

Alibre Design Exercise Manual

Introduction to creating parts and assemblies

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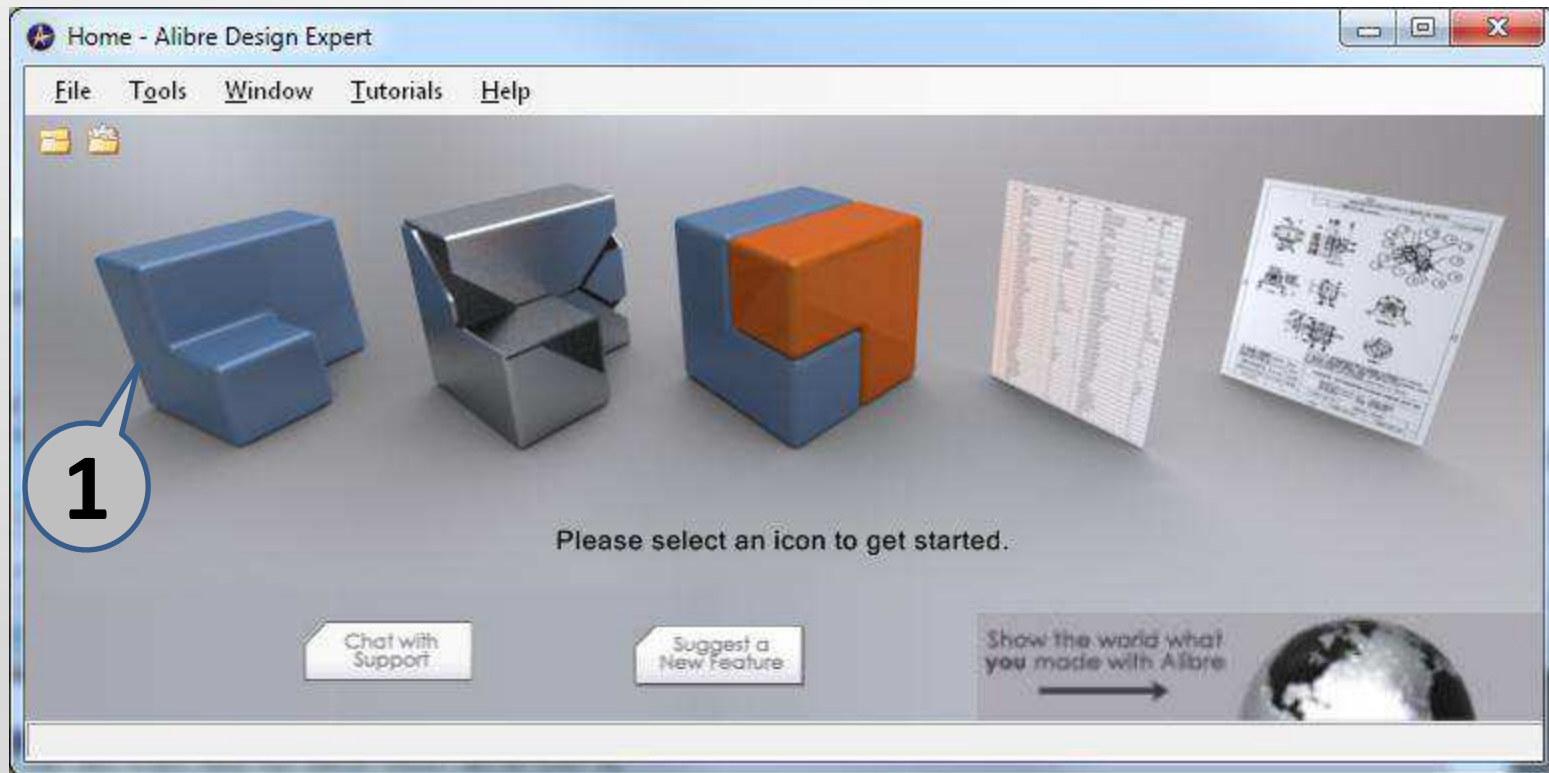
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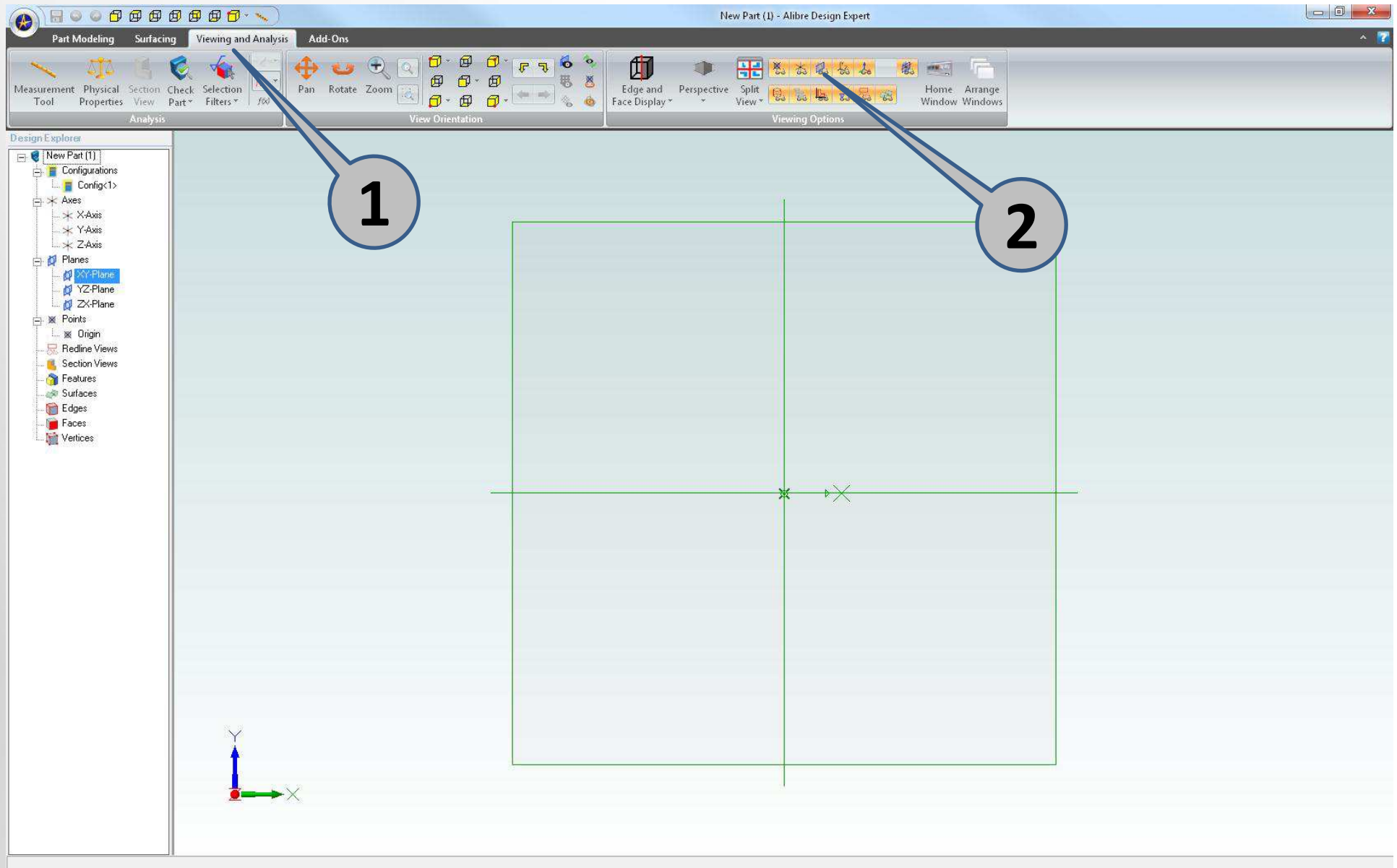
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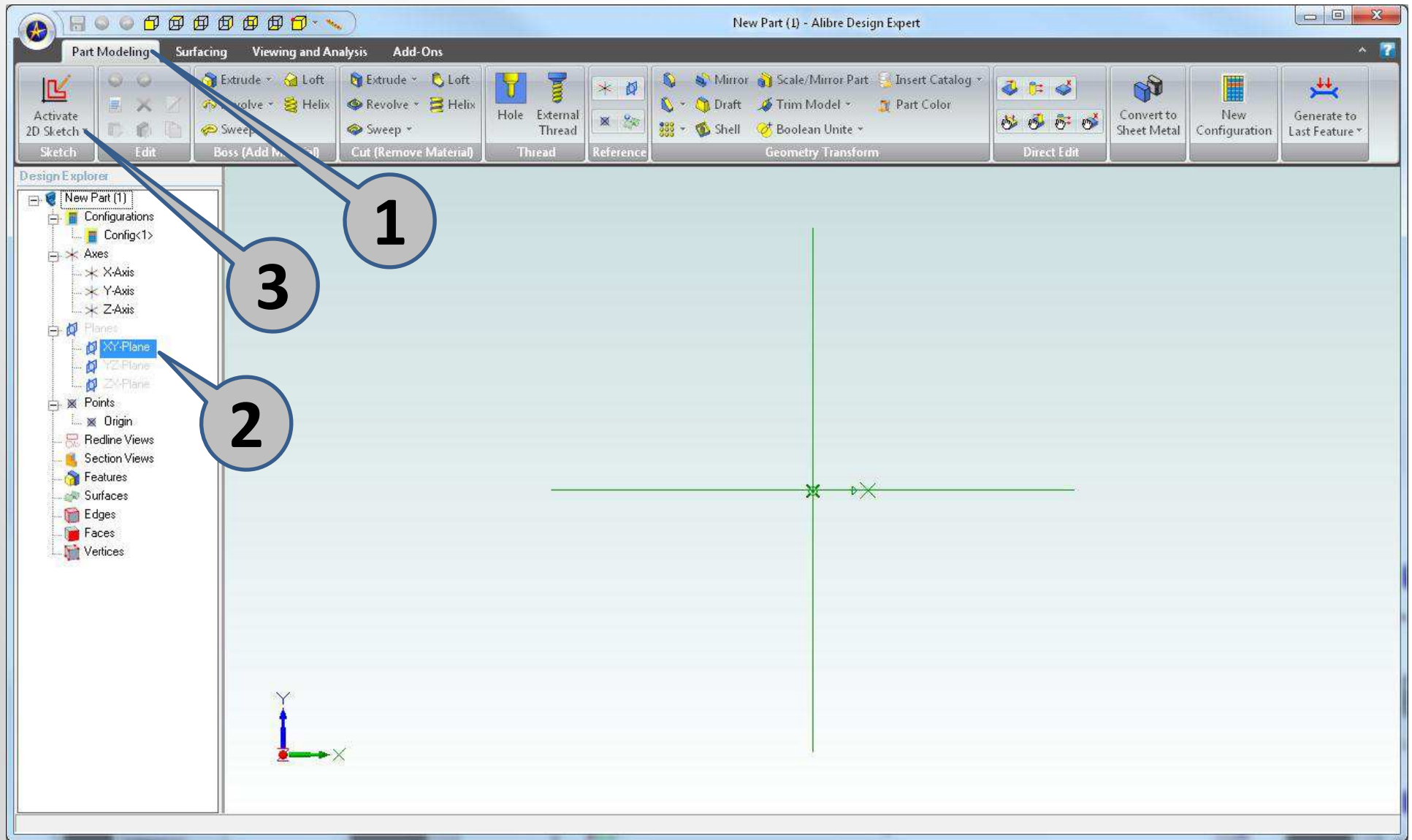
Creating the link



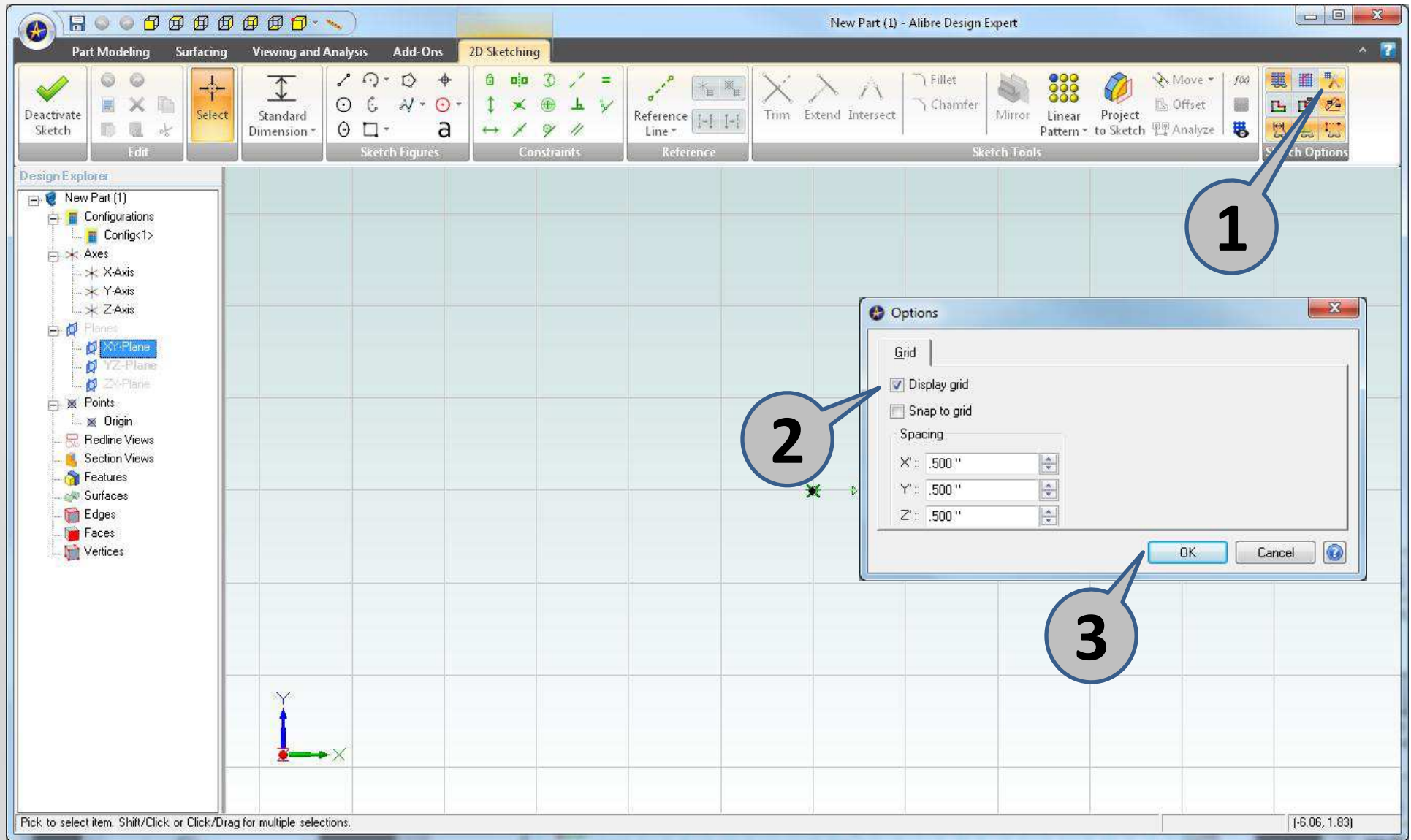
1. Open a new **Part** workspace from Alibre Design's **Home** window.



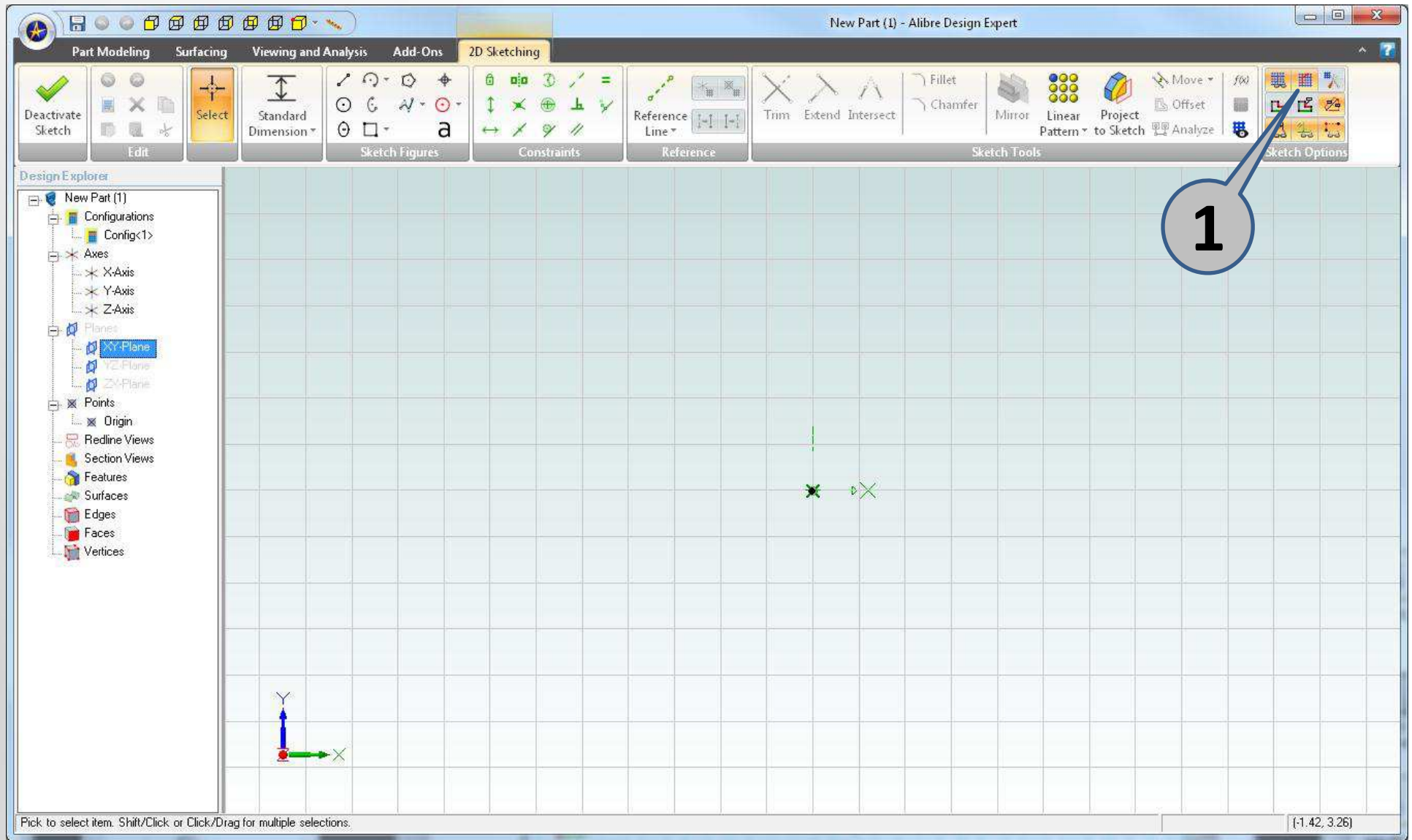
1. Click on the **Viewing and Analysis** tab in the ribbon.
2. Click the **Planes** option. This hides reference planes from view in the work area. Planes need not be visible for this tutorial.



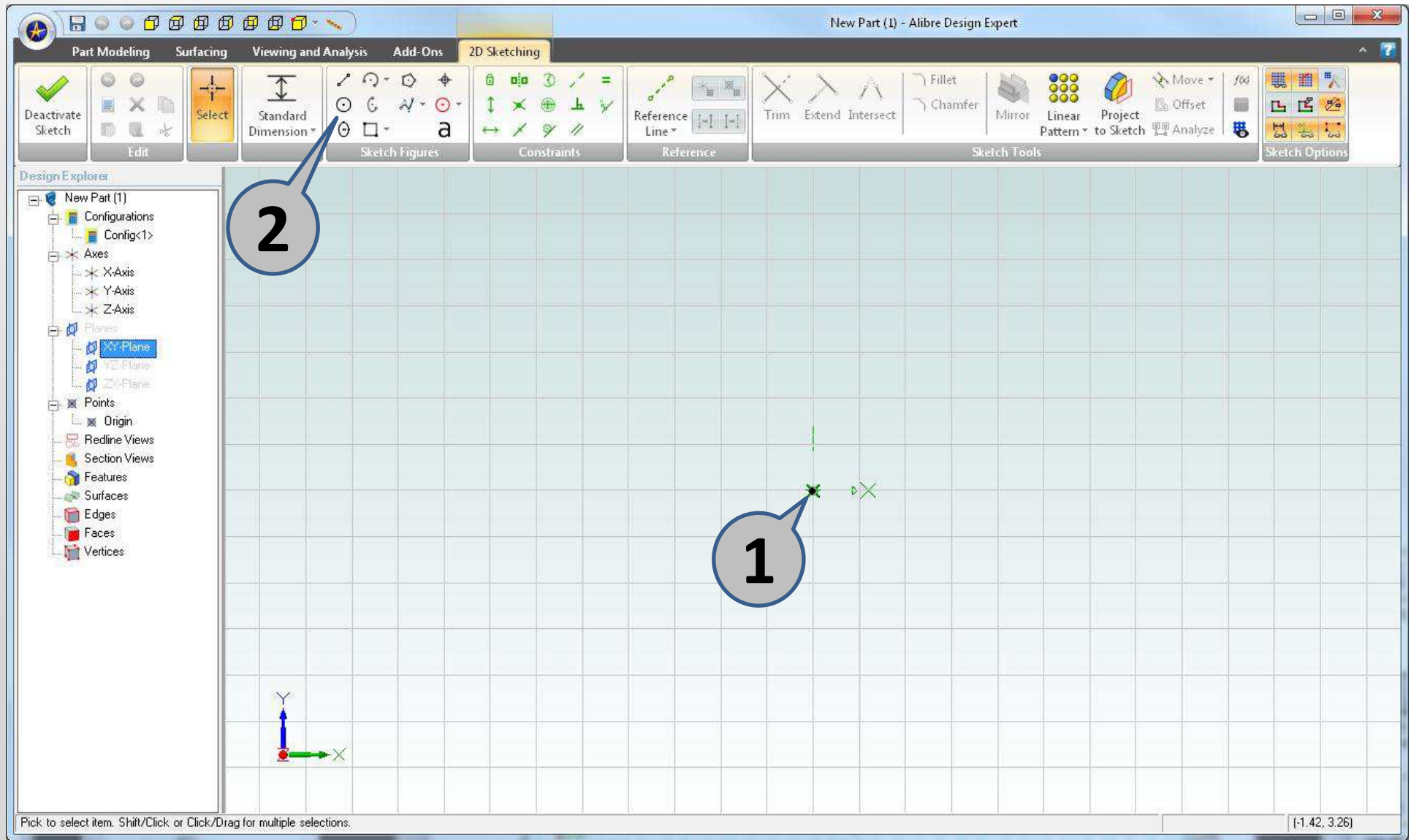
1. If necessary, click on the **Part Modeling** tab in the ribbon.
2. Click on the **XY Plane** in the **Design Explorer**.
3. Click on the **Activate 2D Sketch** icon.



