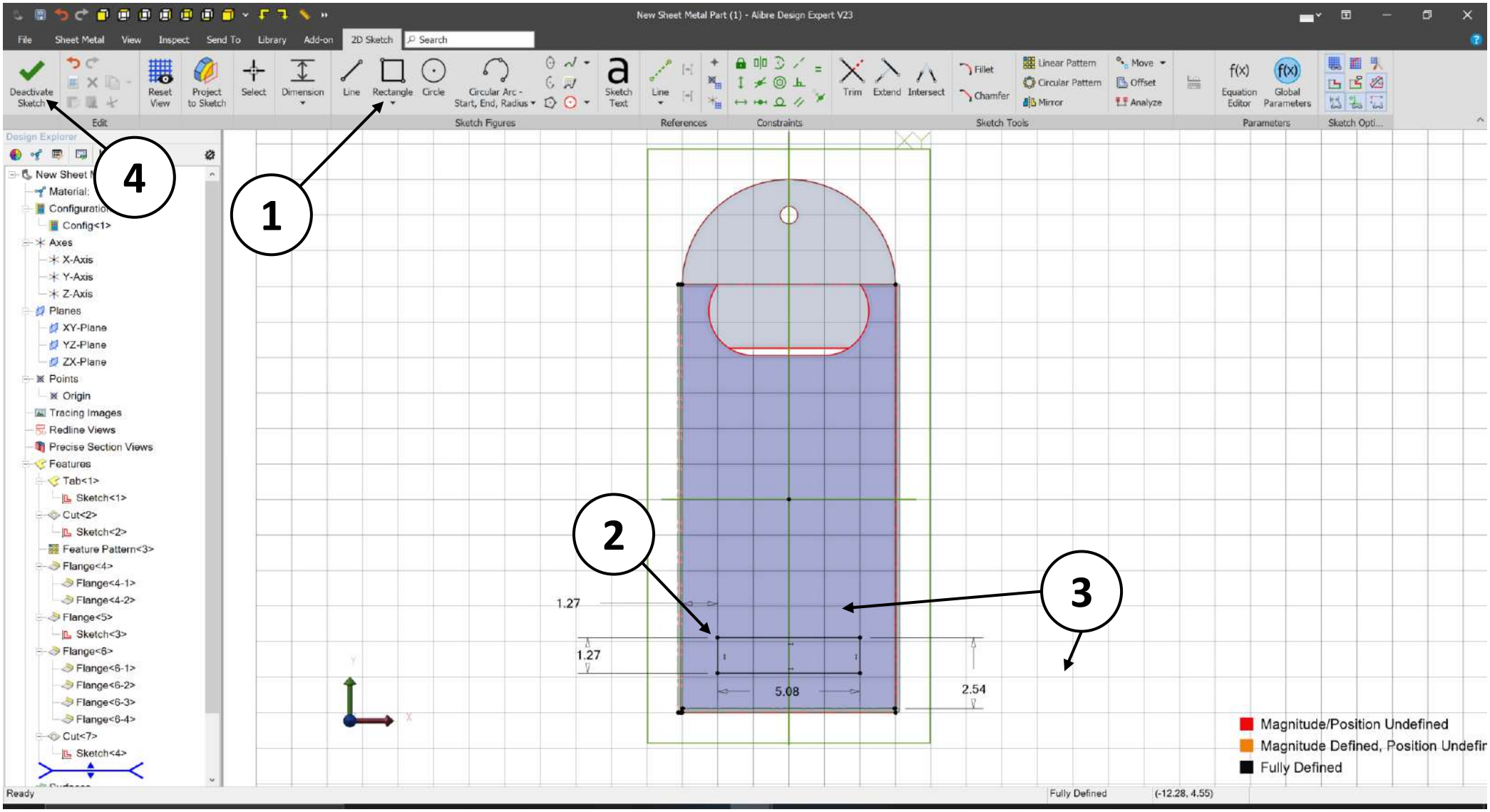
1. Click on the **Mirror** tool and select the sketch lines you just created except for the vertical line, and populate the **Figures to Mirror** box of the Mirror dialog.
2. Click on the **Mirror Axis** box of the mirror figure dialog.
3. Select the vertical line of the sketch for the **Mirror axis**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



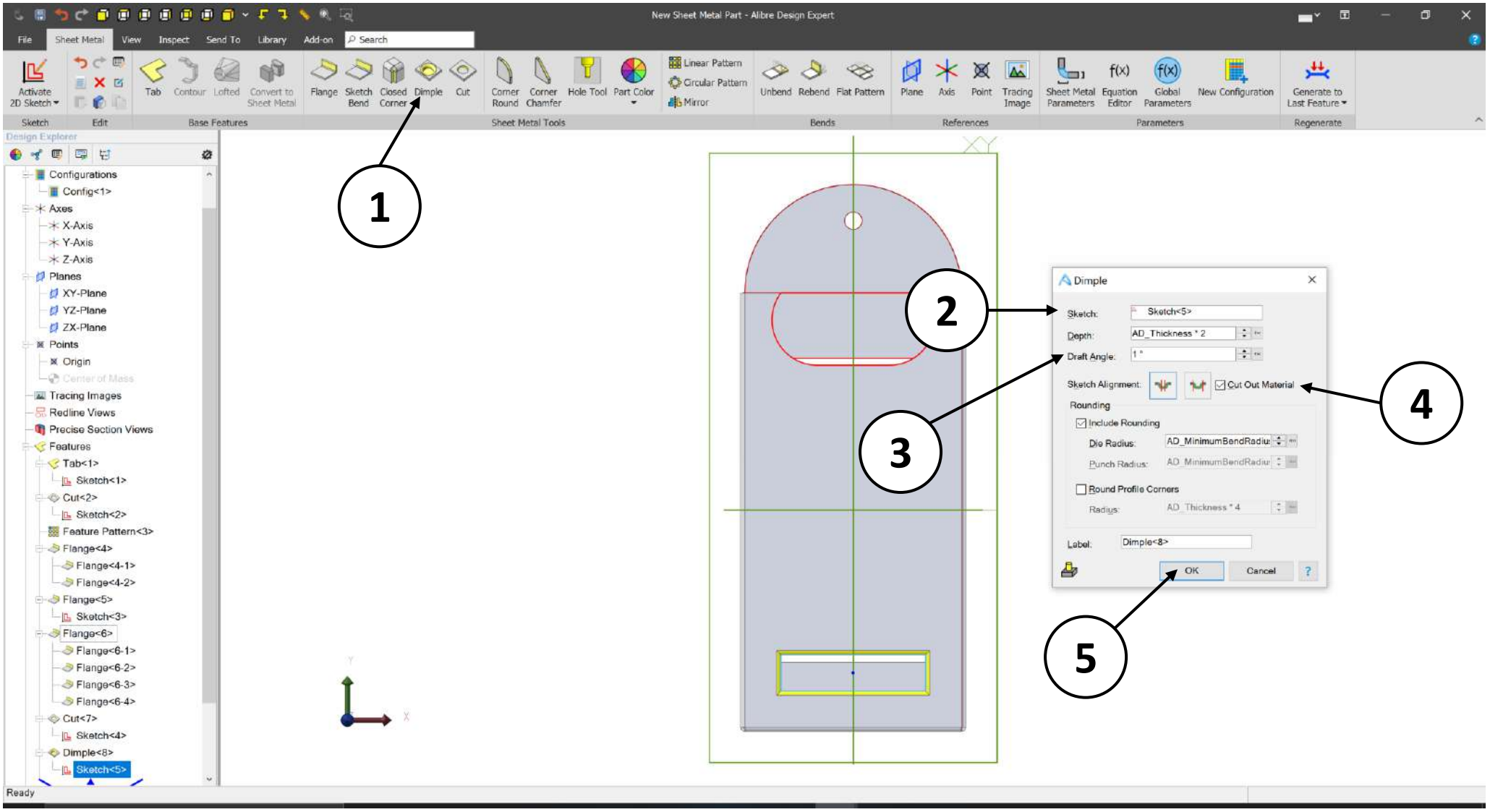
1. Use the **Trim** tool to remove the center line. Click **Deactivate Sketch** (Not shown).
2. Click on the **Cut** tool in the **Sheet Metal Tools** section on the Ribbon.
3. Select the **To Depth** option from the dropdown menu of **Type** in the **Cut** Dialog Box.
4. Click **OK.**



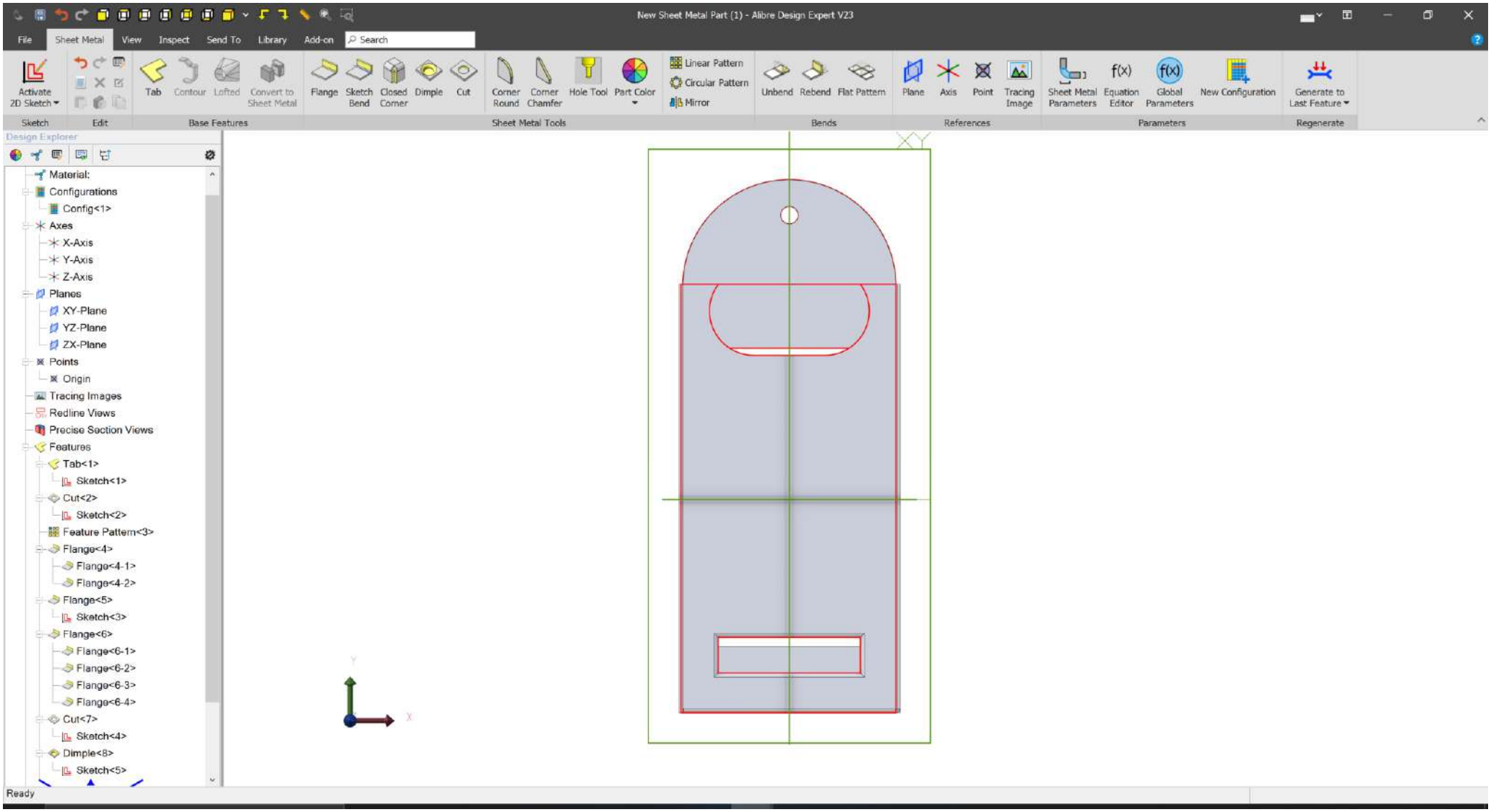
1. Confirm similar results for the **Cut** operation.
2. Right-click on the front face of the part and select the **Activate 2D Sketch** from the Menu or Press **Ctrl+K** on Keyboard.