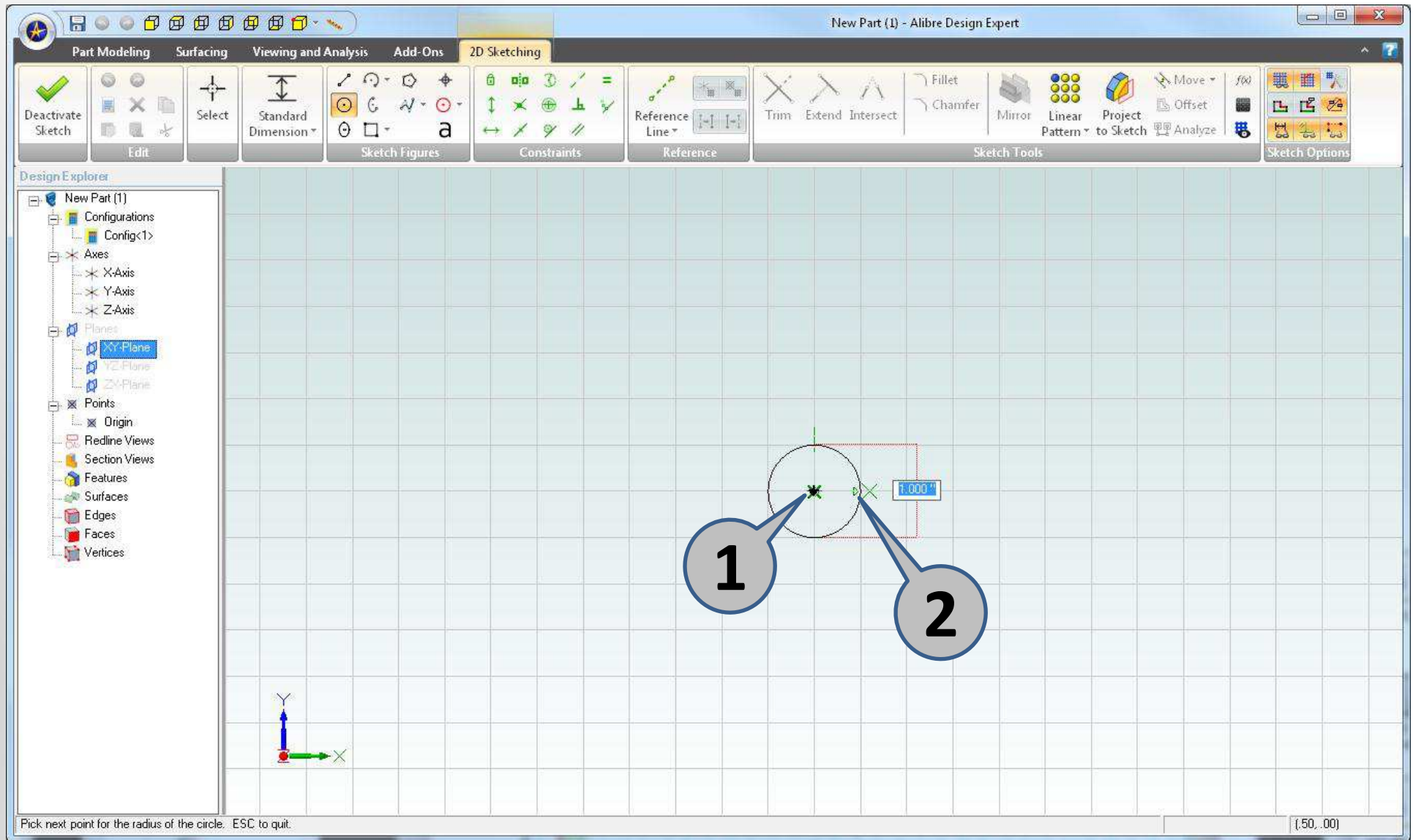
1. Select the **Grid** options and change the values to .500 inch for X, Y, and Z.
2. Enable the **Display Grid** option if it is unchecked.
3. Press the **OK** button.



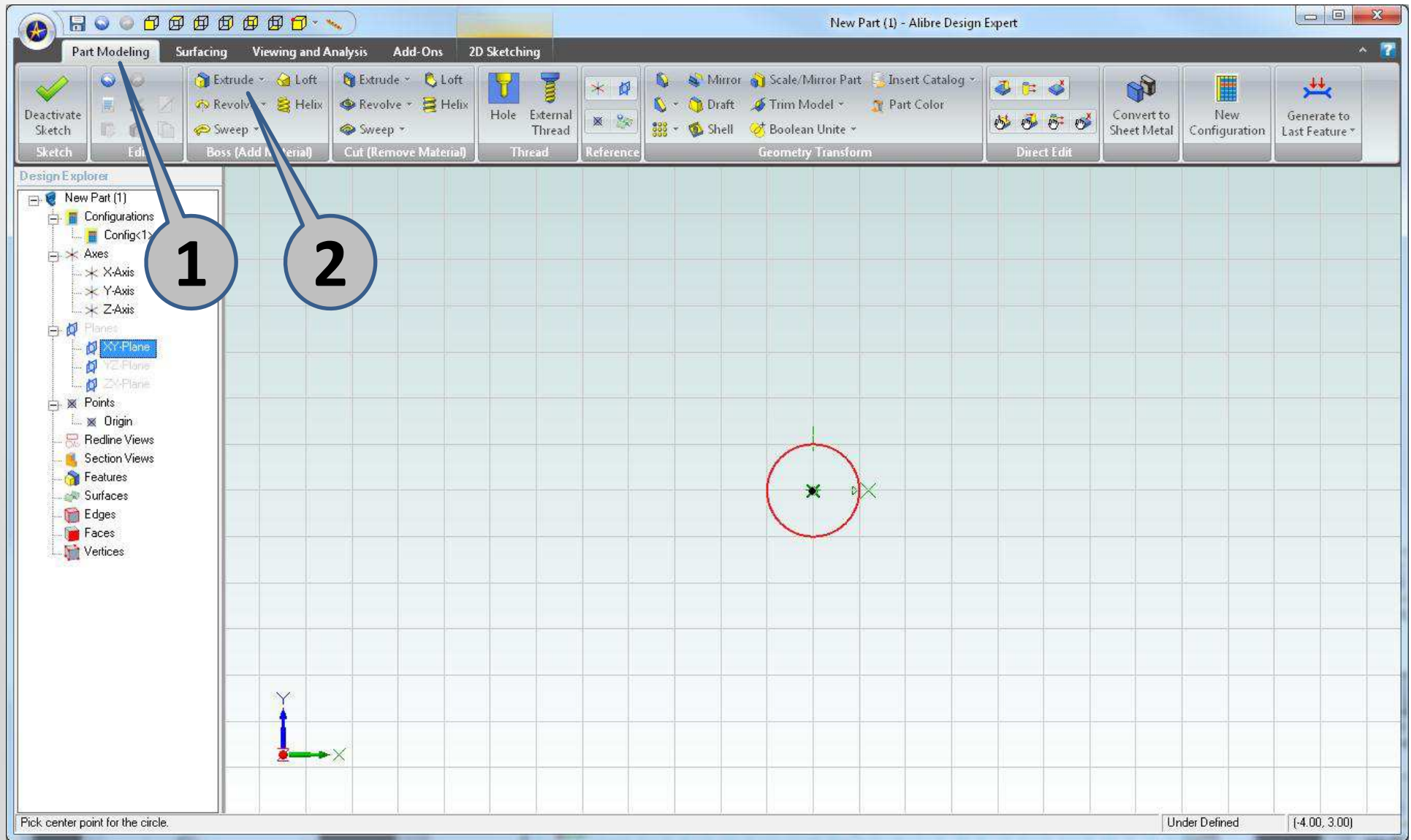
1. Click the **Snap to Grid** option.



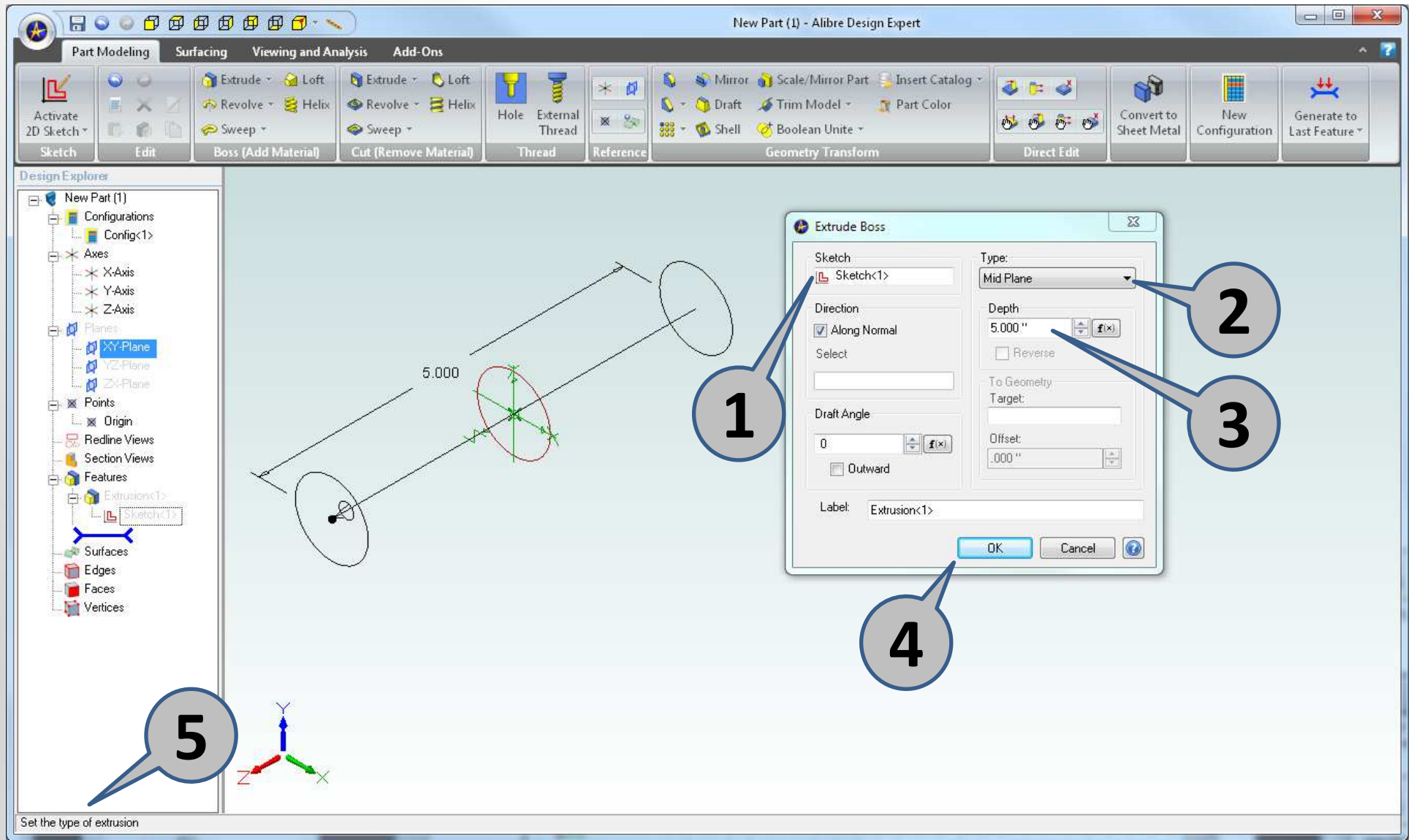
1. Position the mouse cursor near the **Origin** point. Roll the mouse wheel forward to zoom in. The view of the work area should be similar to the screen capture (approximately 5 grid squares visible, vertically).
2. Click on the **Circle** tool on the **Sketch Figures** tab in the ribbon.



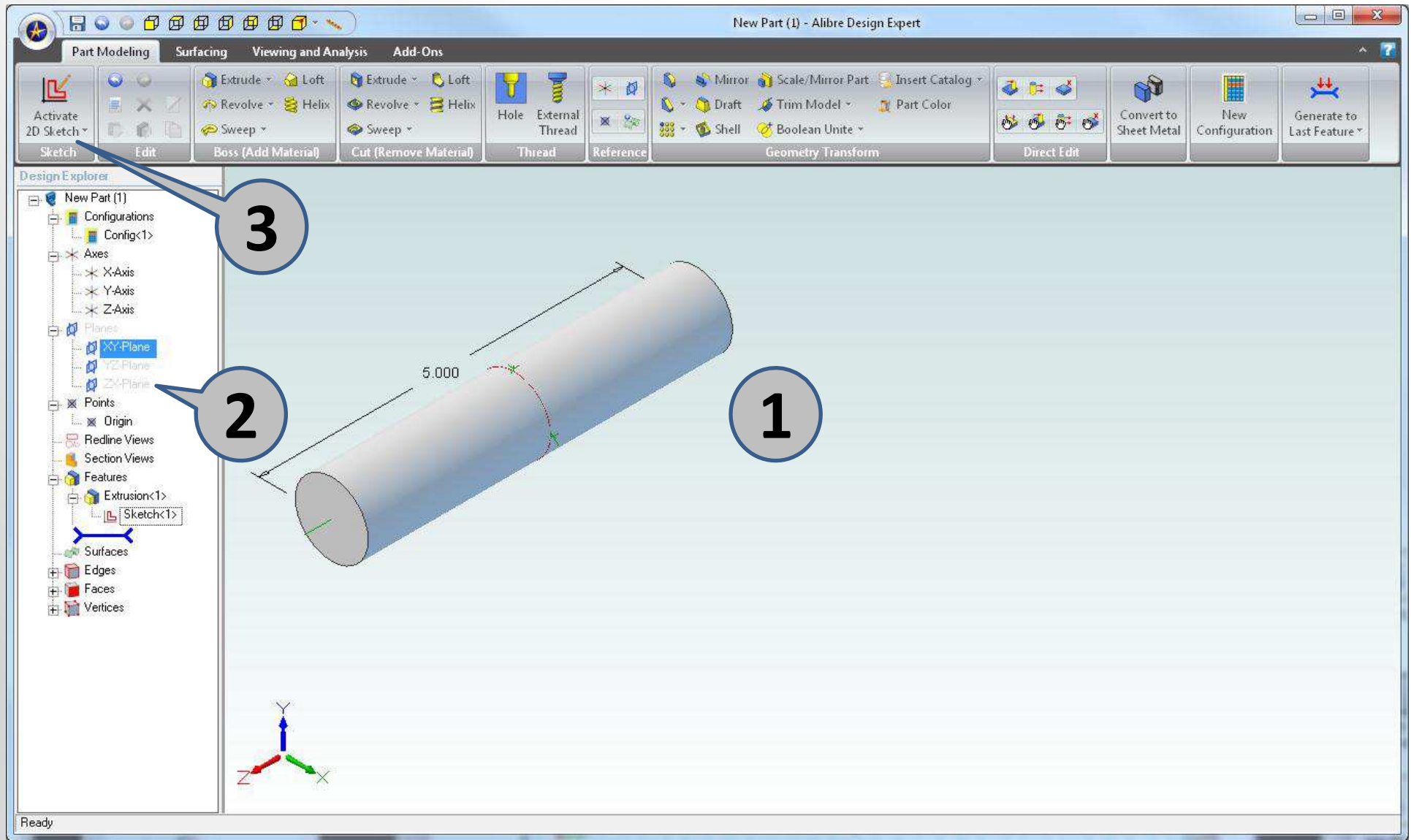
1. Position the mouse cursor over the **Origin** point. The point's color change confirms that the mouse cursor is positioned properly. *Click and release* to create the center point of the circle.
2. Move the mouse cursor a distance of one grid square to the right, along the **X Axis**. *Click and release* to complete the circle; this circle should be 1.000". Confirm that the results are similar to the screen capture.



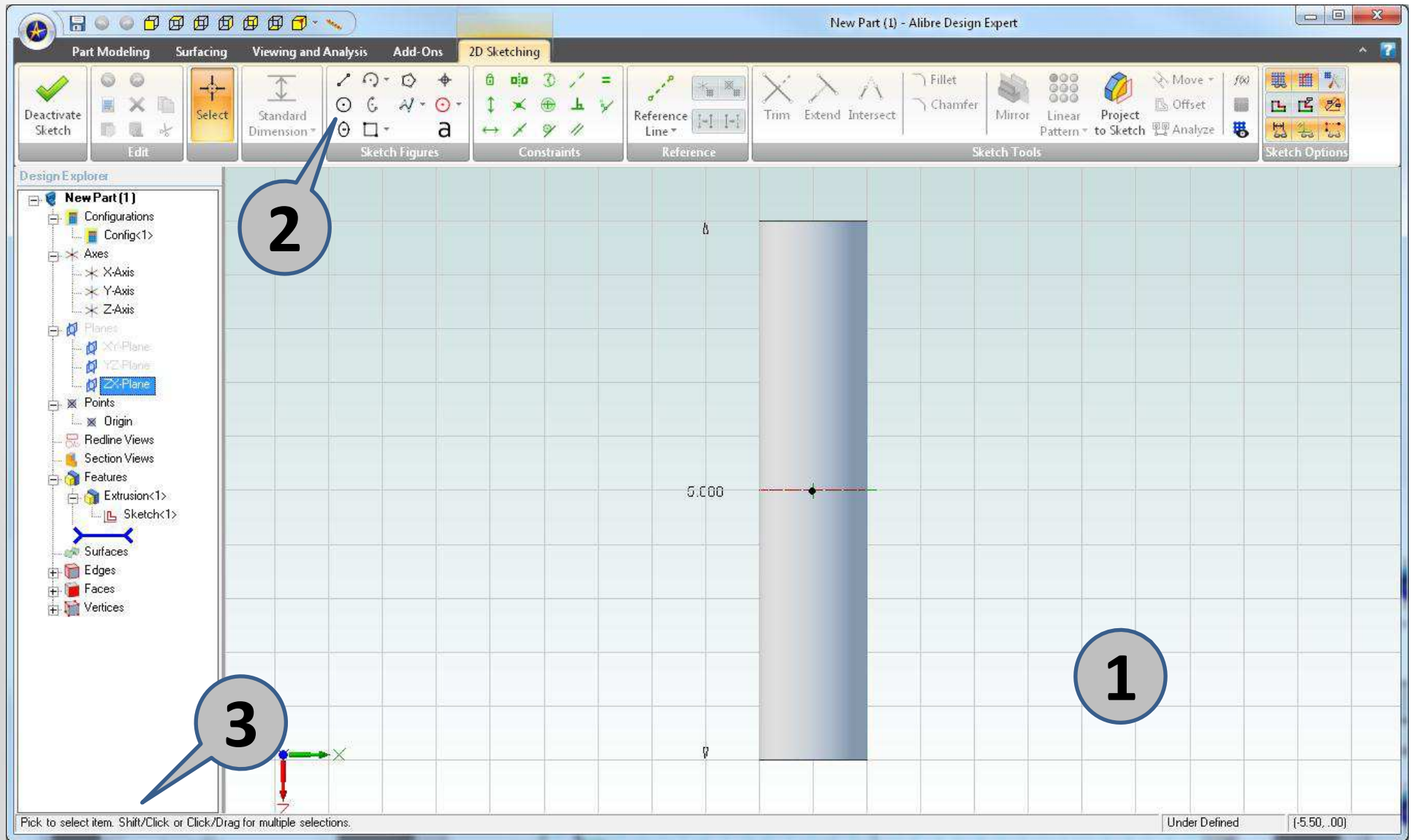
1. Click on the **Part Modeling** tab in the ribbon
2. Click on the **Extrude Boss** icon in the upper-right corner of the screen.



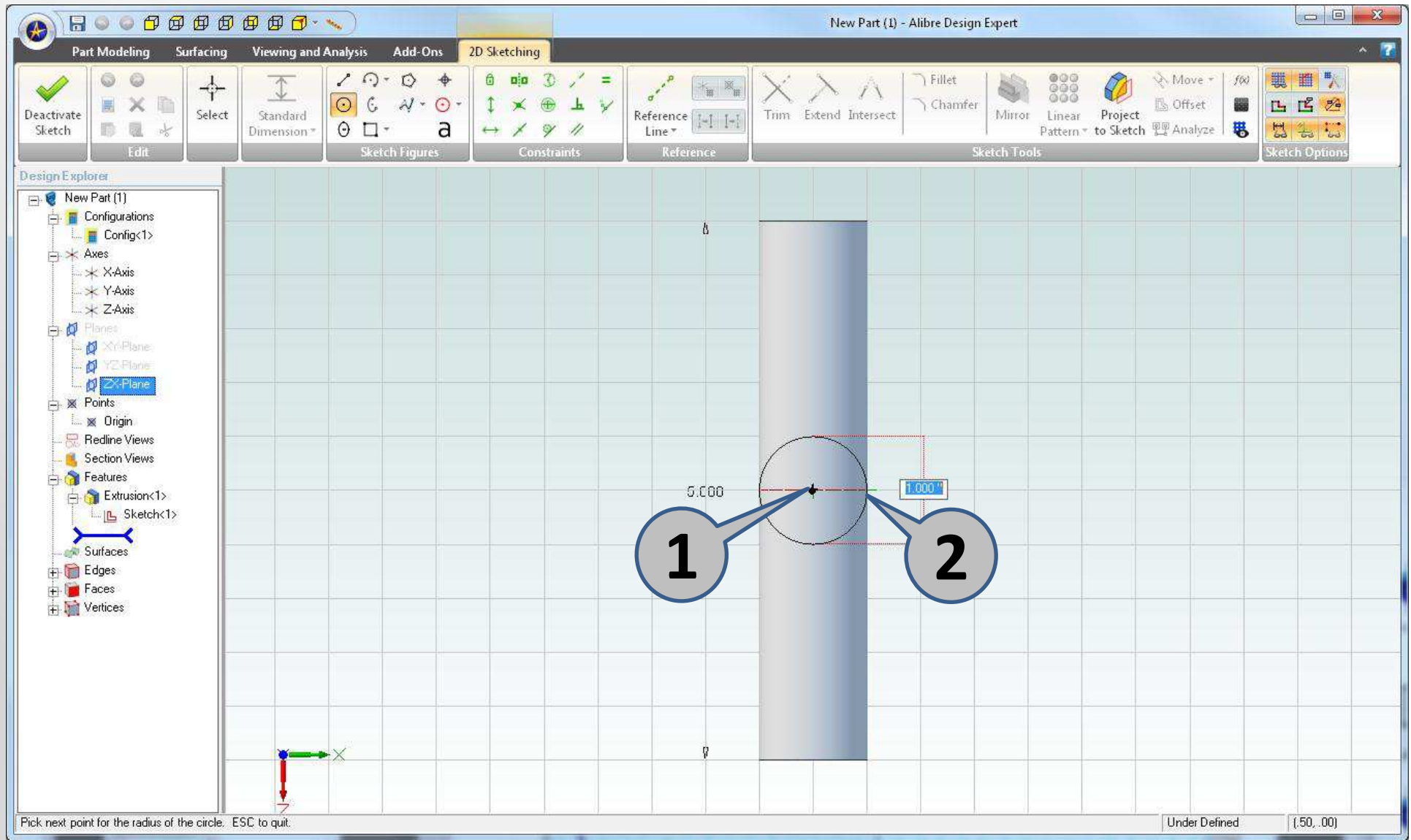
1. In the **Extrude Boss** dialog box, confirm that the Sketch entry box contains **Sketch<1>**.
2. Change the **Type** drop-down menu to the **Mid Plane** option.
3. Confirm that the **Depth** entry box has a value of **5 inches**. Enter that value if it does not.
4. Click **OK**.
5. Alibre Design will often tell you what it expects.



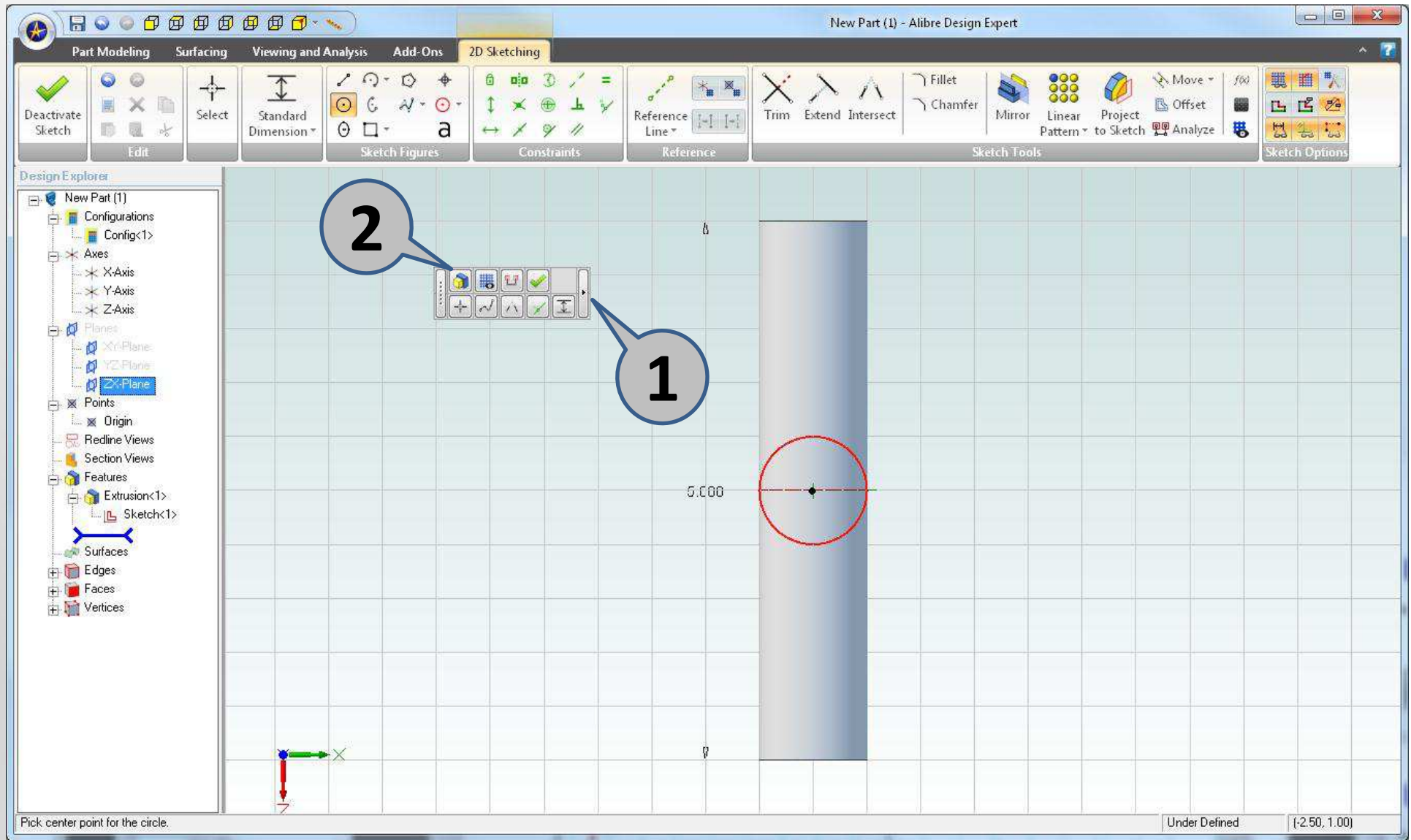
1. Confirm similar results for the Extrude Boss operation.
2. Select the **ZX** Plane in the **Design Explorer**.
3. Click **Activate 2D Sketch** icon in the **Part Modeling** tab.



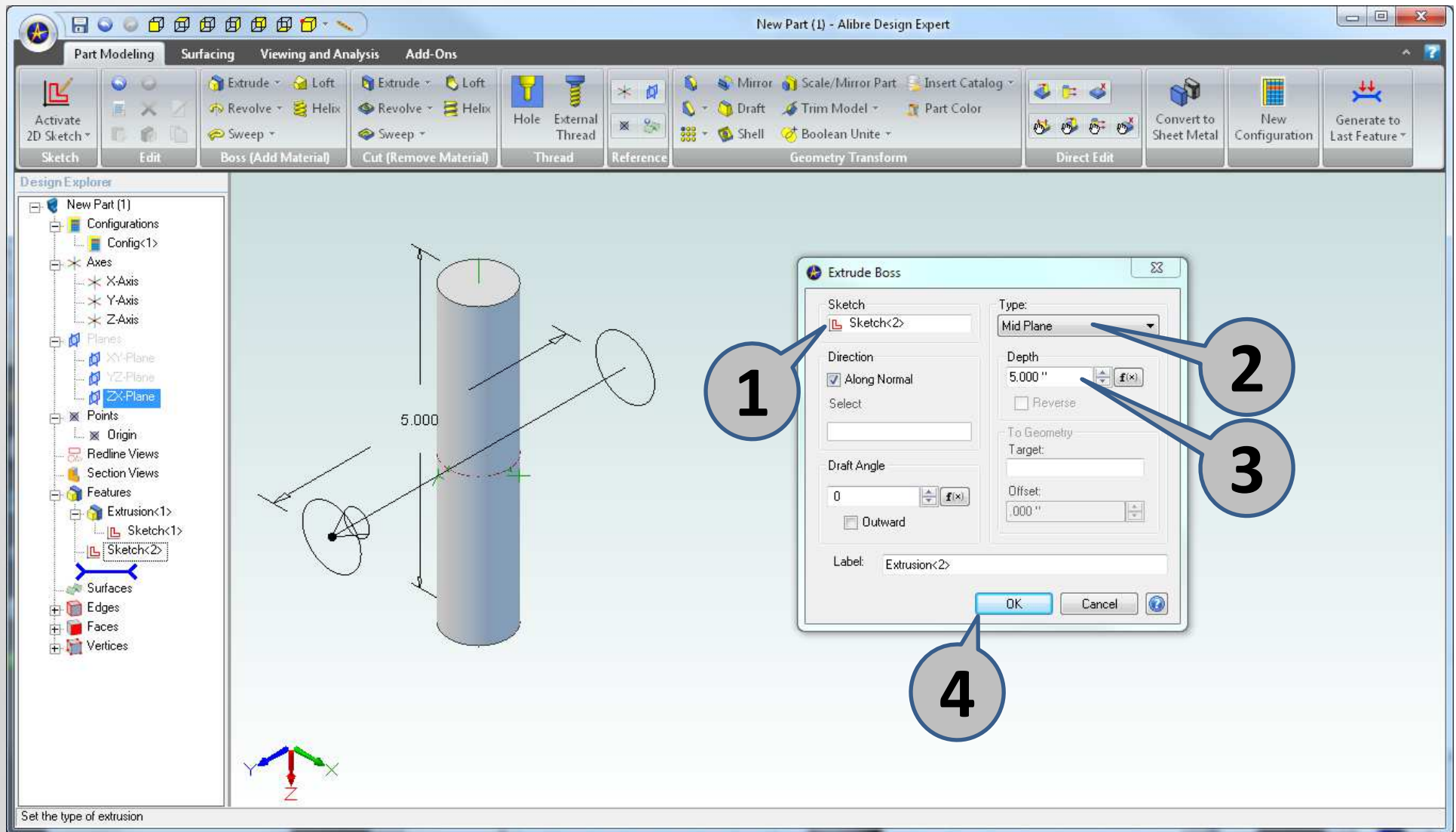
1. The view of the work area should be similar to the screen capture.
2. Click on the **Circle tool**.
3. Alibre Design will often tell you what it expects.



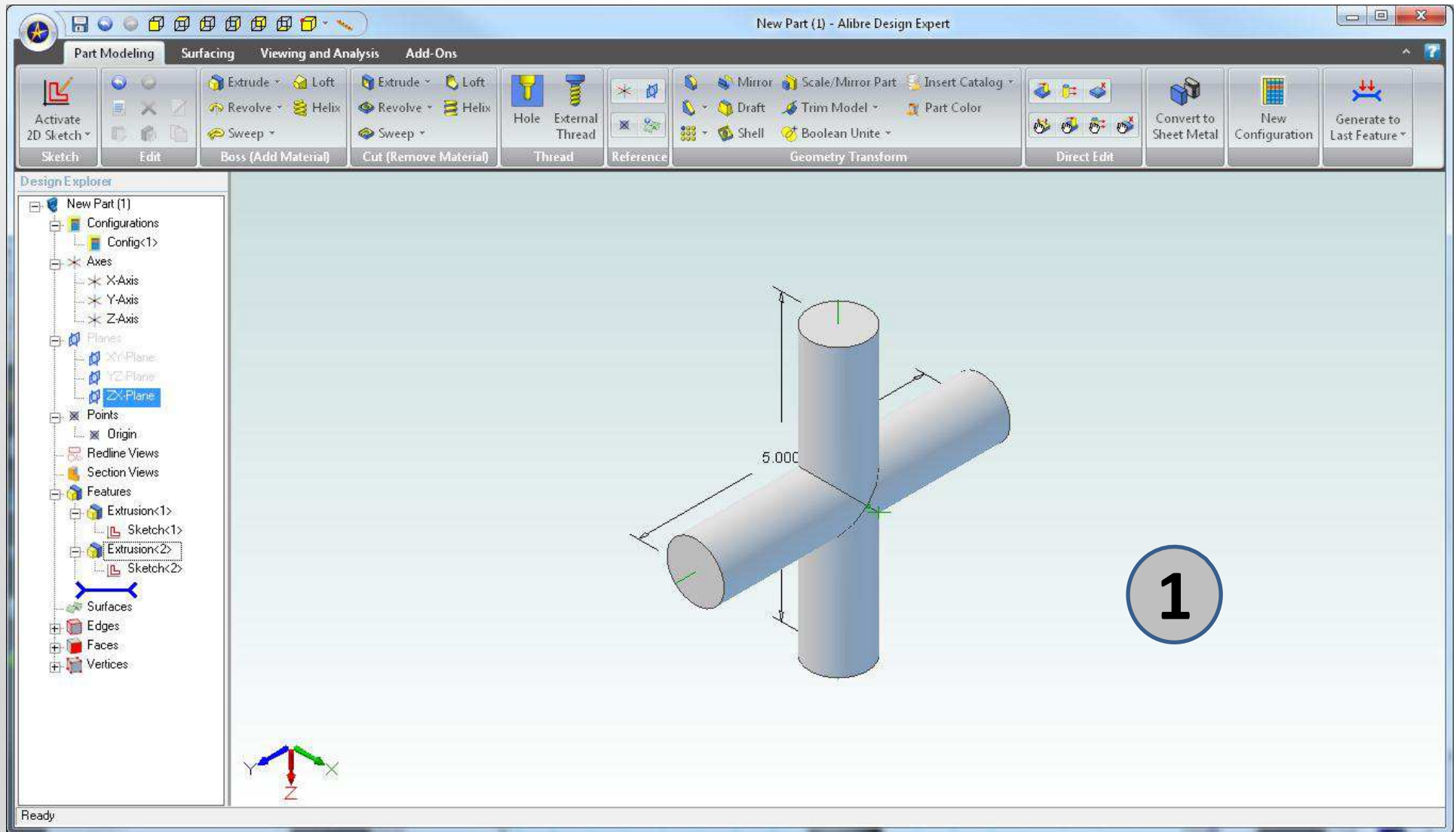
1. Position the mouse cursor over the **Origin** point. *Click and release* to create the center of the circle.
2. Move the mouse cursor to the position indicated; this circle should also be 1.000". *Click and release* to complete the circle.



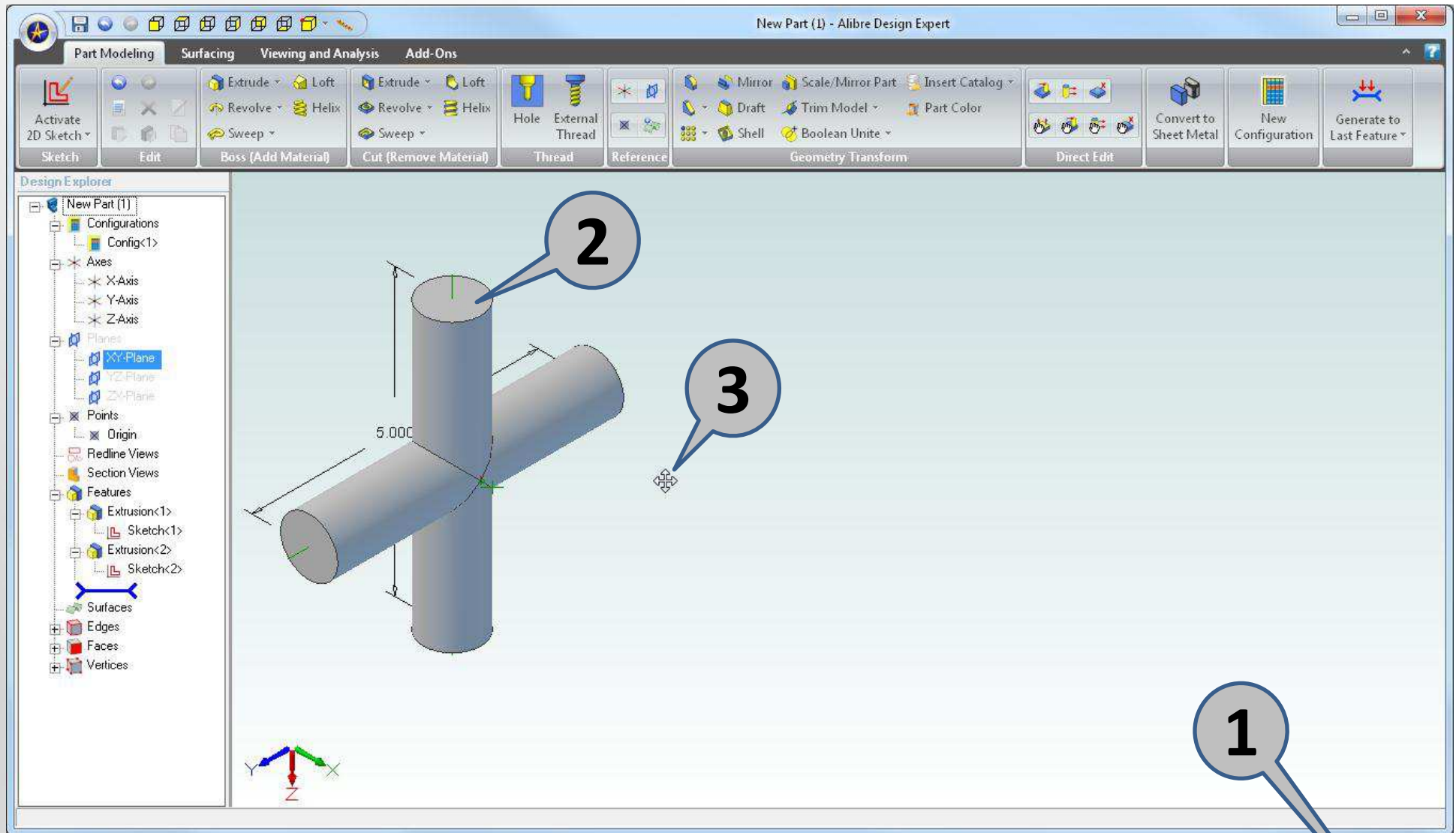
1. Press the spacebar and a context sensitive menu will appear.
2. Click on **Extrude Boss** icon.



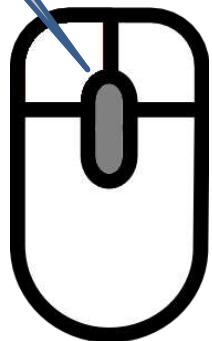
1. In the **Extrude Boss** dialog box, confirm that the **Sketch** entry box contains **Sketch<2>**.
2. Change the **Type** drop-down menu to the **Mid Plane** option.
3. Confirm that the **Depth** entry box has a value of **5 inches**. Enter that value if it does not.
4. Click **OK**.