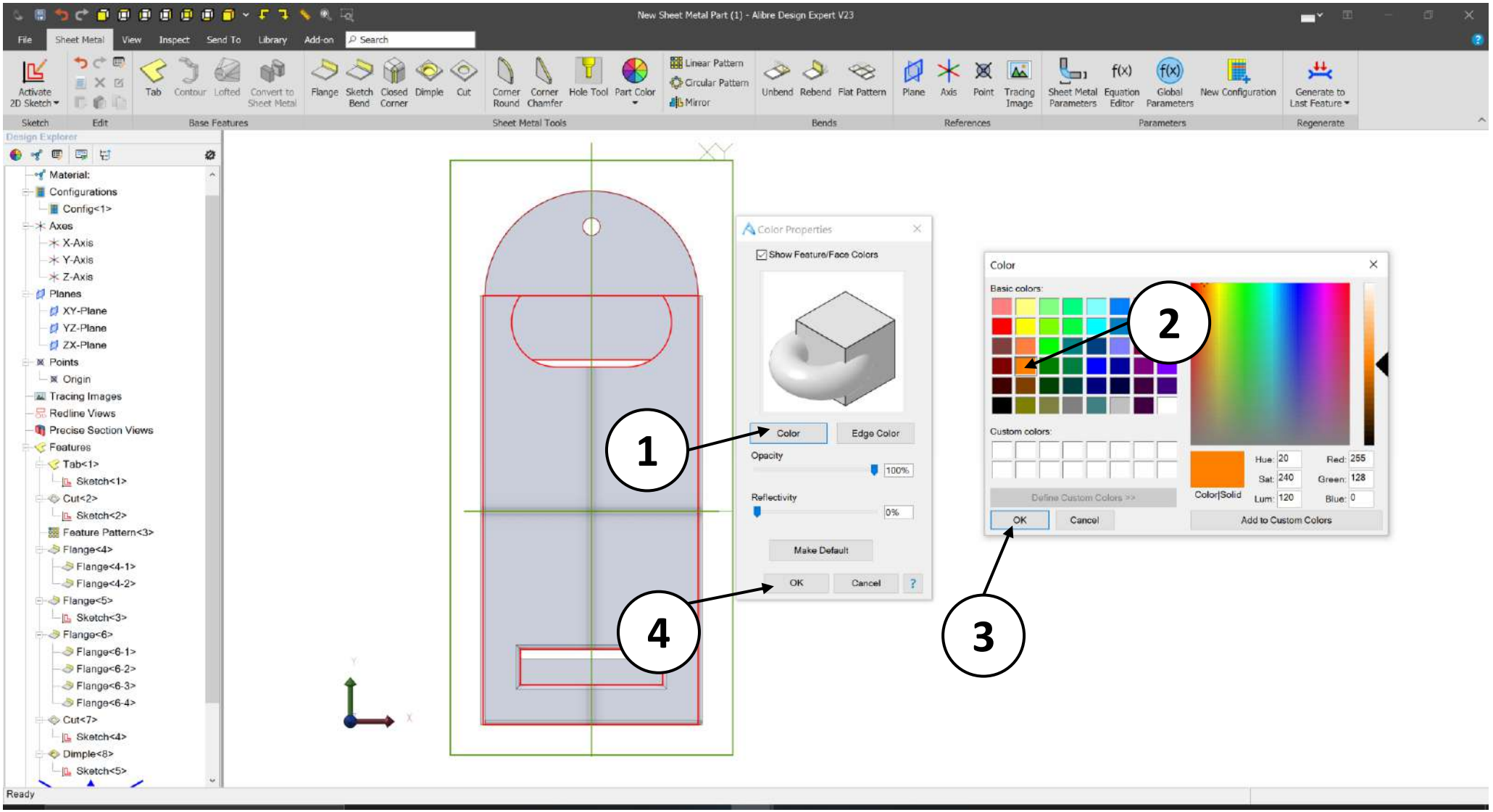
1. Select the **Rectangle** tool in the **Sketch Figures** section on the Ribbon.
2. Sketch a rectangle with a **Length** of **5.08cm** and a **Width 1.27cm.**
3. Use the **Dimension** tool to enter **2.54cm** between the top of the rectangle and the bottom edge of the part, and **1.27cm** between the left side of the rectangle and the left edge of the part.
4. Click **Deactivate Sketch.**



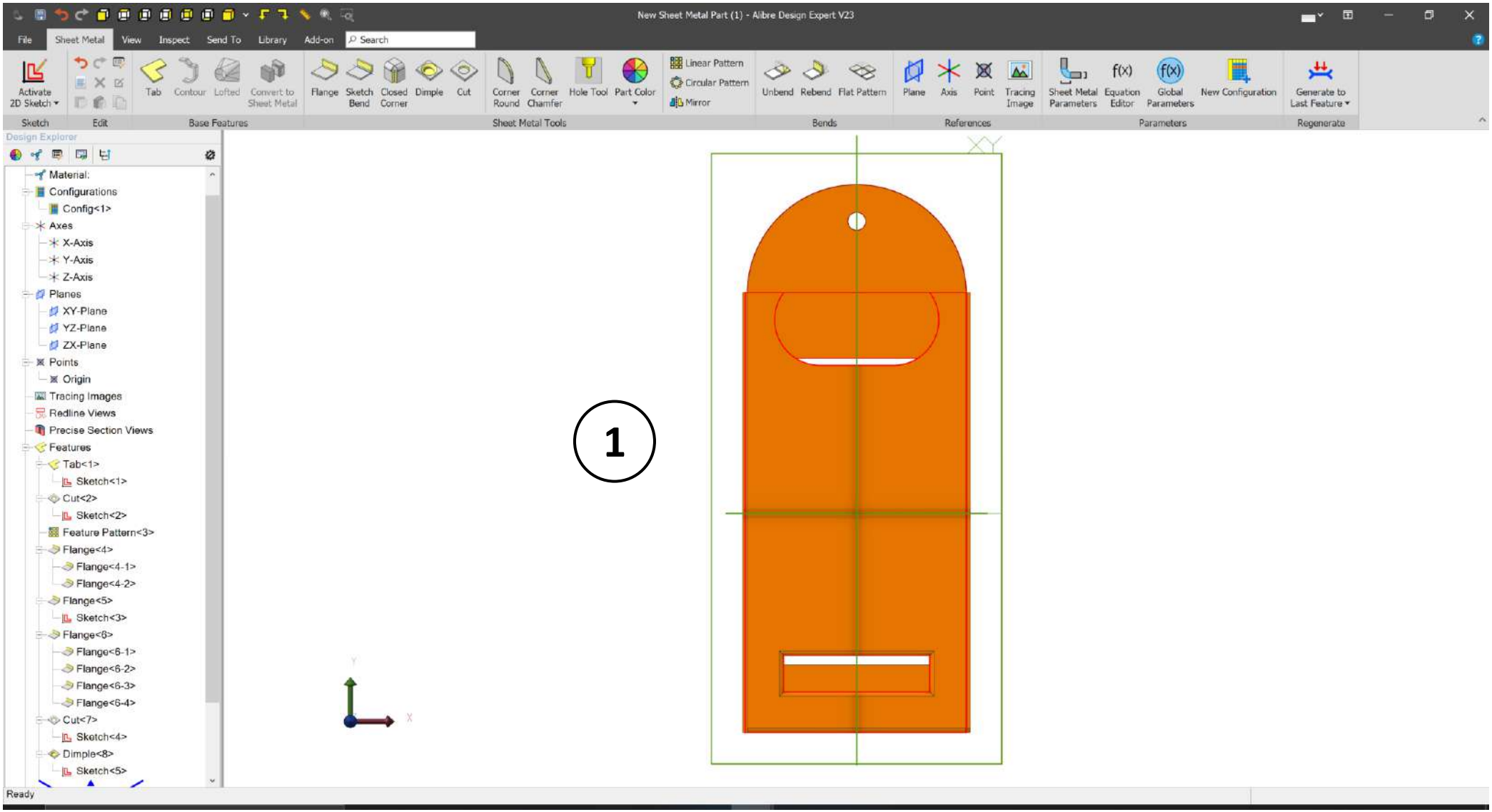
1. Click on the **Dimple** tool in **Sheet Metal Tools** section in the Ribbon.
2. Confirm that the sketch you just created appears in the Sketch field of the Dimple dialog.
3. Input a **1 degree** draft angle in the **Draft angle** field of the dimple dialog.
4. Enable the Checkbox named **Cut Out Material**.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