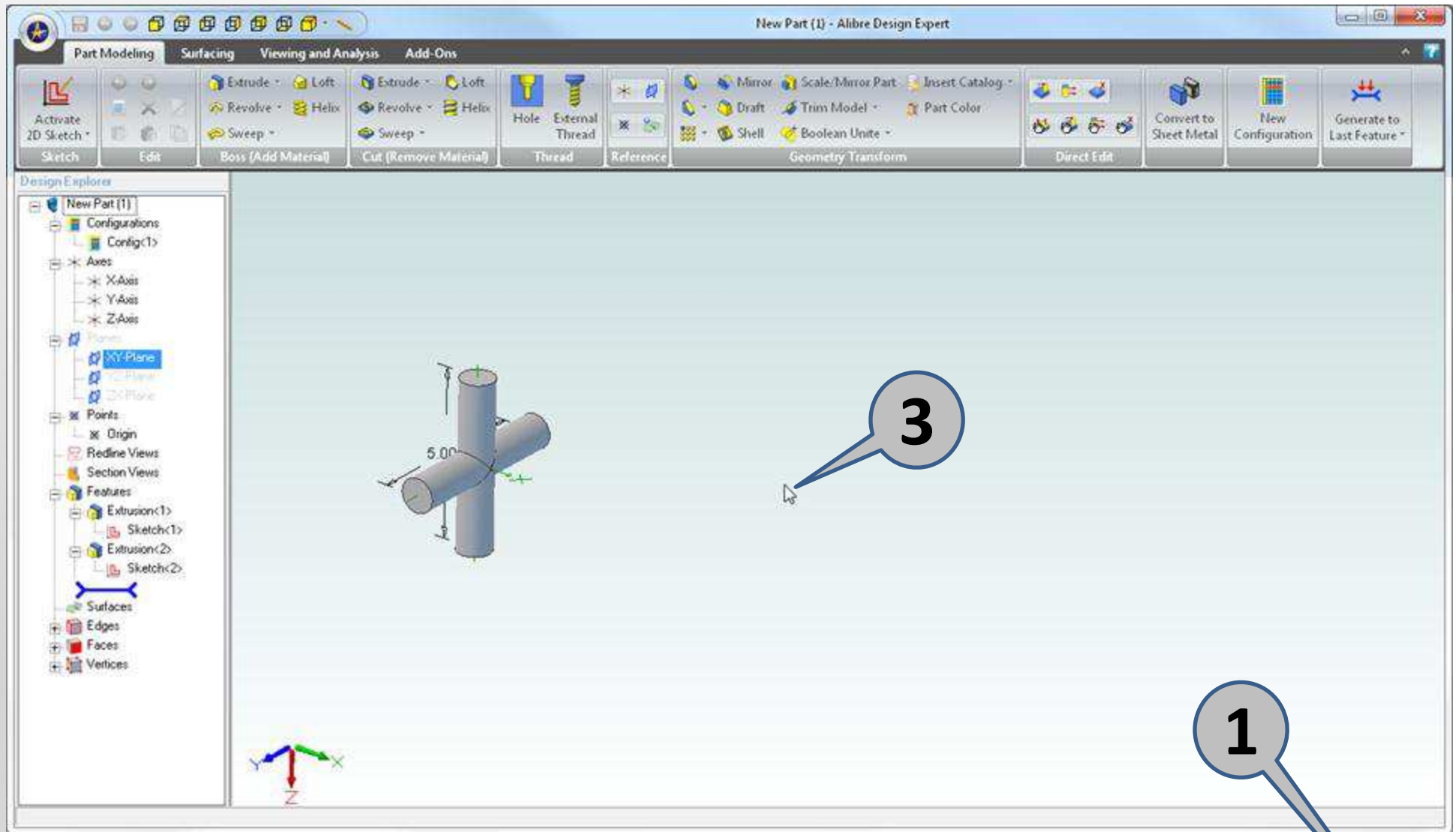


1. Confirm similar results for the **Extrude Boss** operation.

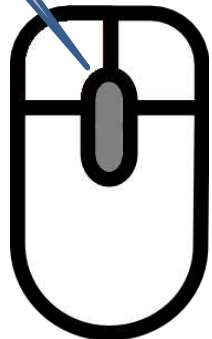


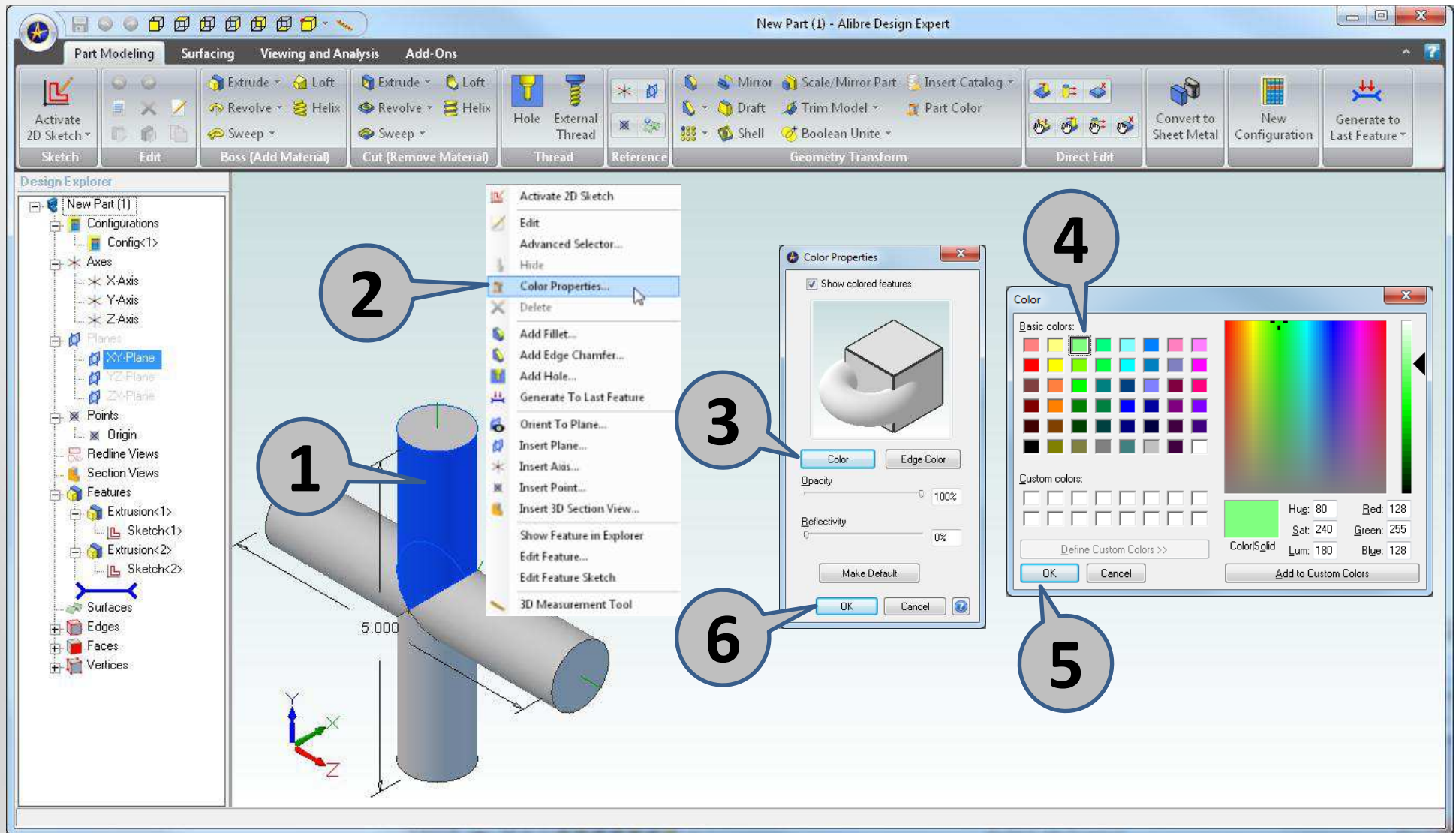
1. You can pan the model by holding down the middle mouse button (scroll button) and moving the mouse. This will keep the model in the same orientation in the workspace but move it left, right, up, and down.
2. Press and hold the middle mouse button and move the part in the workspace to the left.
3. When you press and hold the middle mouse button you will notice that mouse cursor changes.



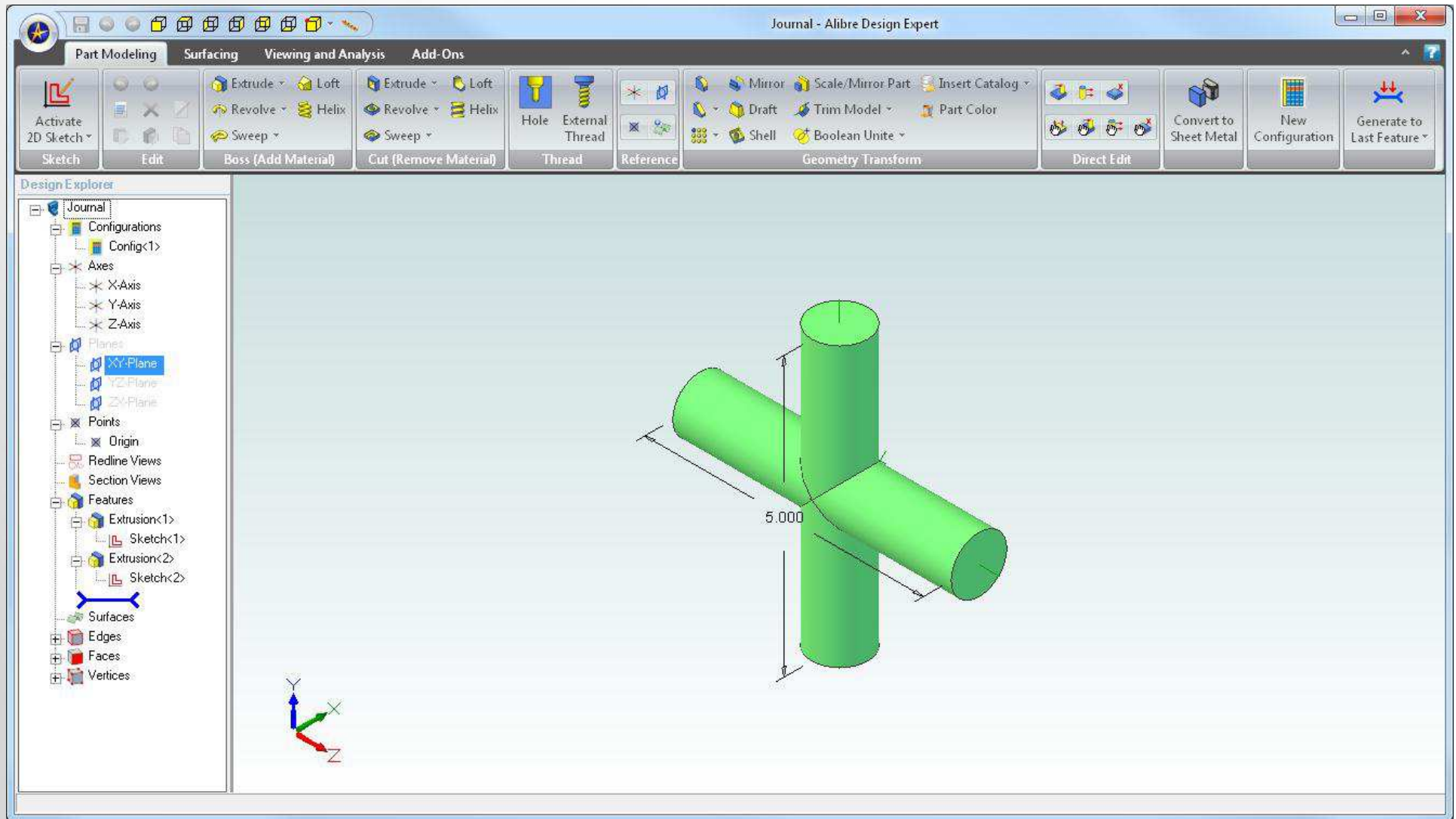


1. By scrolling the middle mouse button you can zoom into or away from the current model. Moving your finger towards you will zoom out and moving your finger away from you will zoom in.
2. The zoom action will be centered around the location of the mouse.
3. With the mouse cursor in the center of the workspace, zoom away from the model.





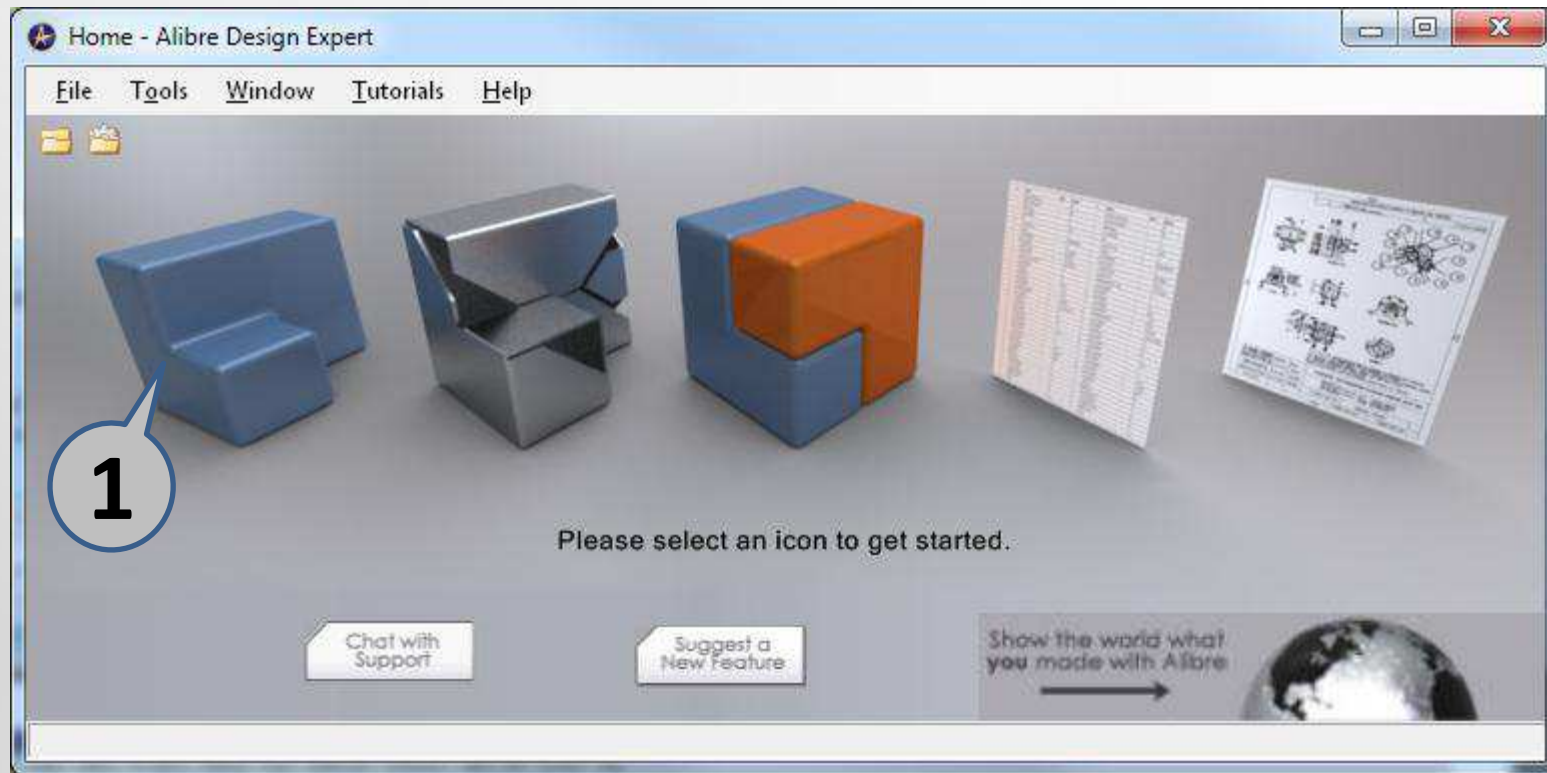
1. Right click anywhere on the part.
2. From the pop-up menu select Color Properties.
3. Click on the **Color** button in the **Color Properties** dialog box.
4. Select a bright green color in the **Color** dialog box by clicking on one of the colored squares in the **Basic colors area**.
5. Click **OK** in the **Color** dialog box.
6. Click **OK** in the **Color Properties** dialog box.



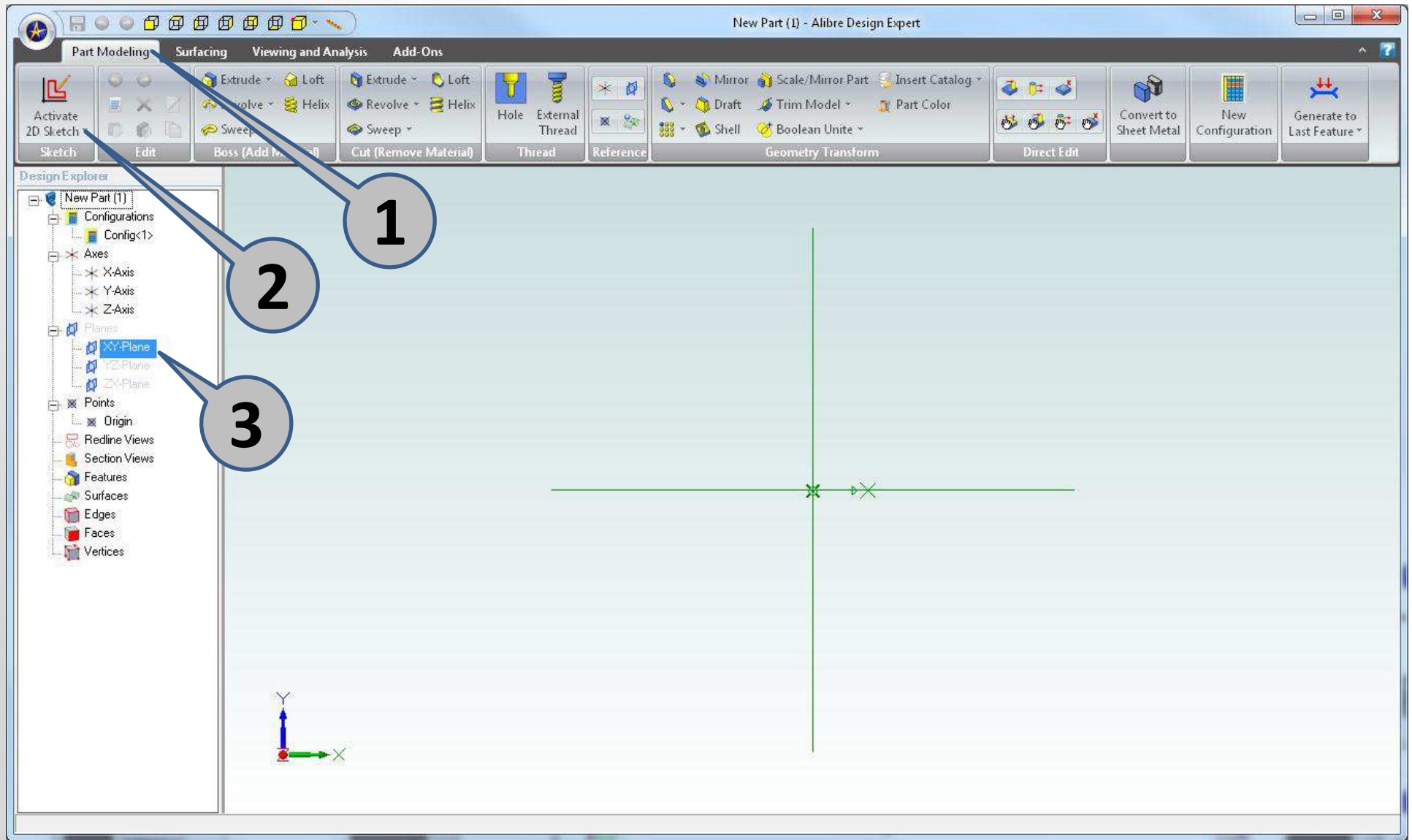
1. Confirm similar results for the color change.
2. Save this Part to a convenient location on your hard drive with the name: **Journal**.
(not shown)
3. Close the Part workspace (not shown).