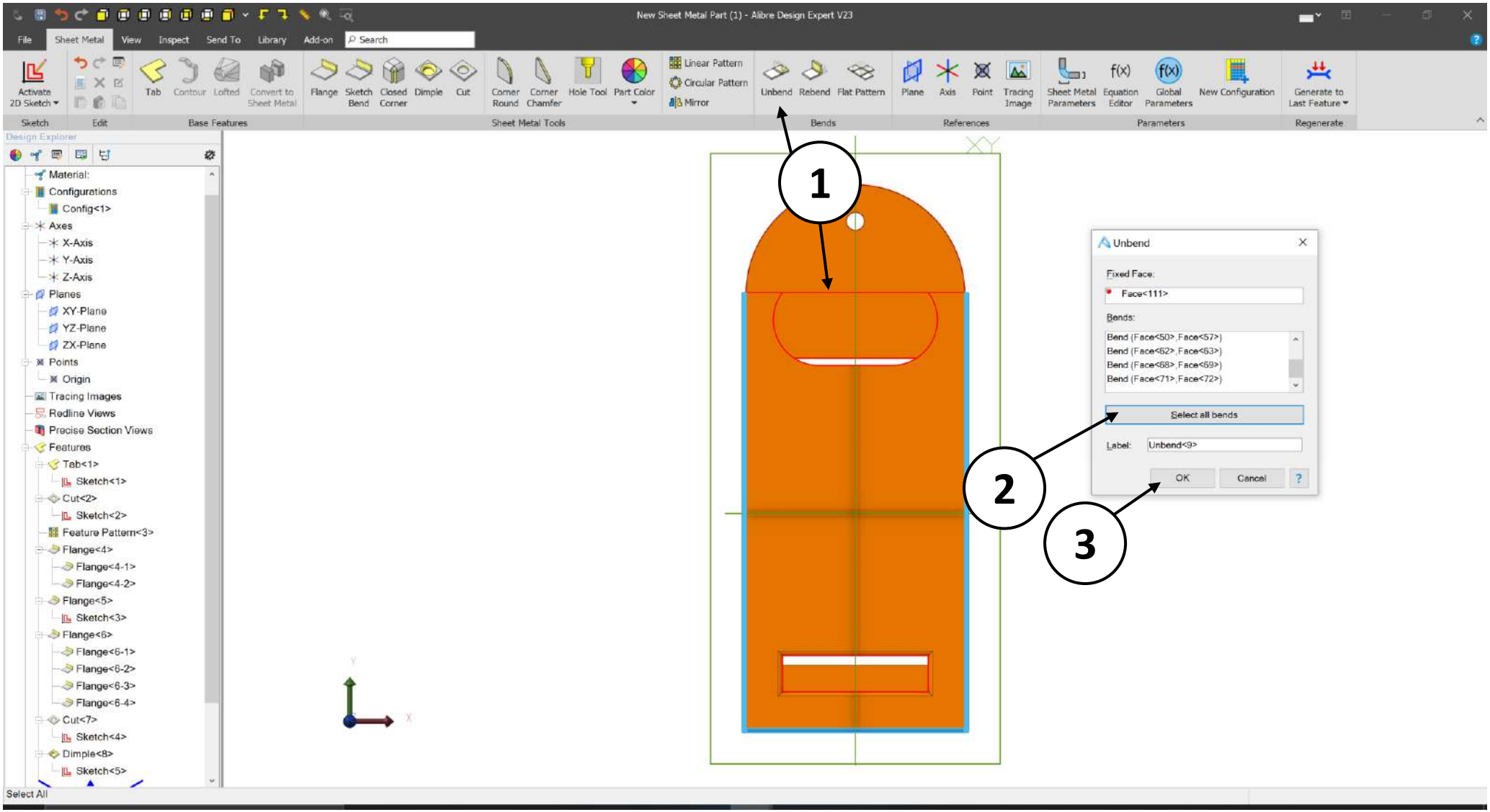
1. Click the **Part Color** button in the **Sheet Metal Tools** section of the Ribbon.



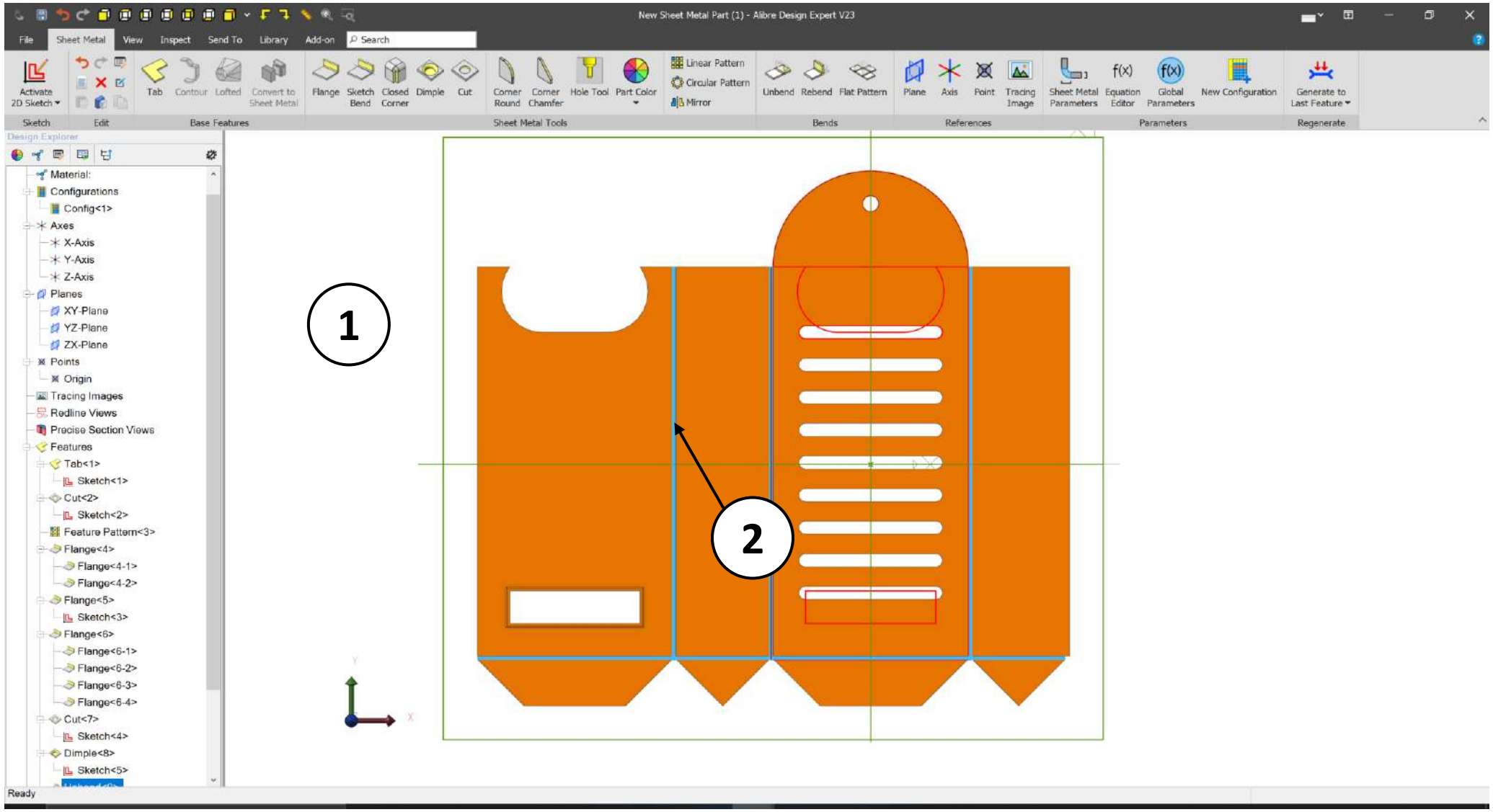
1. Click on **Color** in the **Color Properties** dialog.
2. Select a bright orange color in the **Color** dialog box by clicking on one of the colored tiles in the **Basic colors** area**.**
3. Click **OK** in the **Color** dialog box**.**
4. Click **OK** in the **Color Properties** dialog box.



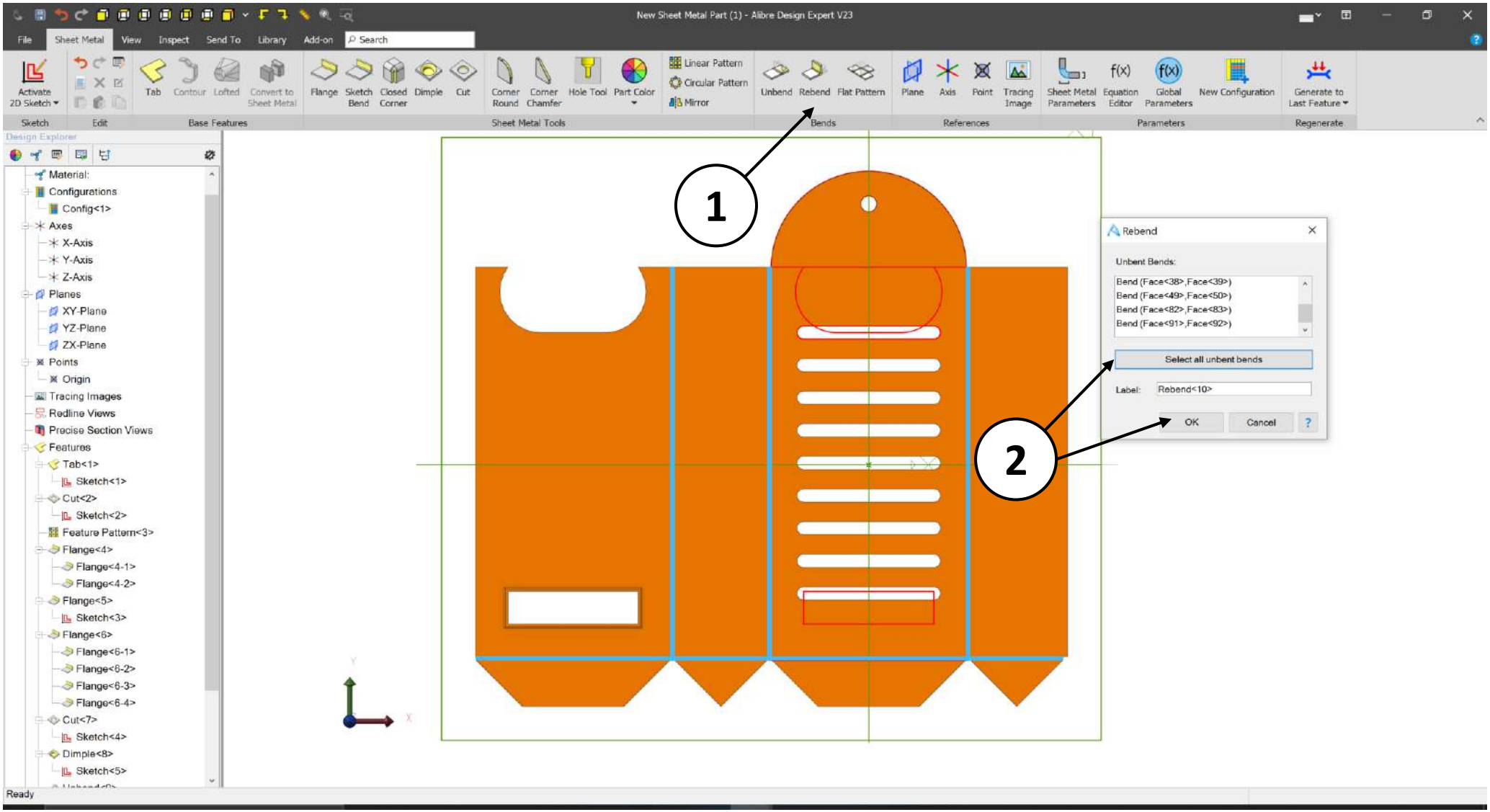
1. Confirm similar results for the **Color Properties** operation.