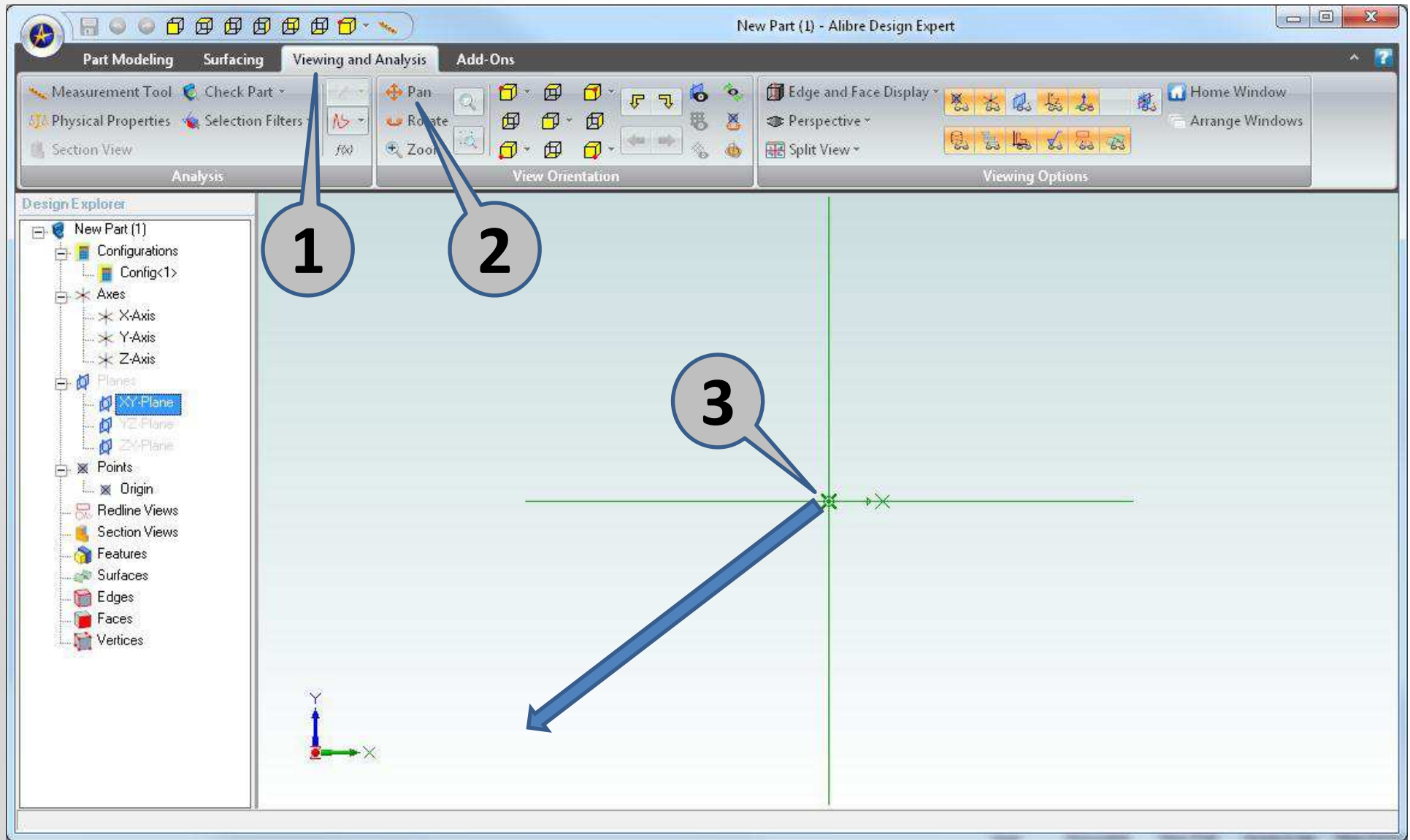
Creating the crank



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

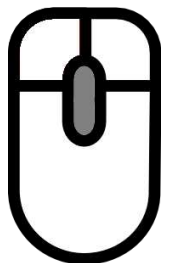


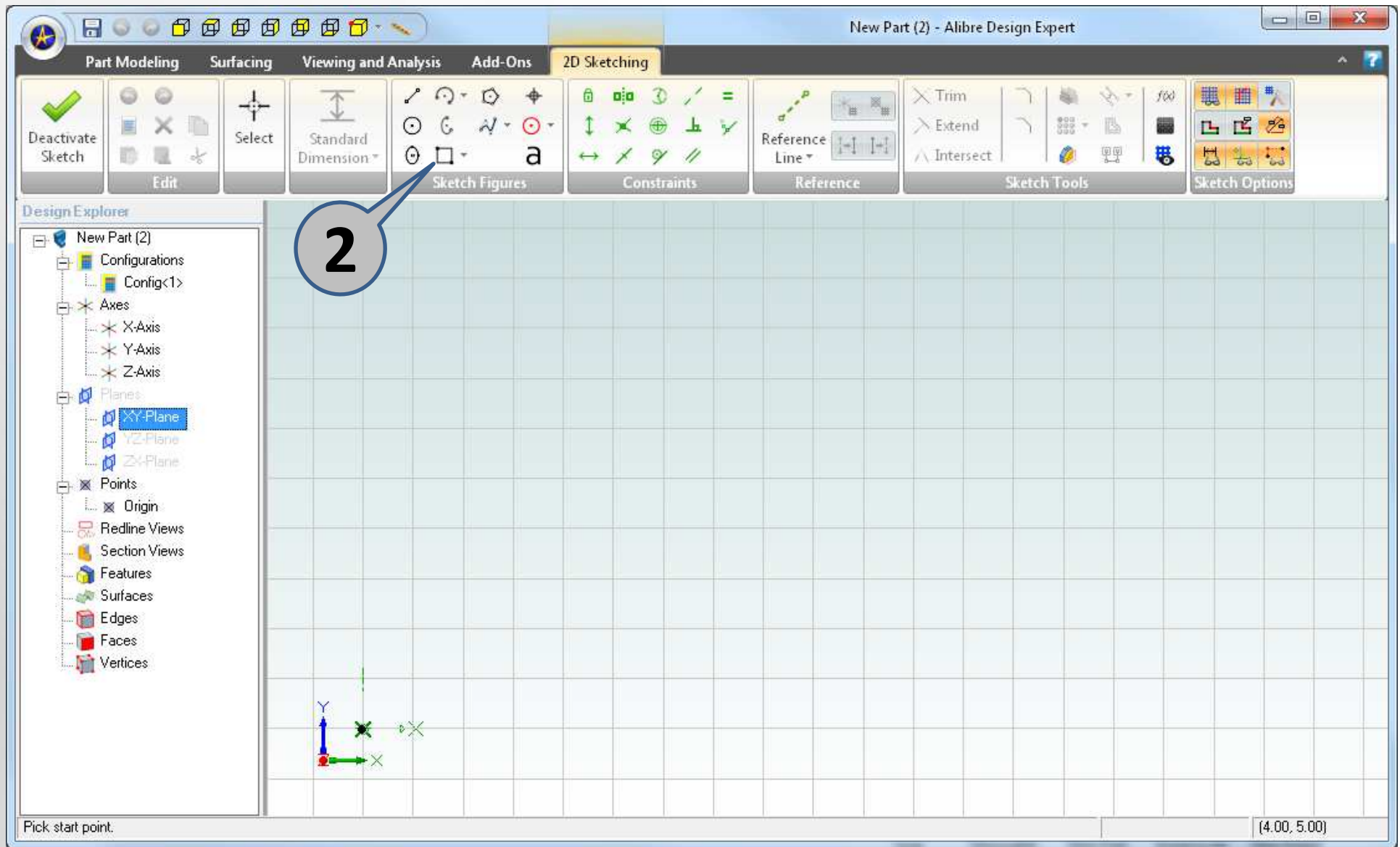
1. If necessary, click on the **Part Modeling** tab in the ribbon.
2. Click on the **XY Plane** in the **Design Explorer**.
3. Click on the **Activate 2D Sketch** icon.



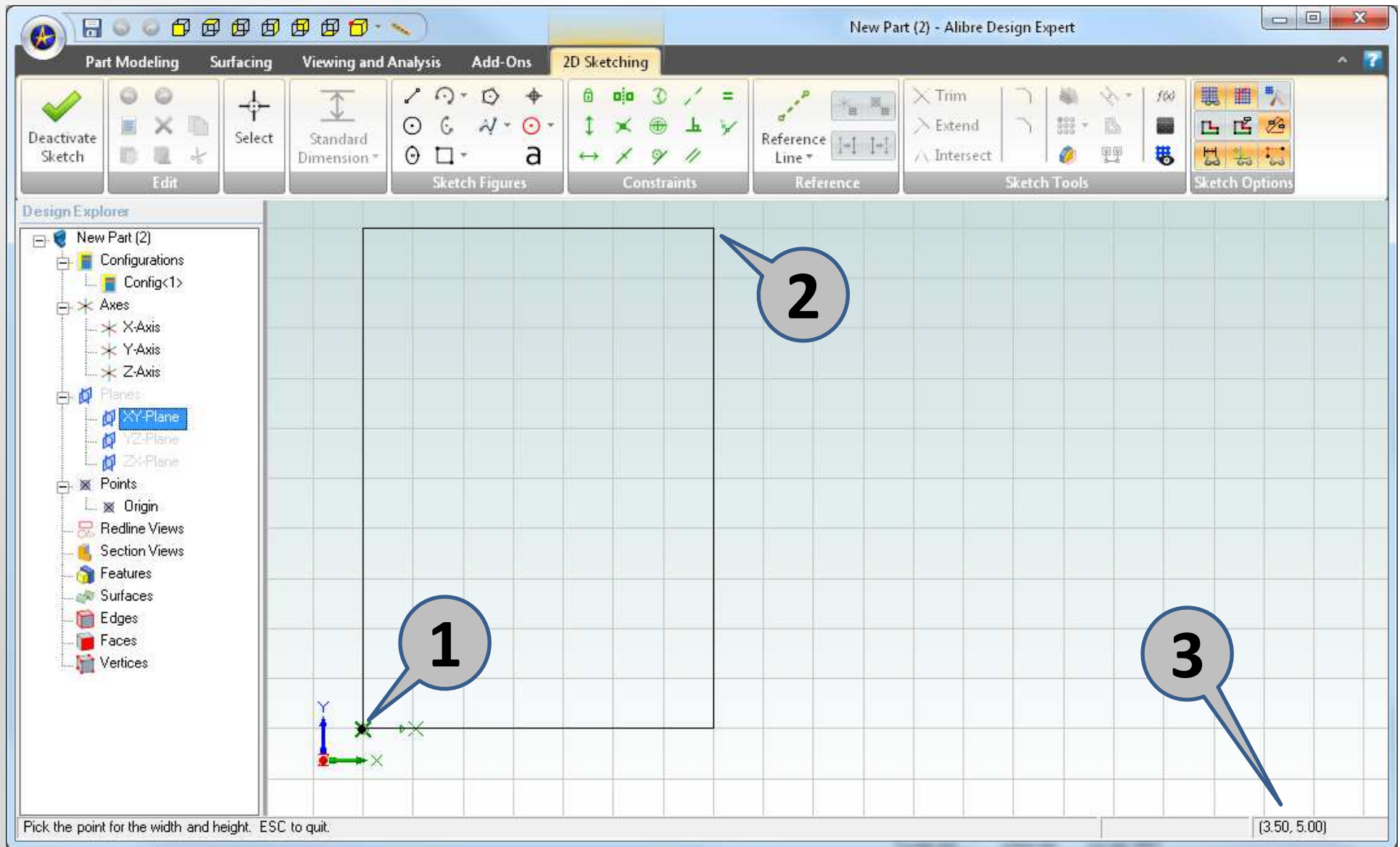
1. Click on the **Viewing and Analysis** tab in the ribbon.
2. Click on the **Pan** icon.
3. Position the mouse cursor near the **Origin** point. *Click and hold*; then drag the view downwards and to the left. Release the mouse button.

*You can also pan the model by holding down the middle mouse button and dragging.

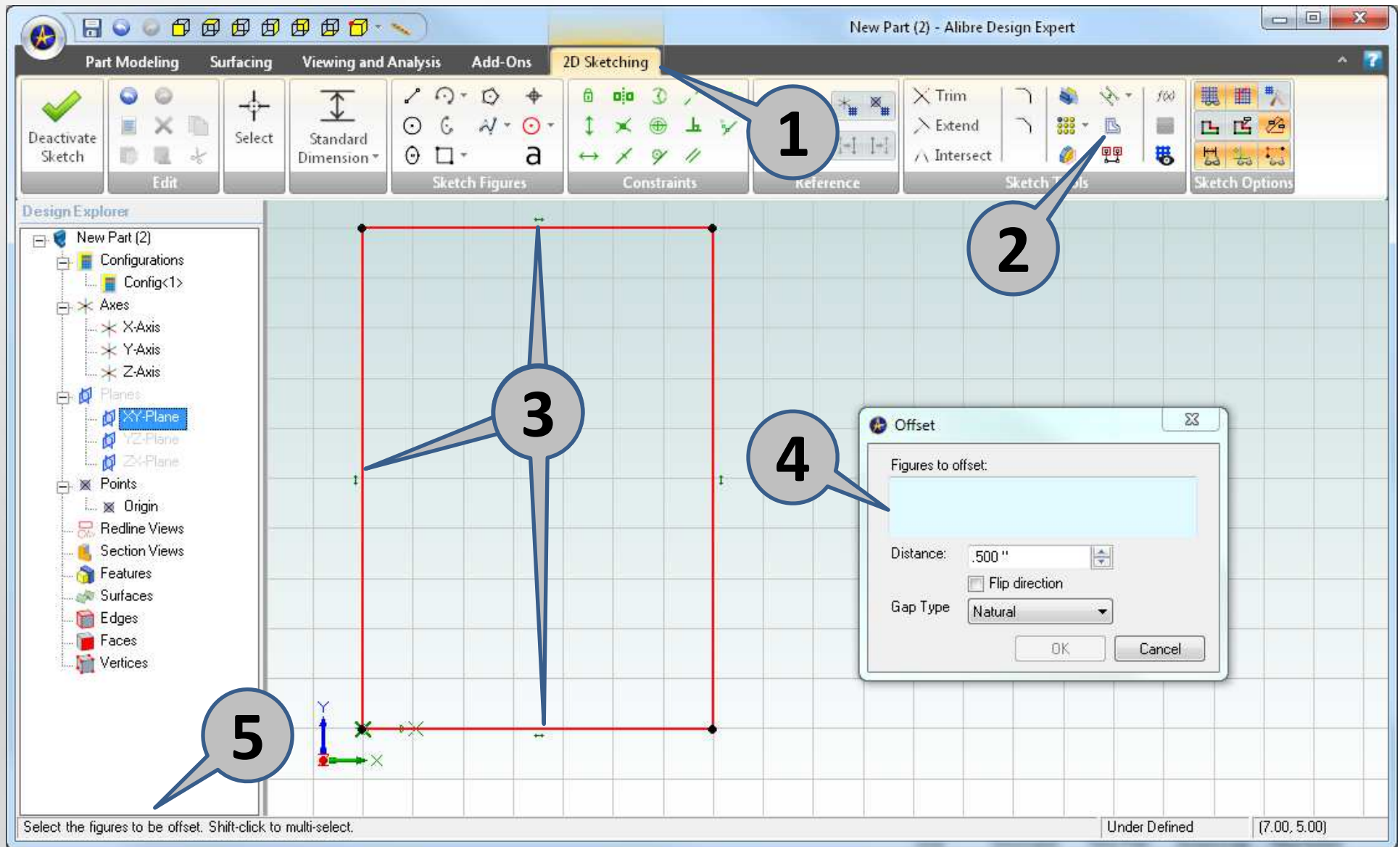




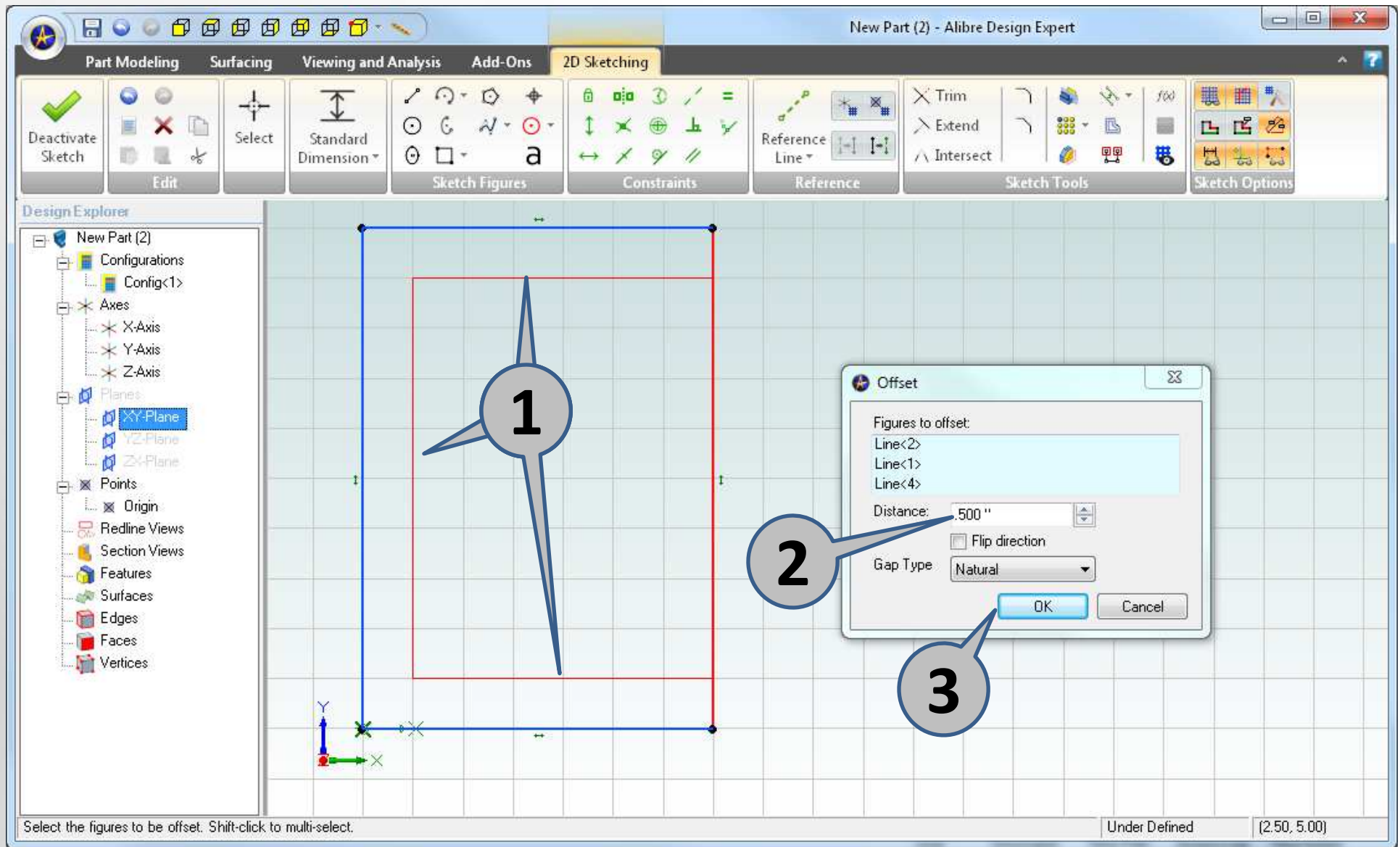
1. Click on the **Rectangle by Two Corners** tool in the **Sketch Figures** group in the **2D Sketching** tab.



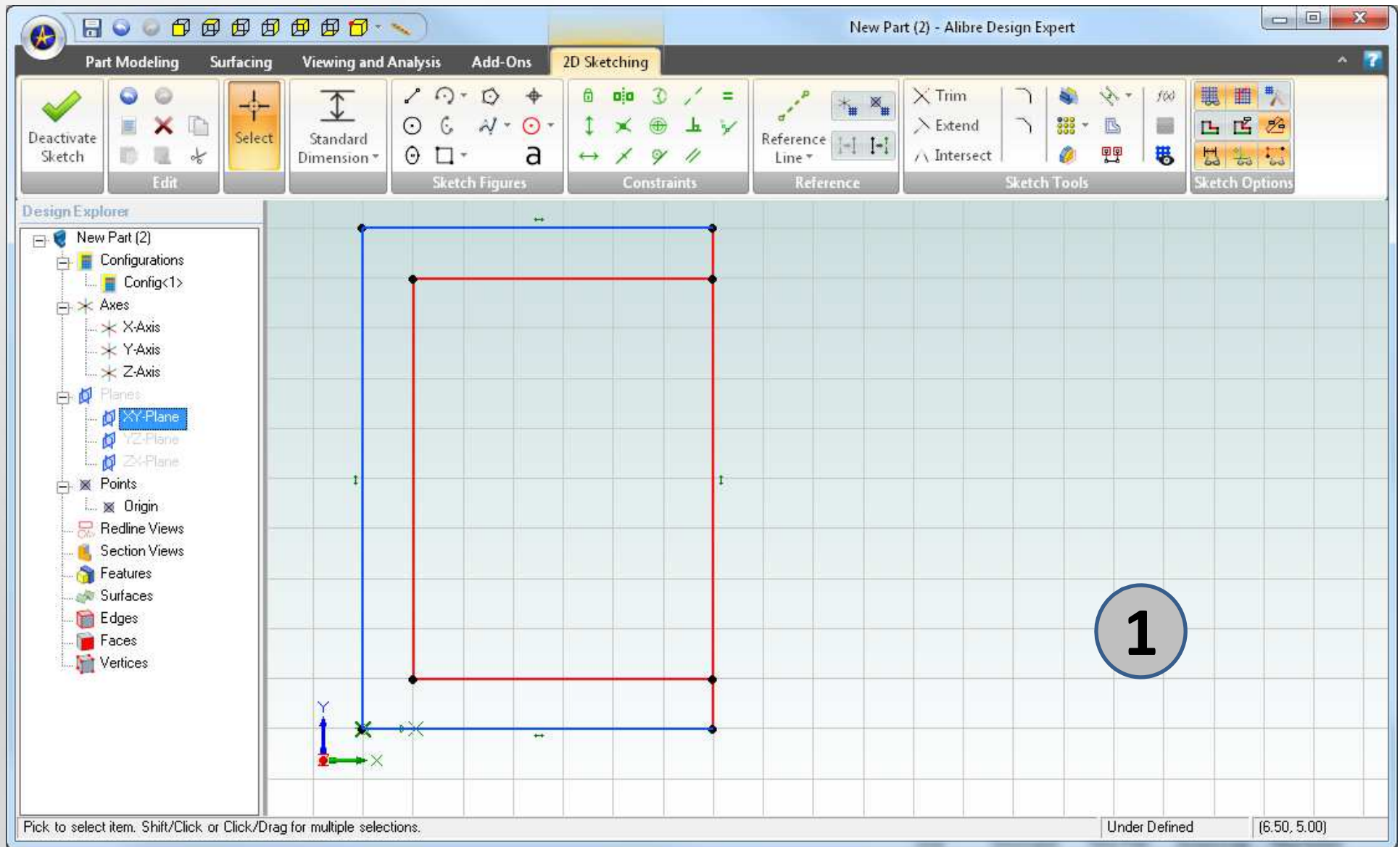
1. Position the mouse cursor over the **Origin** point. Click and release to create the first corner of the rectangle.
2. Move the mouse cursor upwards and to the right
3. The measurement indicator at the bottom of the screen should be **Width: 3.500** and **Height: 5.000**. *Click and release* to complete the rectangle.



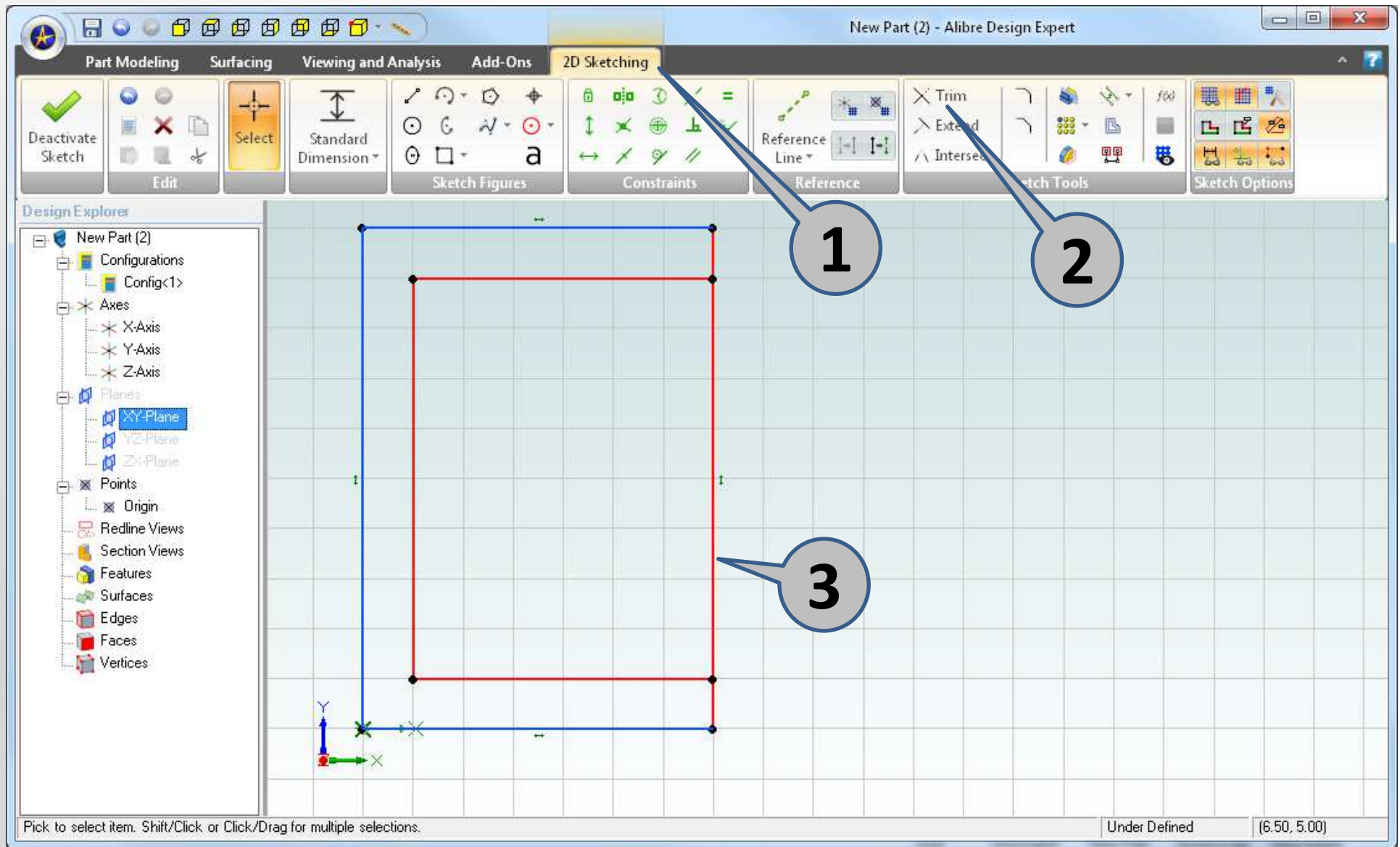
1. Make sure you are in in the **2D Sketching** tab.
2. Select the **Offset** command.
3. Click to select the top, left and bottom segments of the rectangle. The segments will change color upon selection.
4. Note that as you select them, the segments will appear in the **Figures to Offset** box (not shown).
5. Alibre Design will often tell you what it expects.