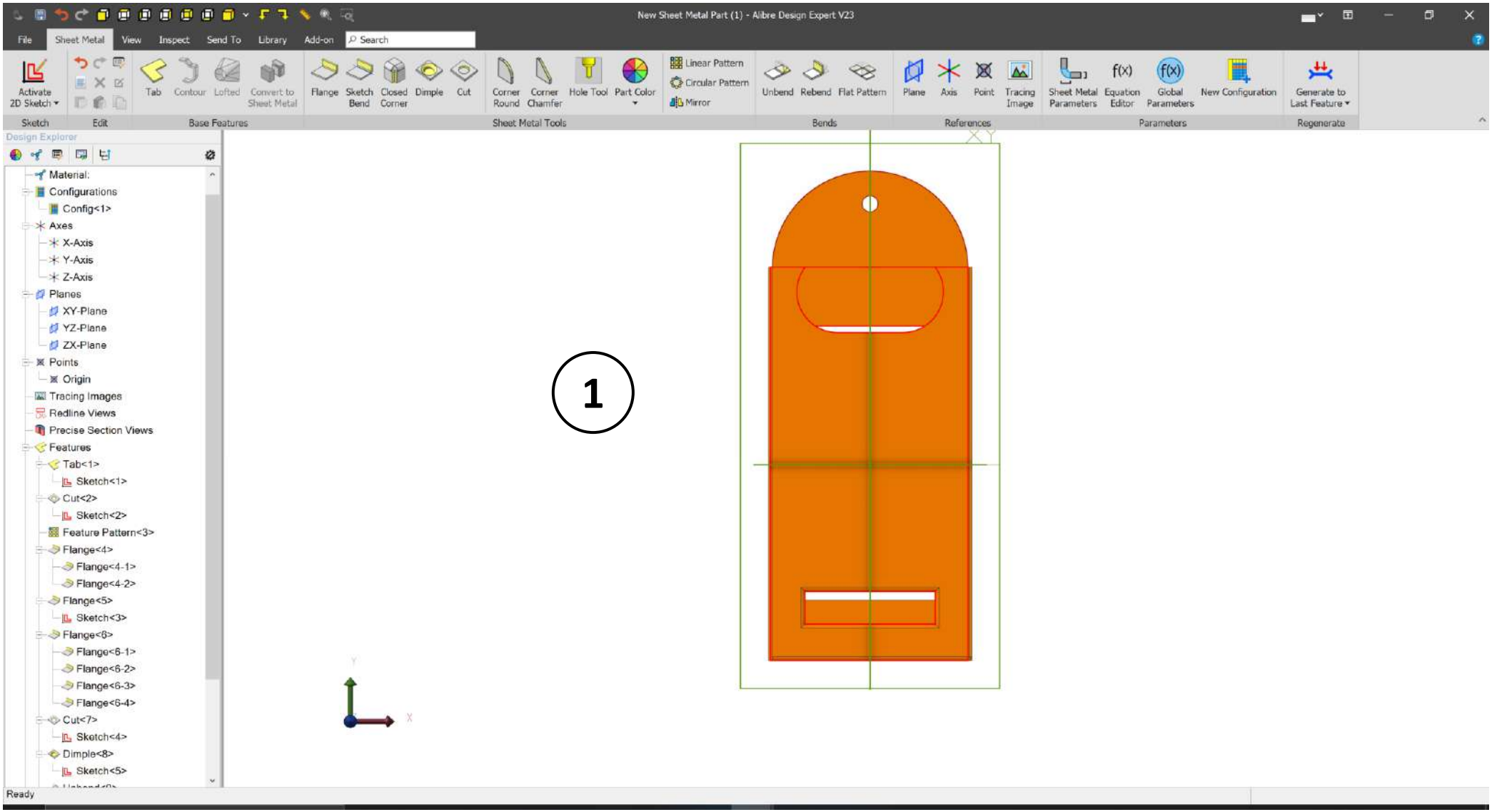
1. Click the **Unbend** tool and select the back face of the part as the **Fixed Face**.
2. Click on the **Select All Bends** option.
3. Click **OK.**



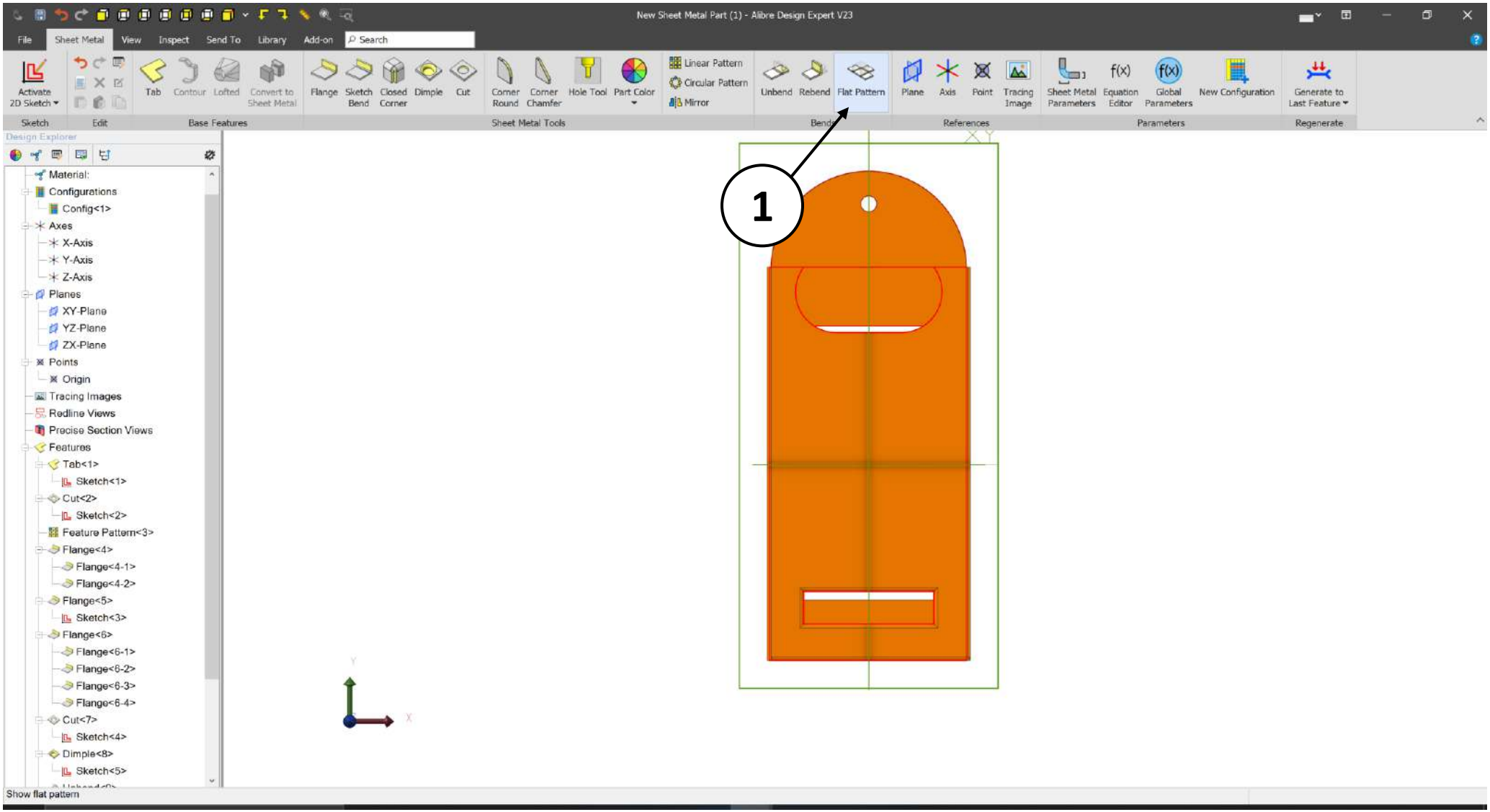
1. Confirm the results of the **Unbend** tool.
2. Blue color lines show the Bending Pattern.
   *Notice that the Pattern represents the bends as it would on the physical object.



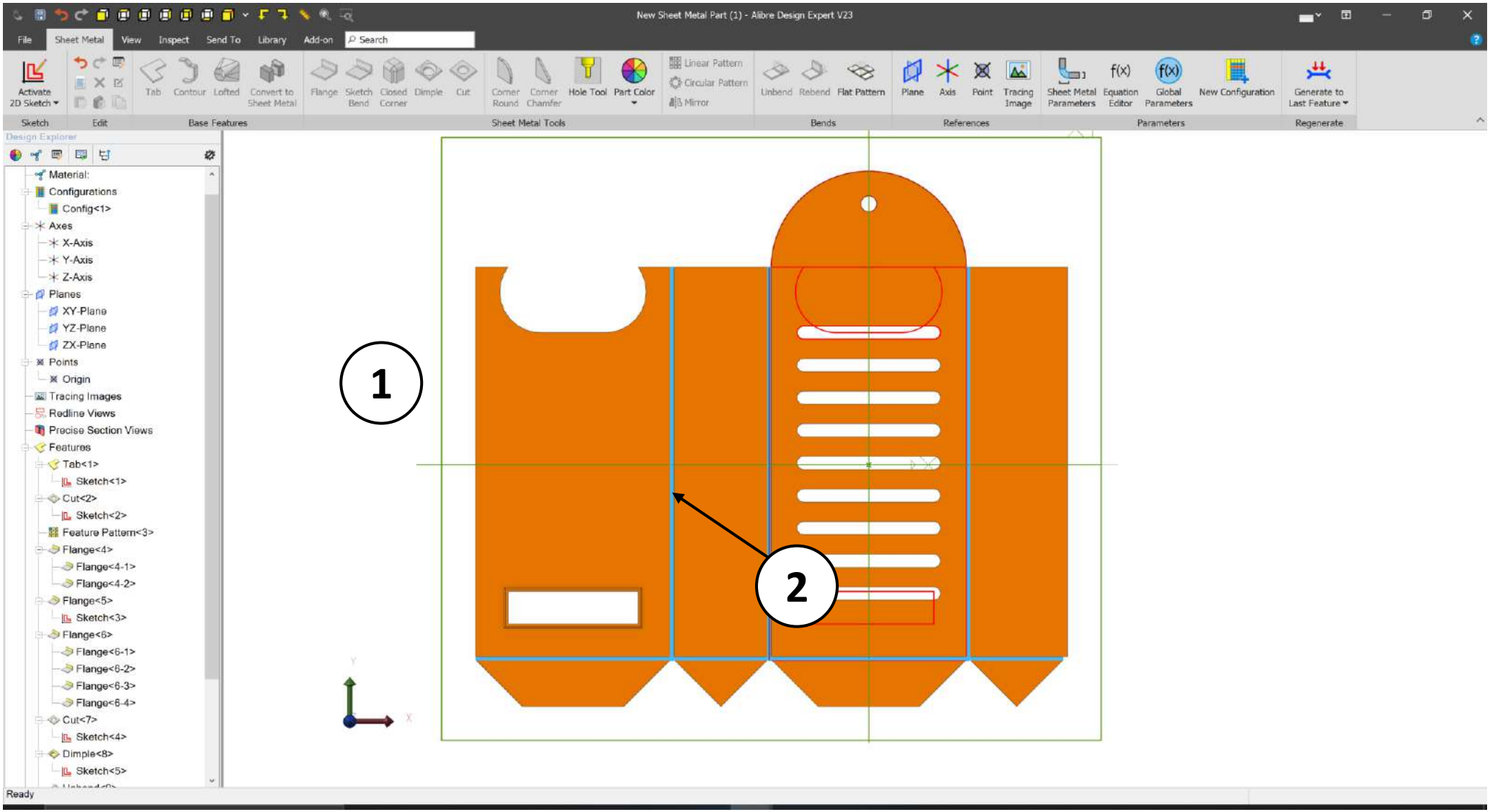
1. Click on the **Rebend** tool.
2. Click **Select all unbent bends**, and click **OK.**



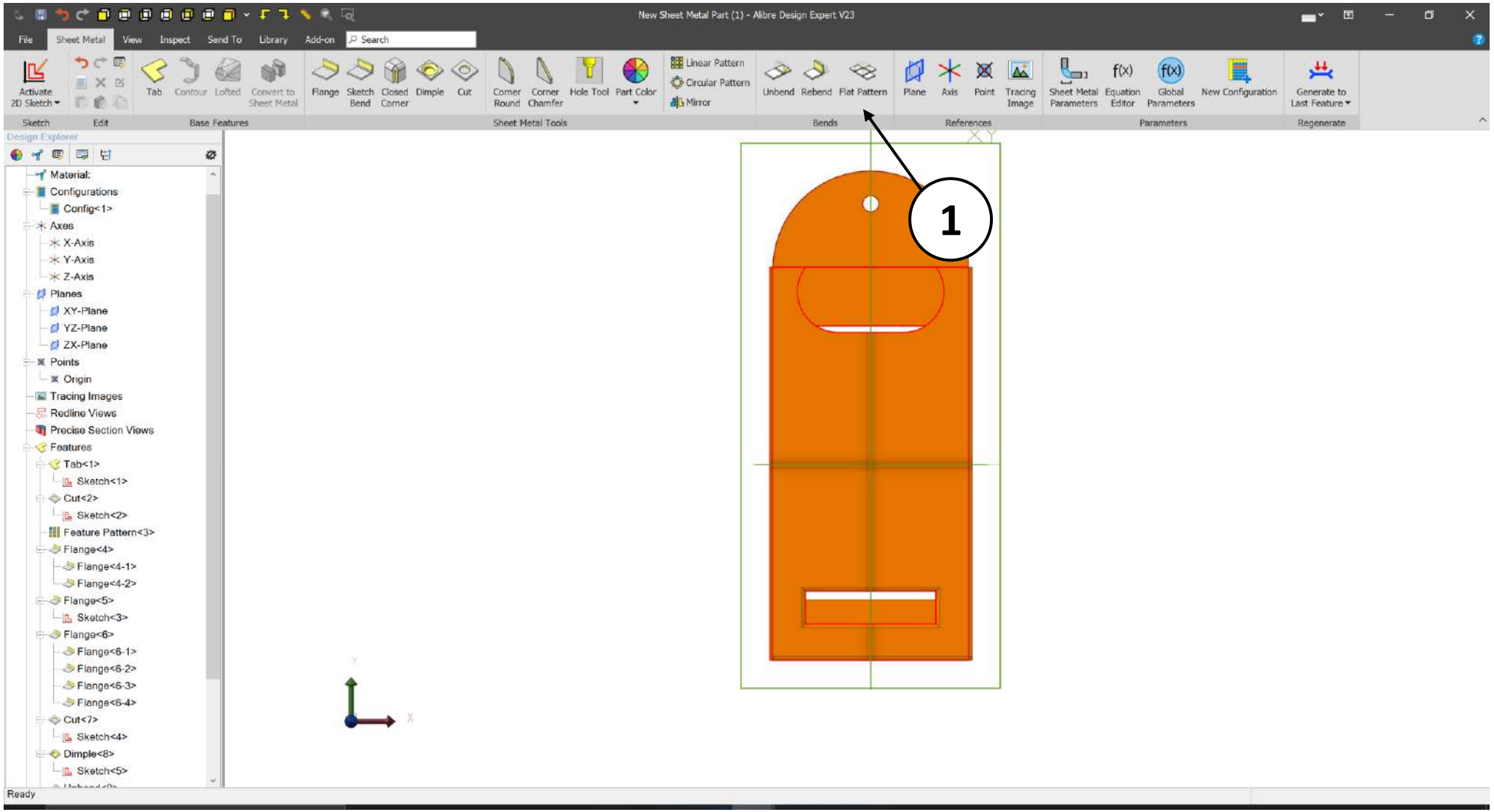
1. Confirm the results of the **Rebend**.



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.