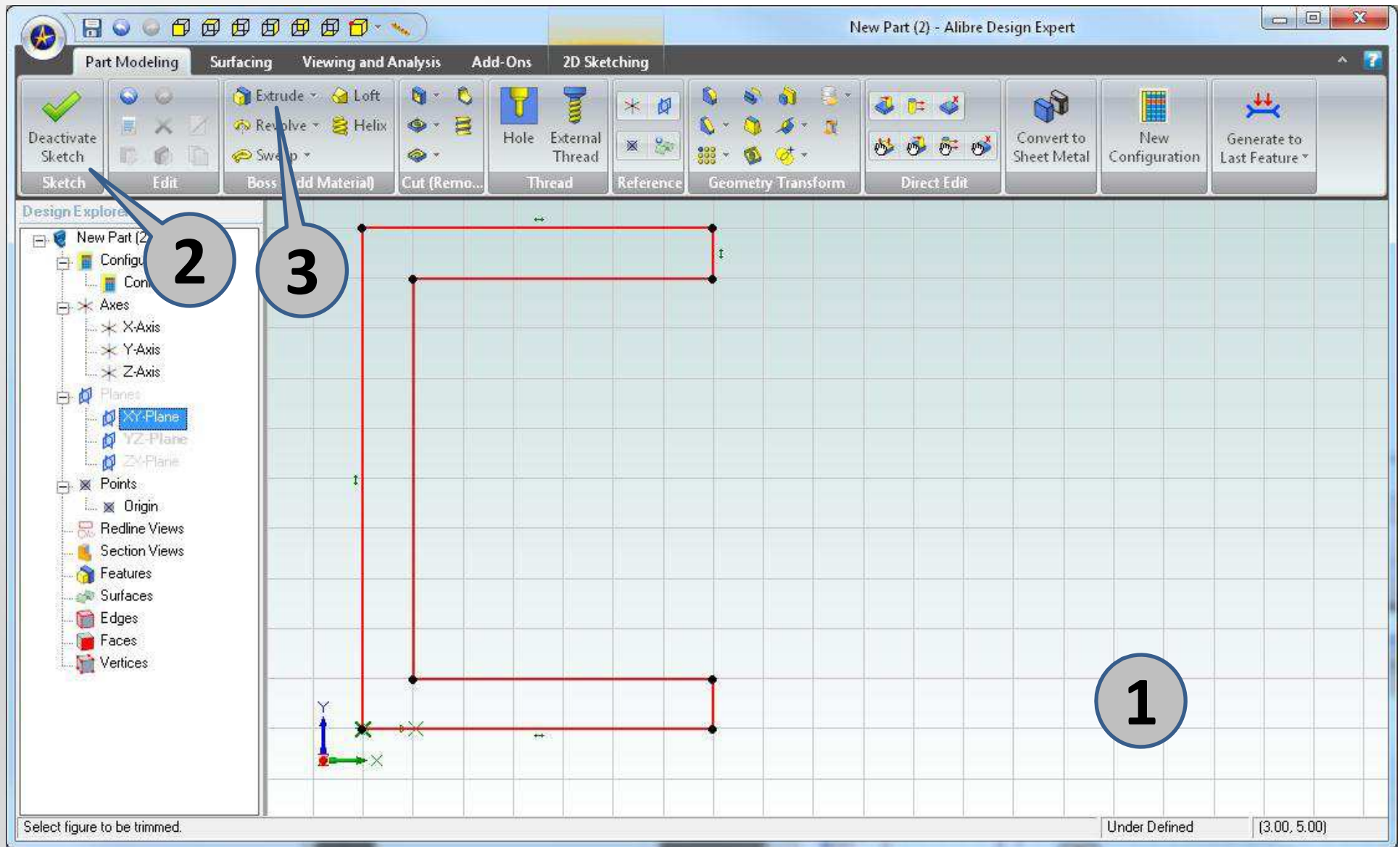
1. Confirm the preview of the **Offset** results.
2. Confirm the **Distance** value in the **Offset** dialog box is **0.5 inches**.
3. Click **OK**.



1. Confirm the result of the **Offset** operation. 3 of the 4 figures of the rectangle should be offset by .5".

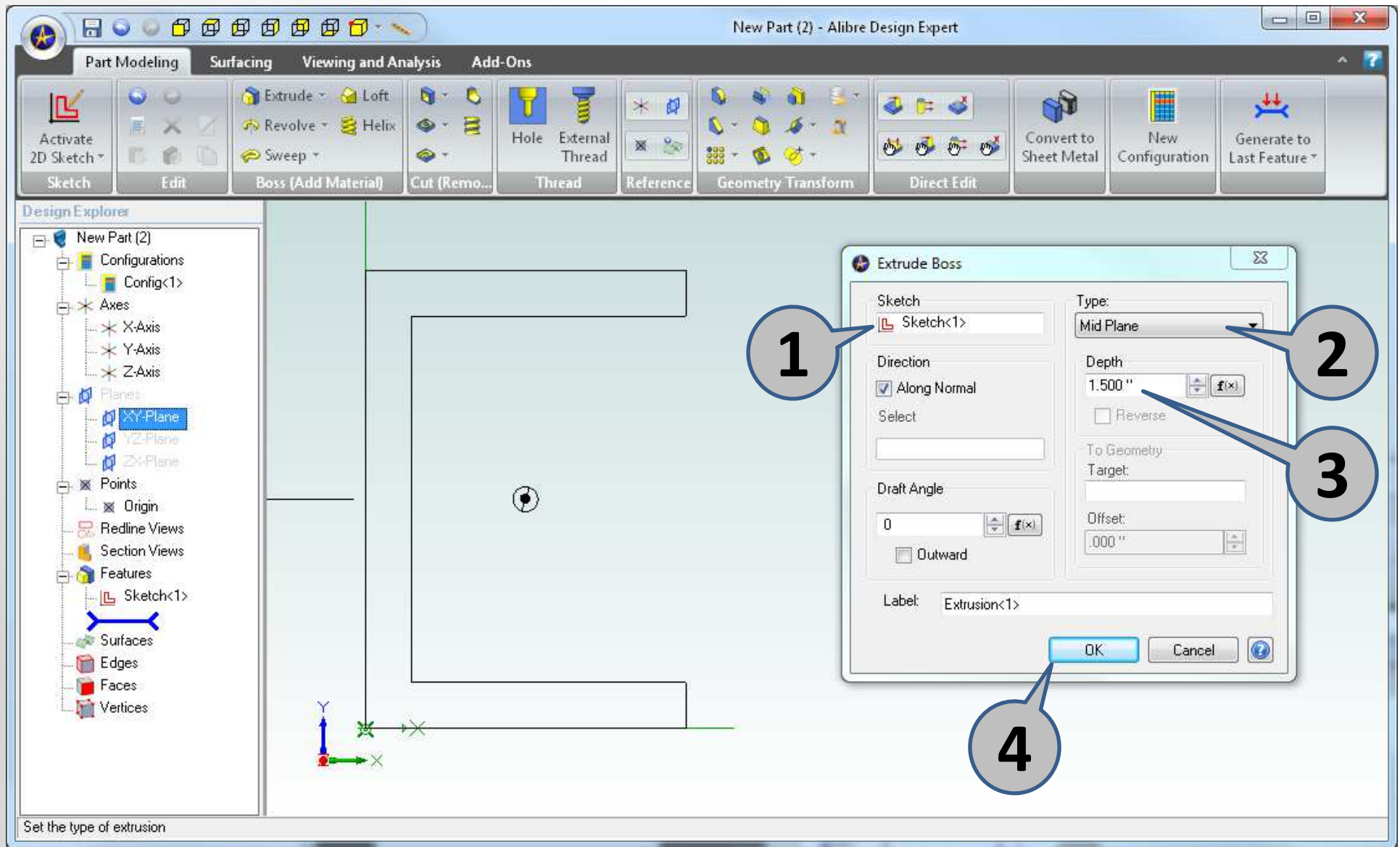


1. Make sure you have the **2D Sketching** tab active.
2. Select the **Trim** command.
3. Click on the line segment indicated. The segment will get deleted.

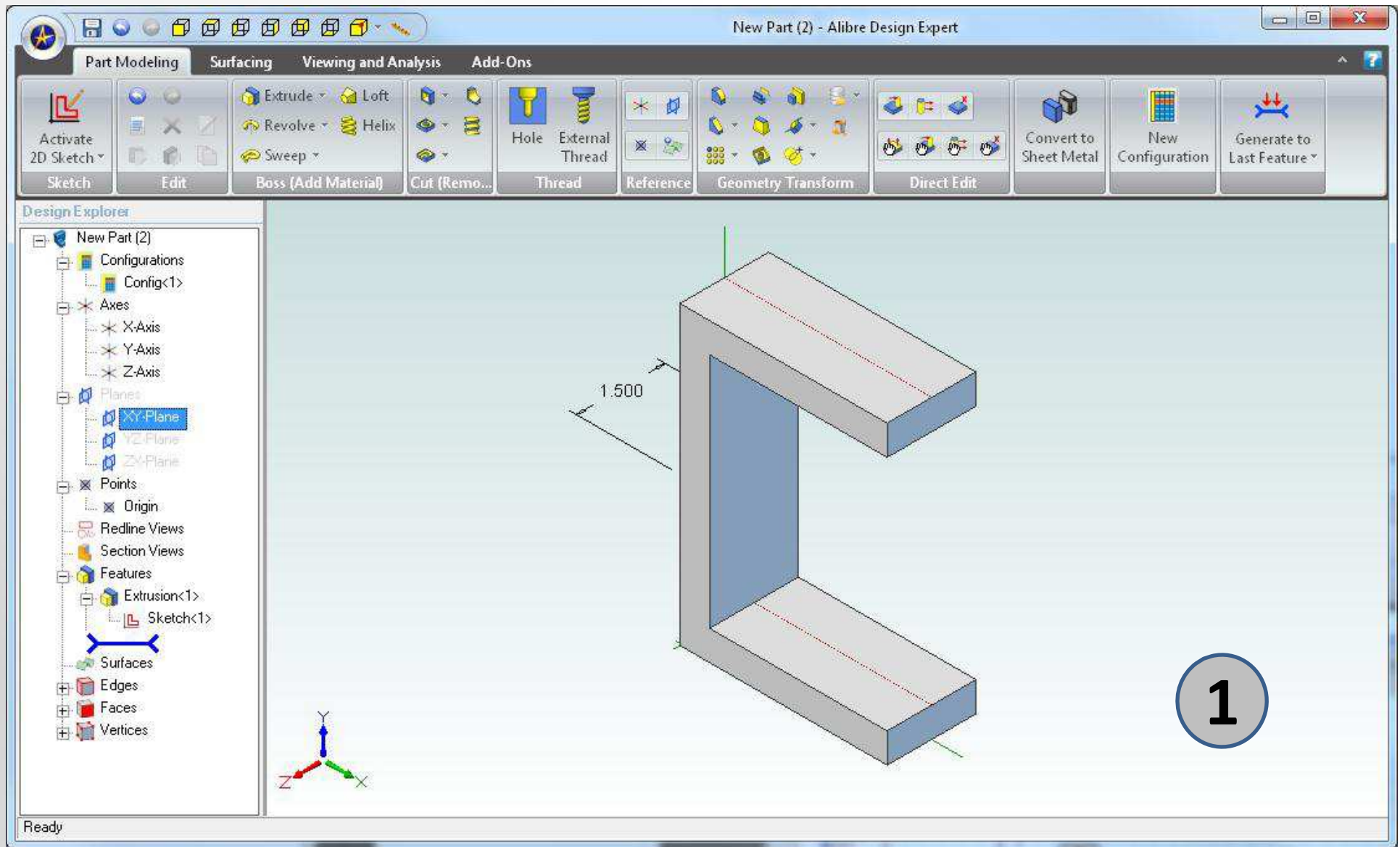


1. Confirm the results of the **Trim** operation.
2. Click on the **Deactivate Sketch** icon in the ribbon.
3. Click the **Extrude Boss** icon.

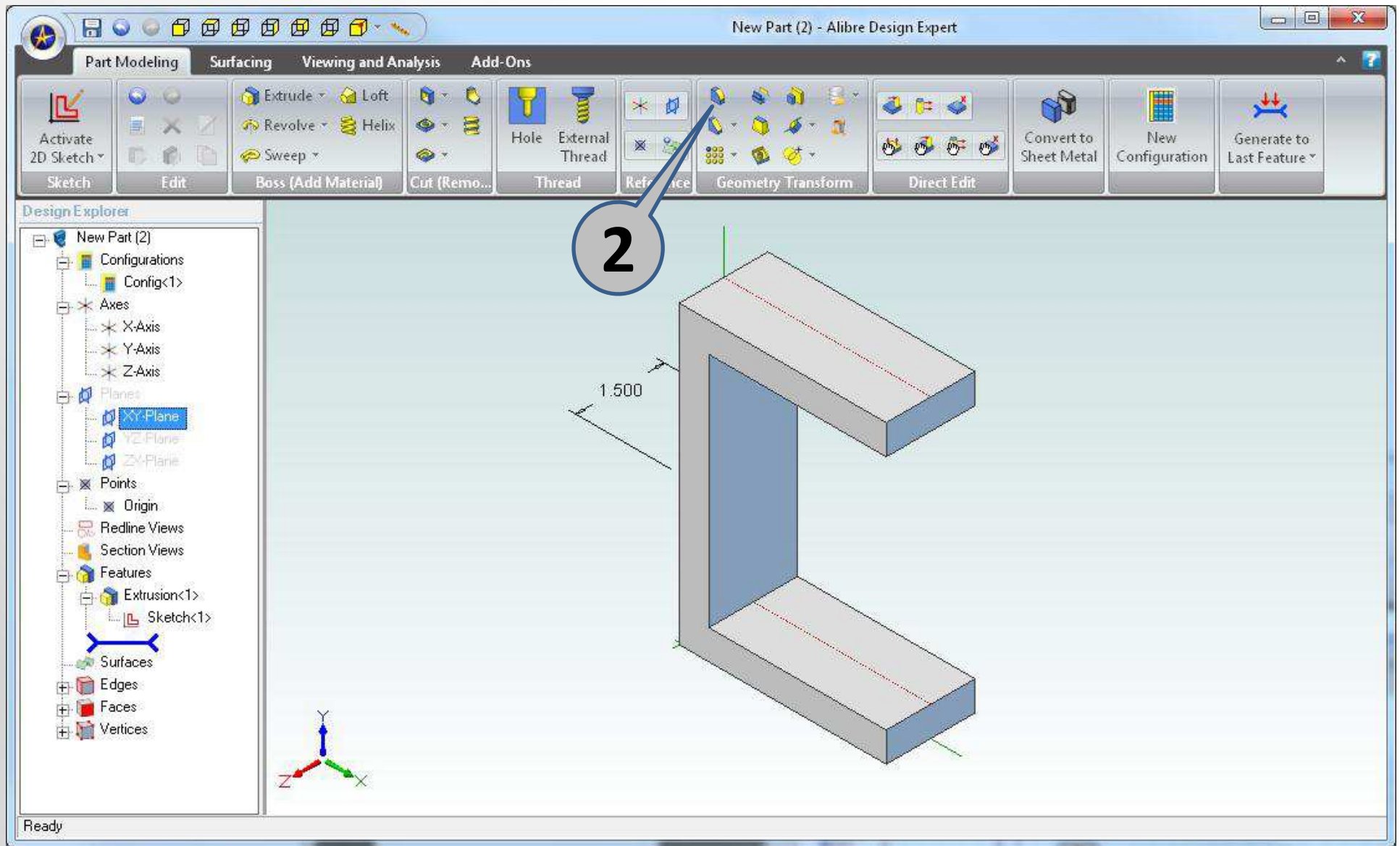
When you click on the **Deactivate Sketch** icon in the **2D Sketching** tab you will automatically be brought to the **2D Drawing** tab.



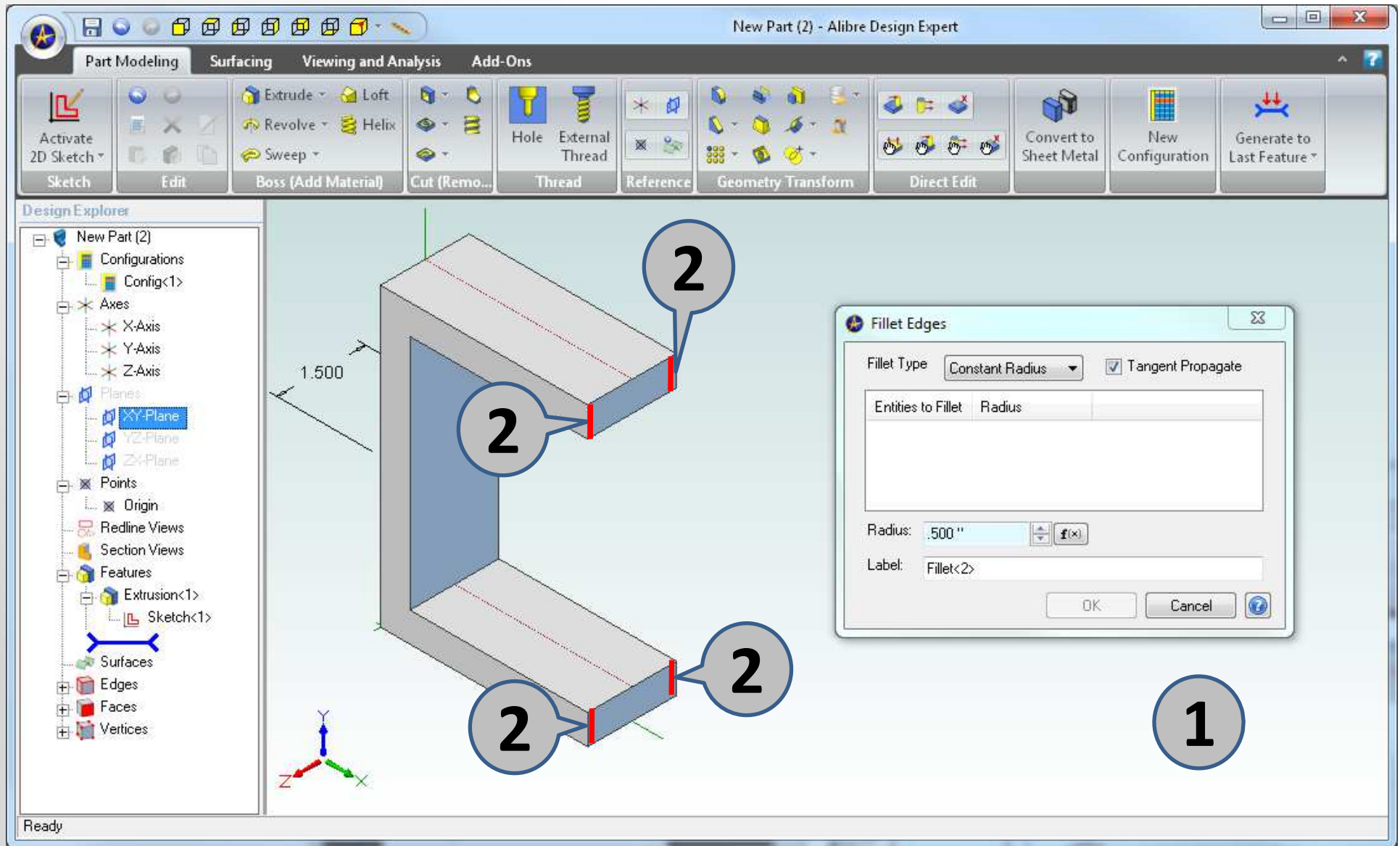
1. Confirm that **Sketch<1>** is the **Sketch** to be extruded.
2. Select **Mid Plane** as the **Type** of extrusion.
3. Enter a value of **1.5 inches** for the **Depth**.
4. Click **OK**.



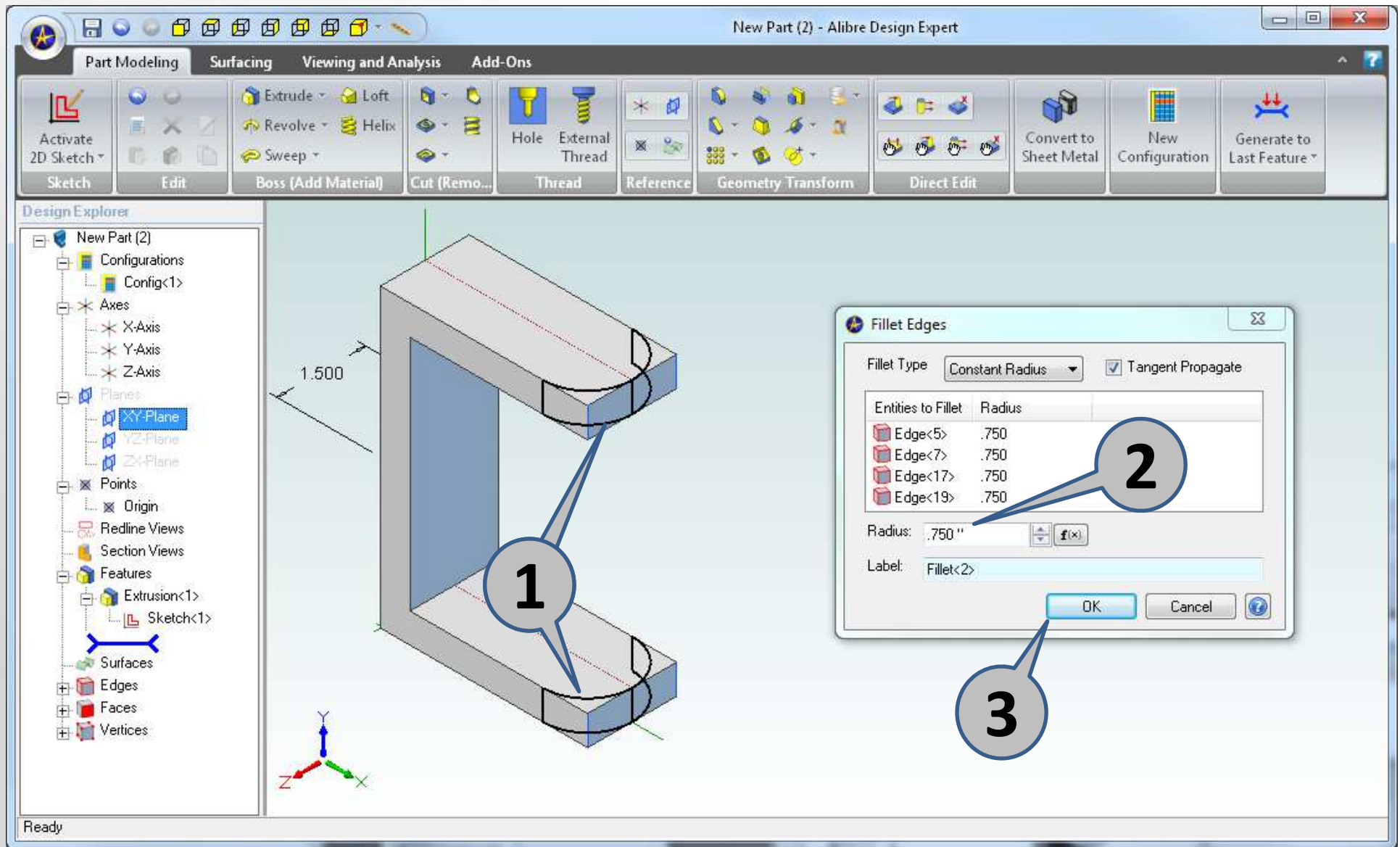
1. Confirm a similar result for the **Extrude Boss** operation.



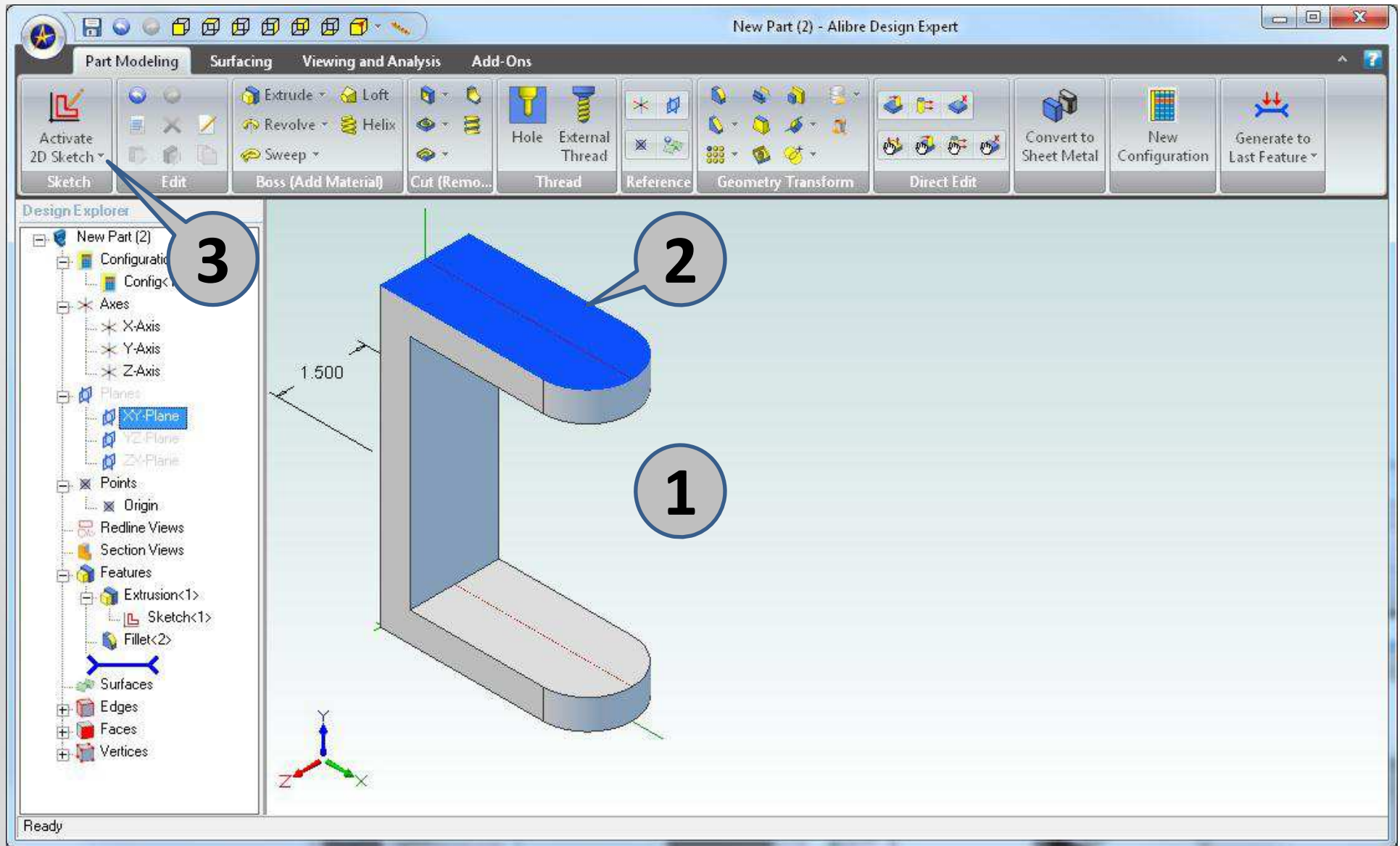
1. Select the **Fillet** icon.



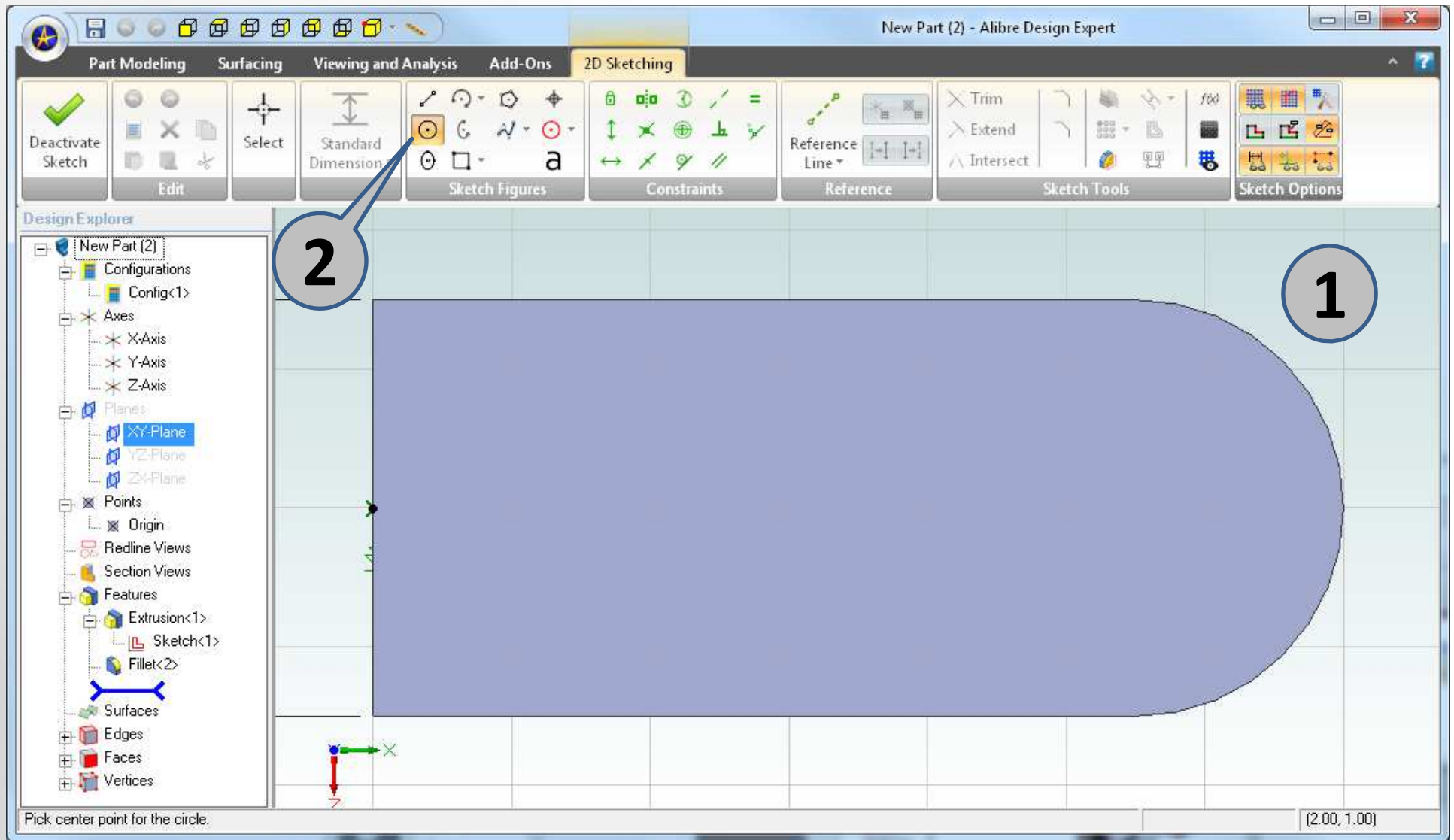
1. Move the **Fillet Edges** dialog box such that the entire part is visible.
2. Click on the four edges indicated.



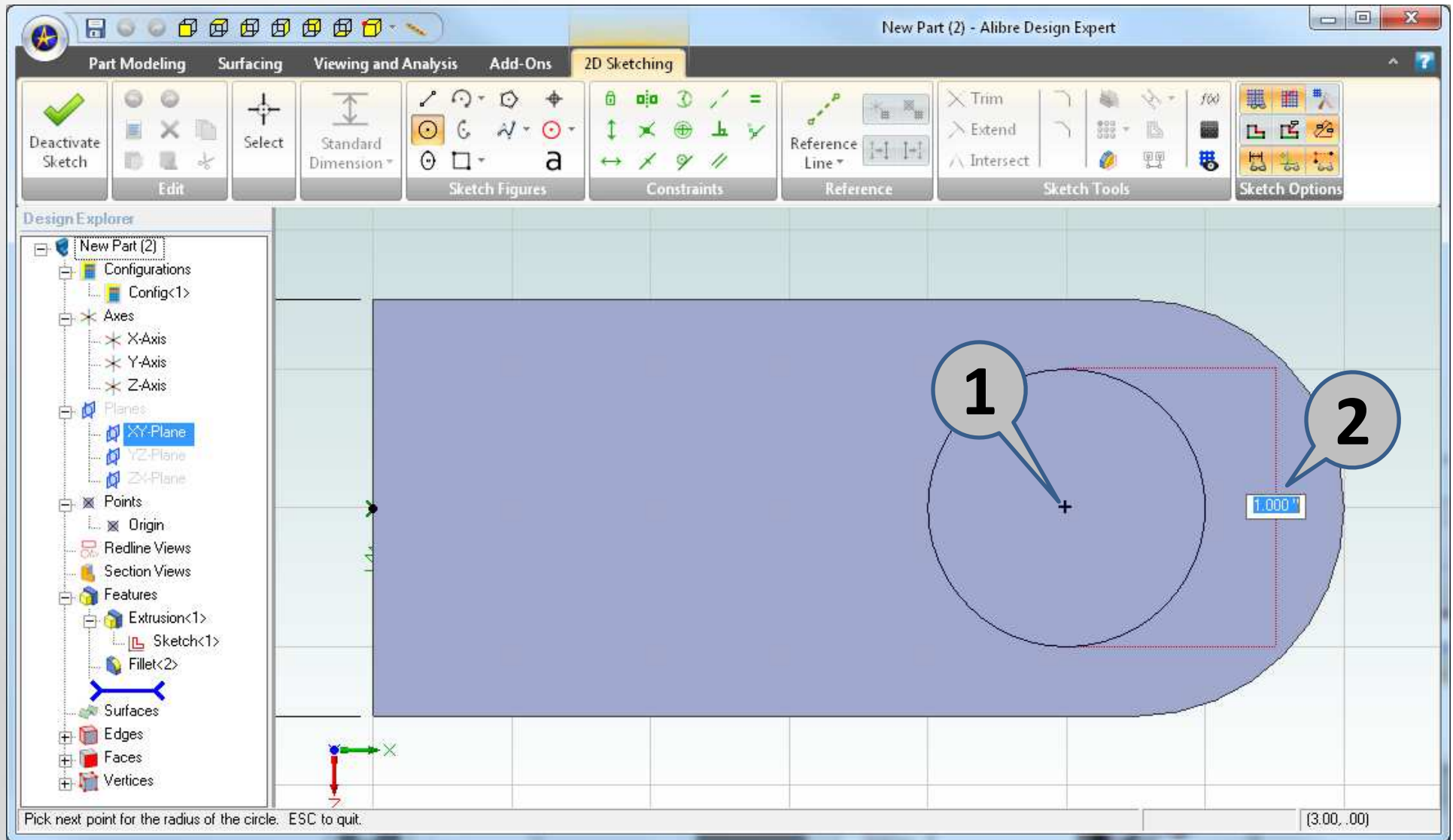
1. A preview of the **Fillet** operation is displayed.
2. Enter a value of **0.75 inch** for the **Radius** in the **Fillet Edges** dialog box.
3. Click **OK**.



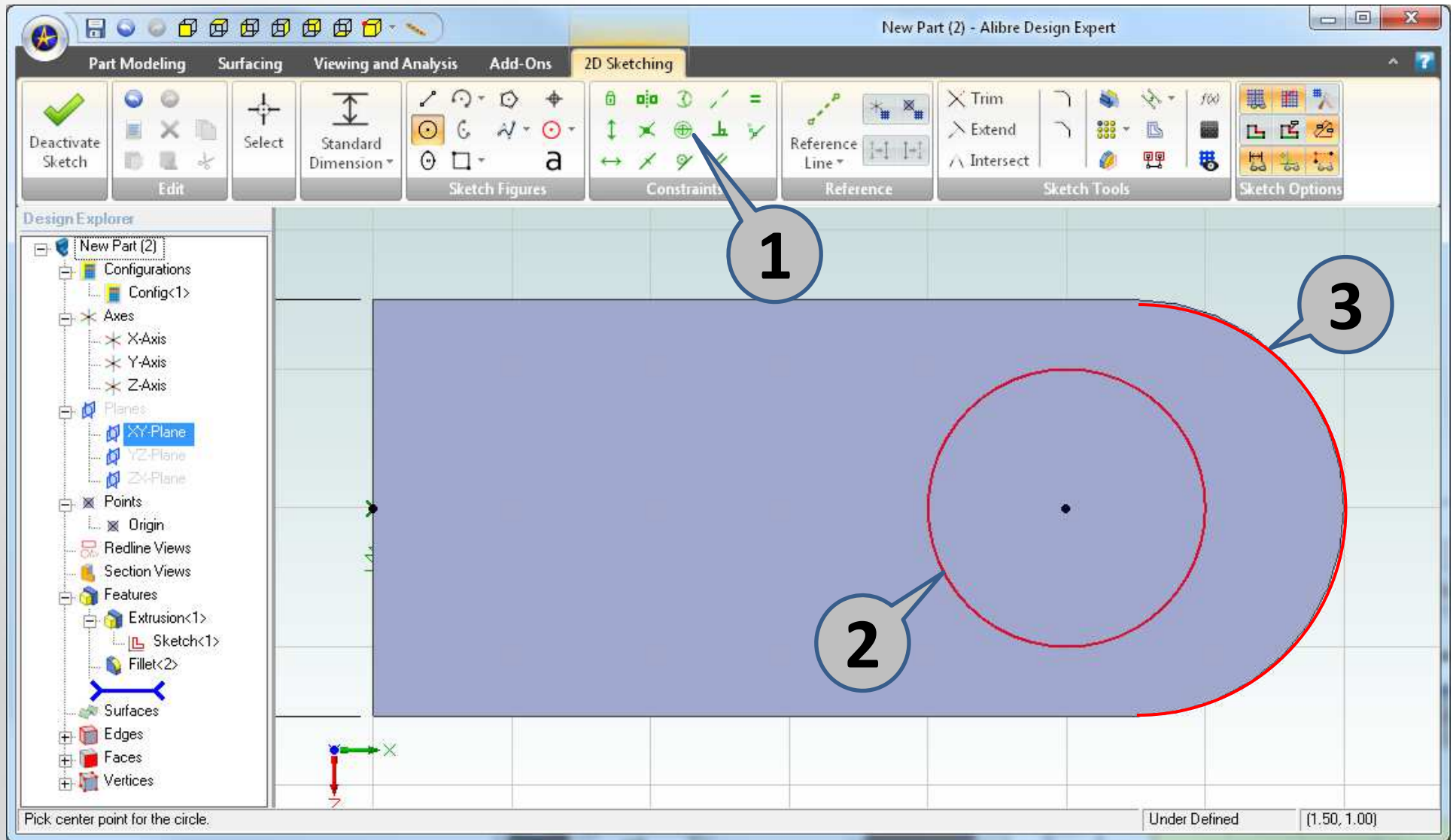
1. Confirm similar results for the **Fillet** operation.
2. Click on the face indicated.
3. Click **Activate 2D Sketch**.



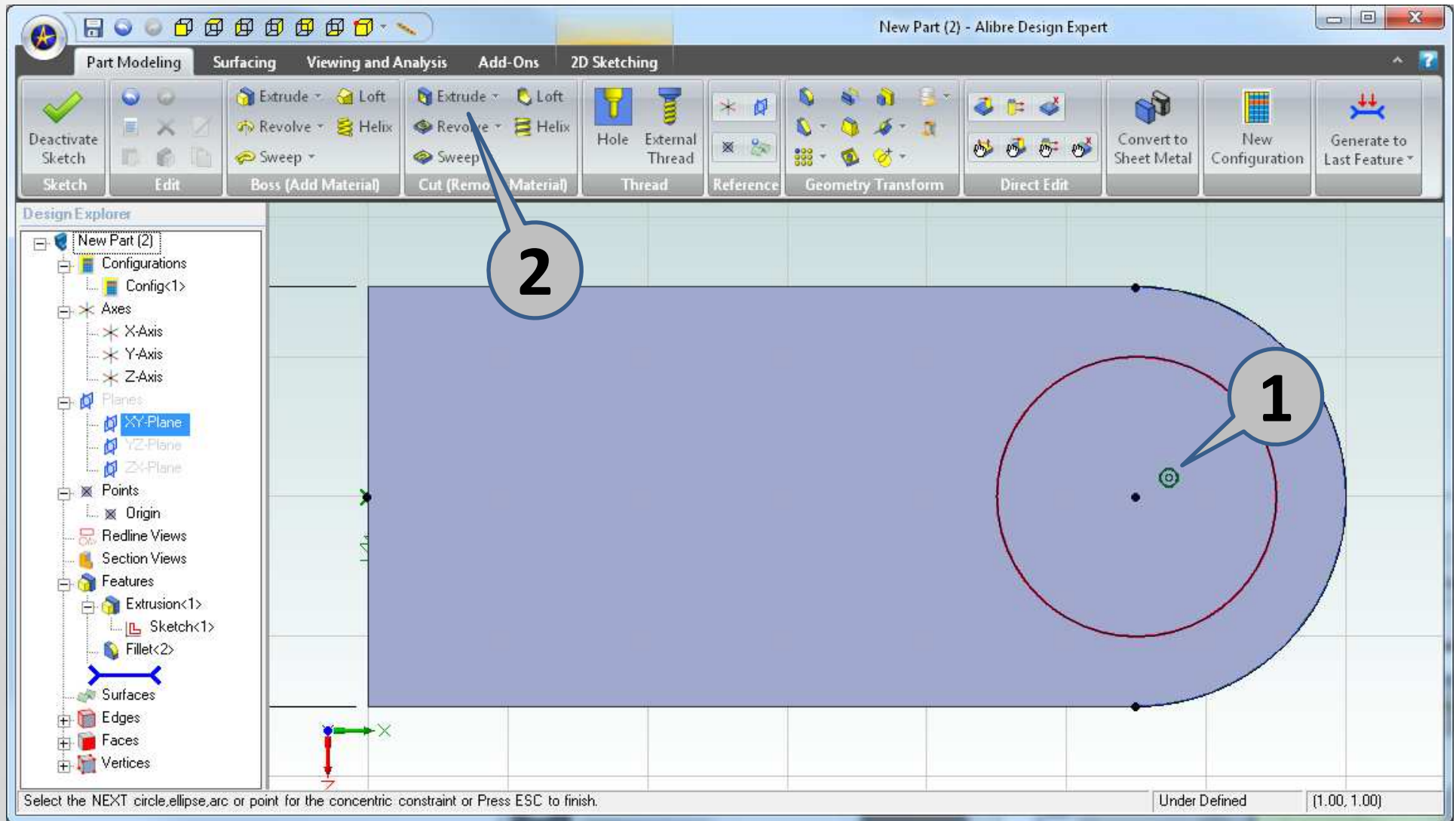
1. Confirm that the view of the work area is similar to the screen capture.
2. Click on the **Circle** tool.



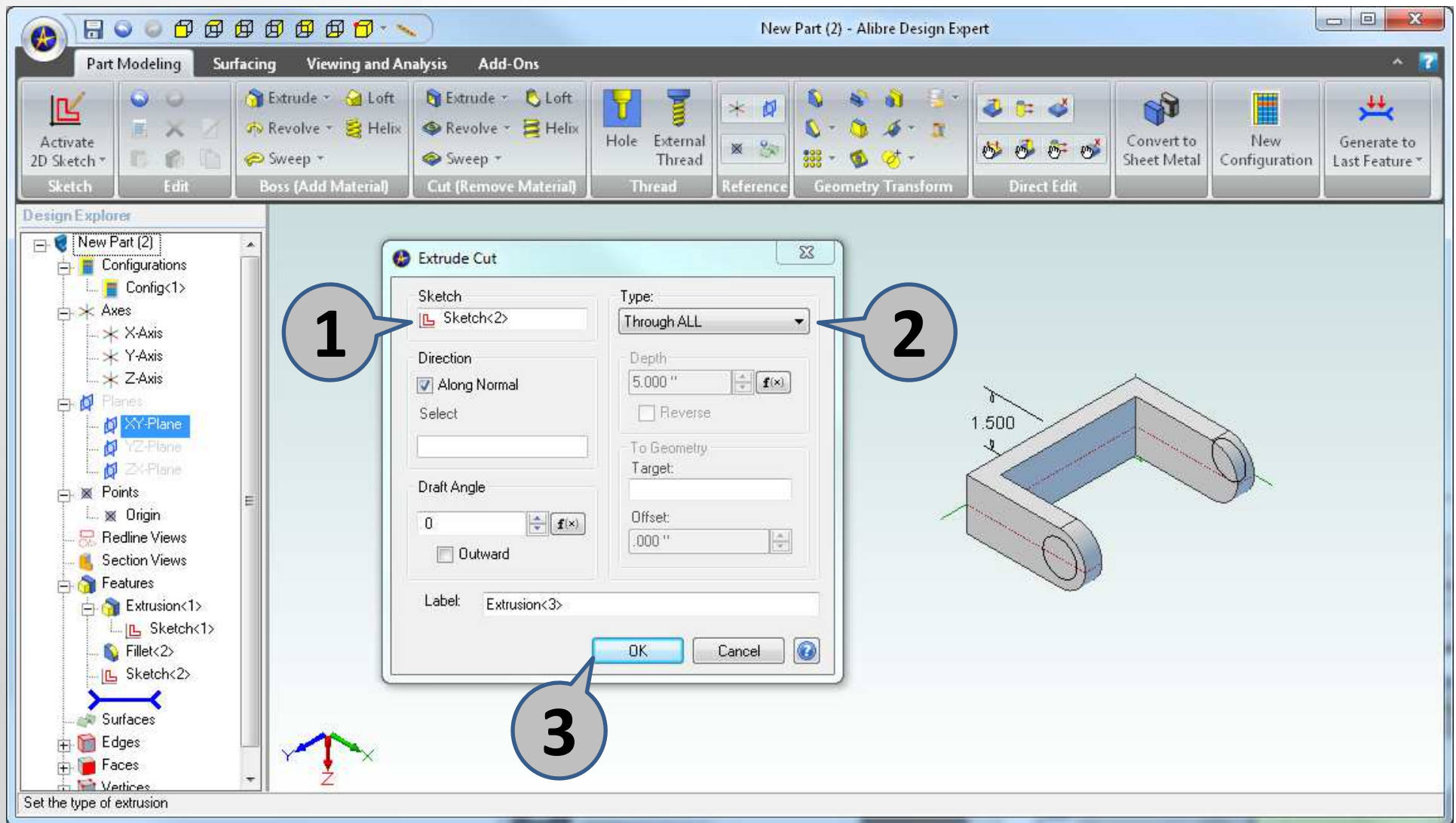
1. Move the mouse cursor to the position shown (2.50" to the right of the origin). Click and release to create the center point of the circle.
2. Move the mouse cursor to the right a distance of one grid square; this will create a $\varnothing 1.000''$. Click and release to complete the circle.



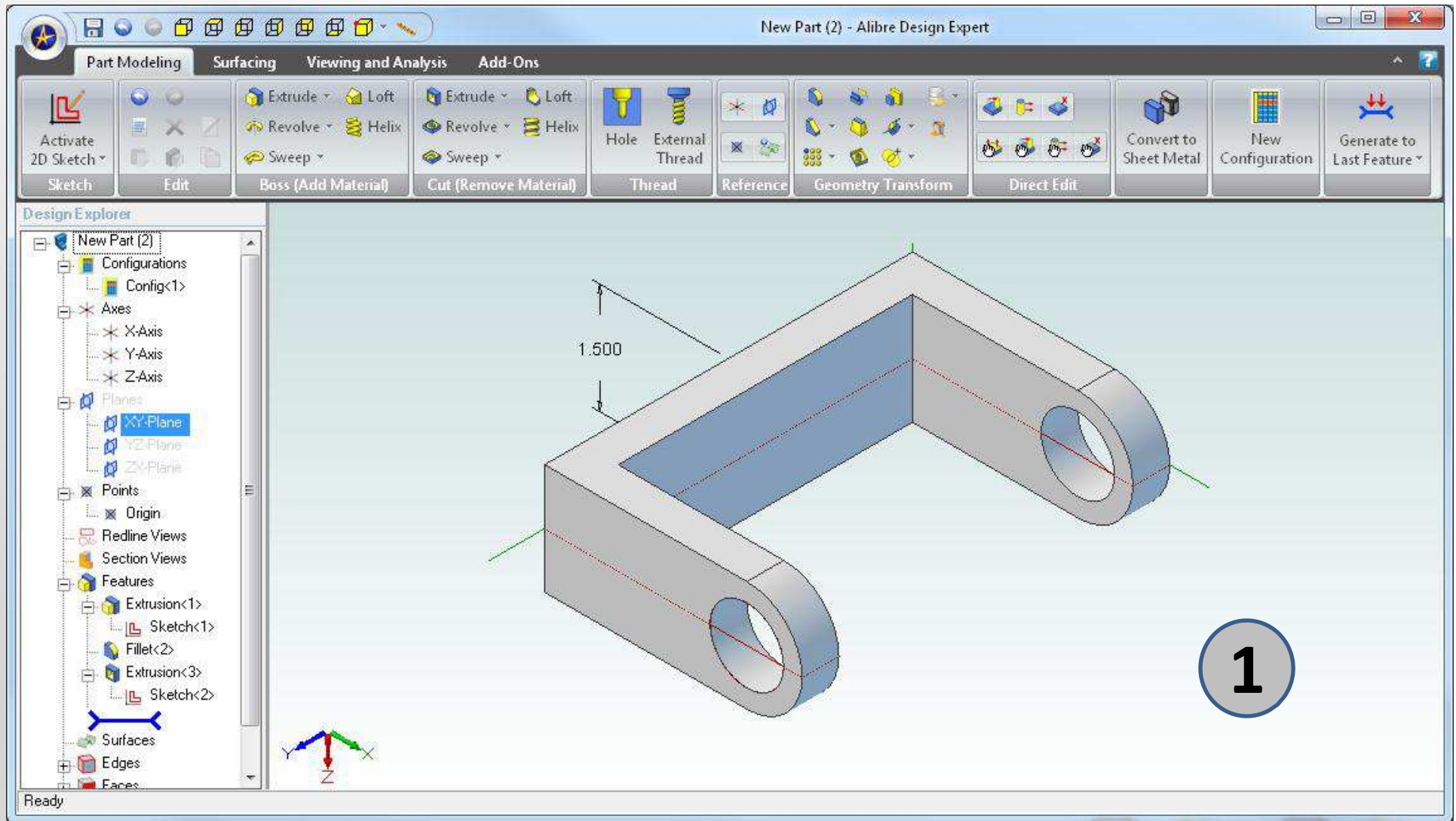
1. Select the **Concentric** option.
2. Click on the perimeter of the circle.
3. Click on the circular edge indicated.



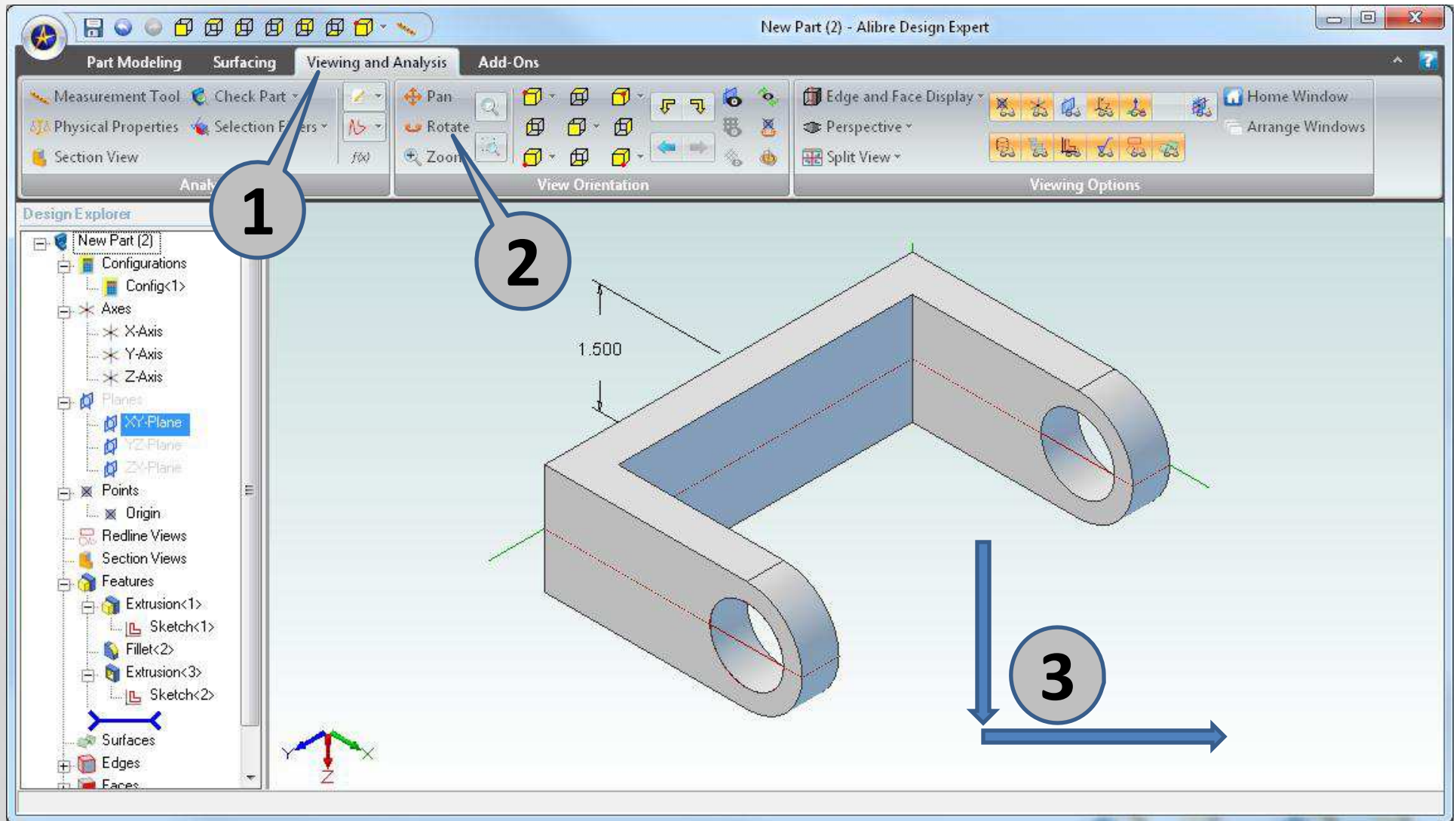
1. Confirm similar results for the **Concentric** constraint. Take note of the Concentric constraint symbol that appears near the circle's center. The circle and the circular edge should be concentric, and the circle's center point should move to the right.
2. Click on **Extrude Cut** tool in the **Cut (Remove Material)** group in the **Part Modeling** tab (or click on the **Deactivate Sketch** icon).



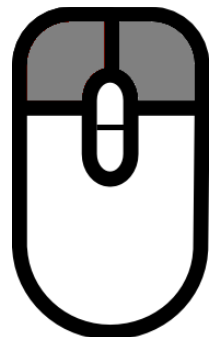
1. In the **Extrude Cut** dialog box, confirm that the **Sketch** entry box displays **Sketch<2>**.
2. Select **Through All** as the **Type** from the drop down menu.
3. Click **OK**.

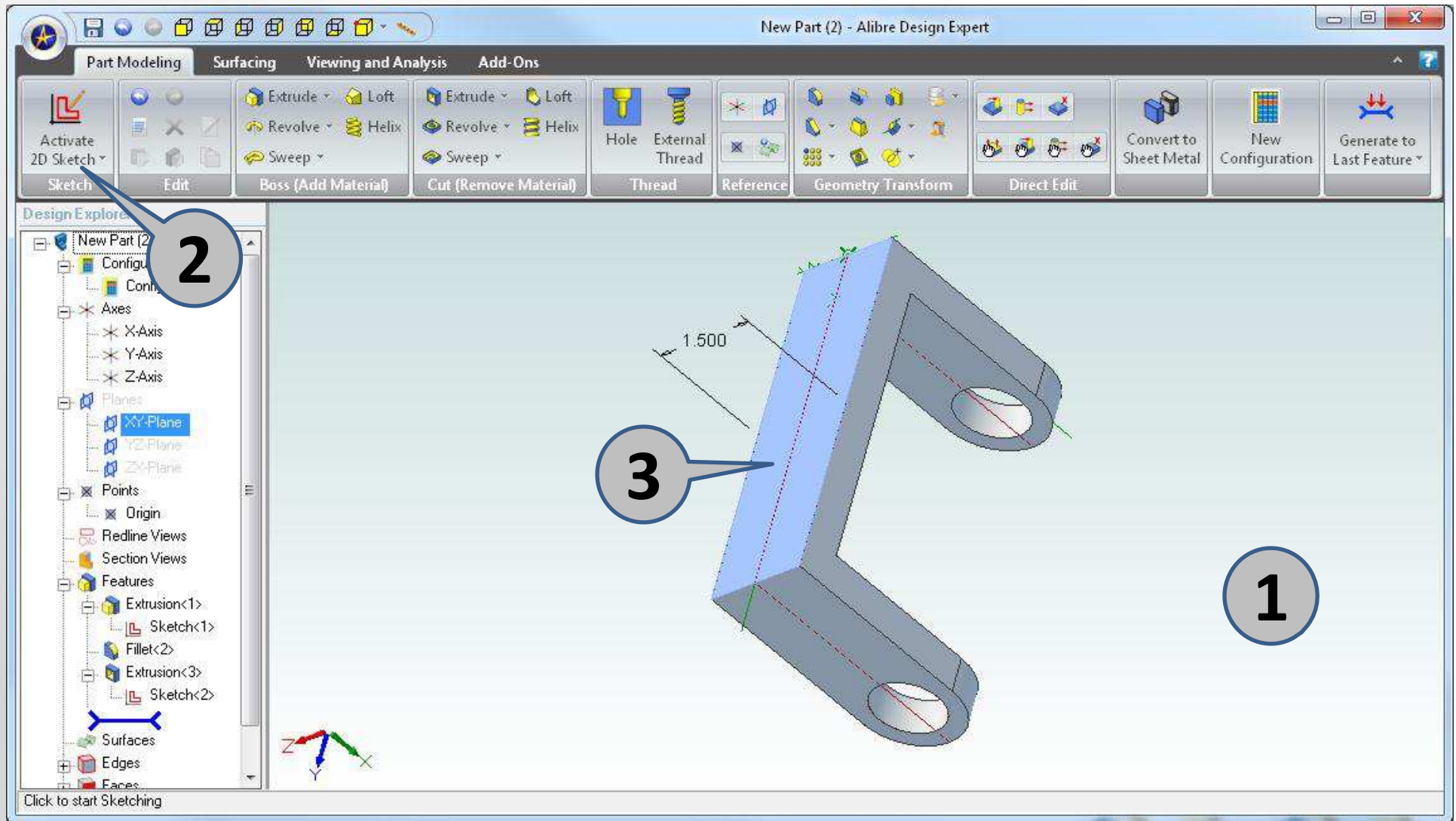


1. Confirm similar results for the **Extrude Cut** operation.

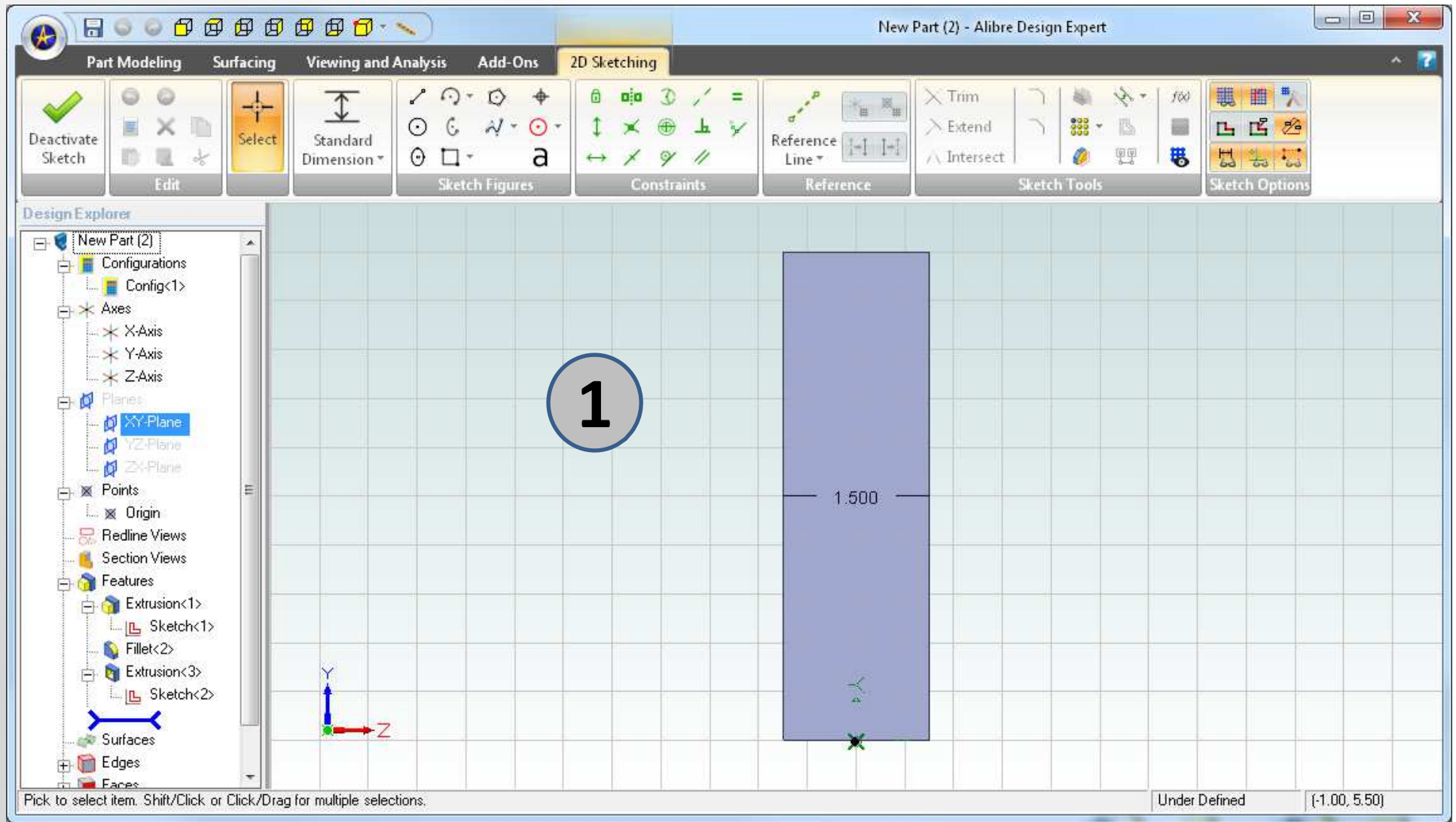


1. Click on the **Viewing and Analysis** tab in the ribbon.
2. Select the **Rotate command**. Alternately, hold down the left and right mouse buttons together over the model.
3. Position the mouse cursor as shown. Click and hold the mouse button while moving the mouse to the right (approximately as shown). Release the mouse button.

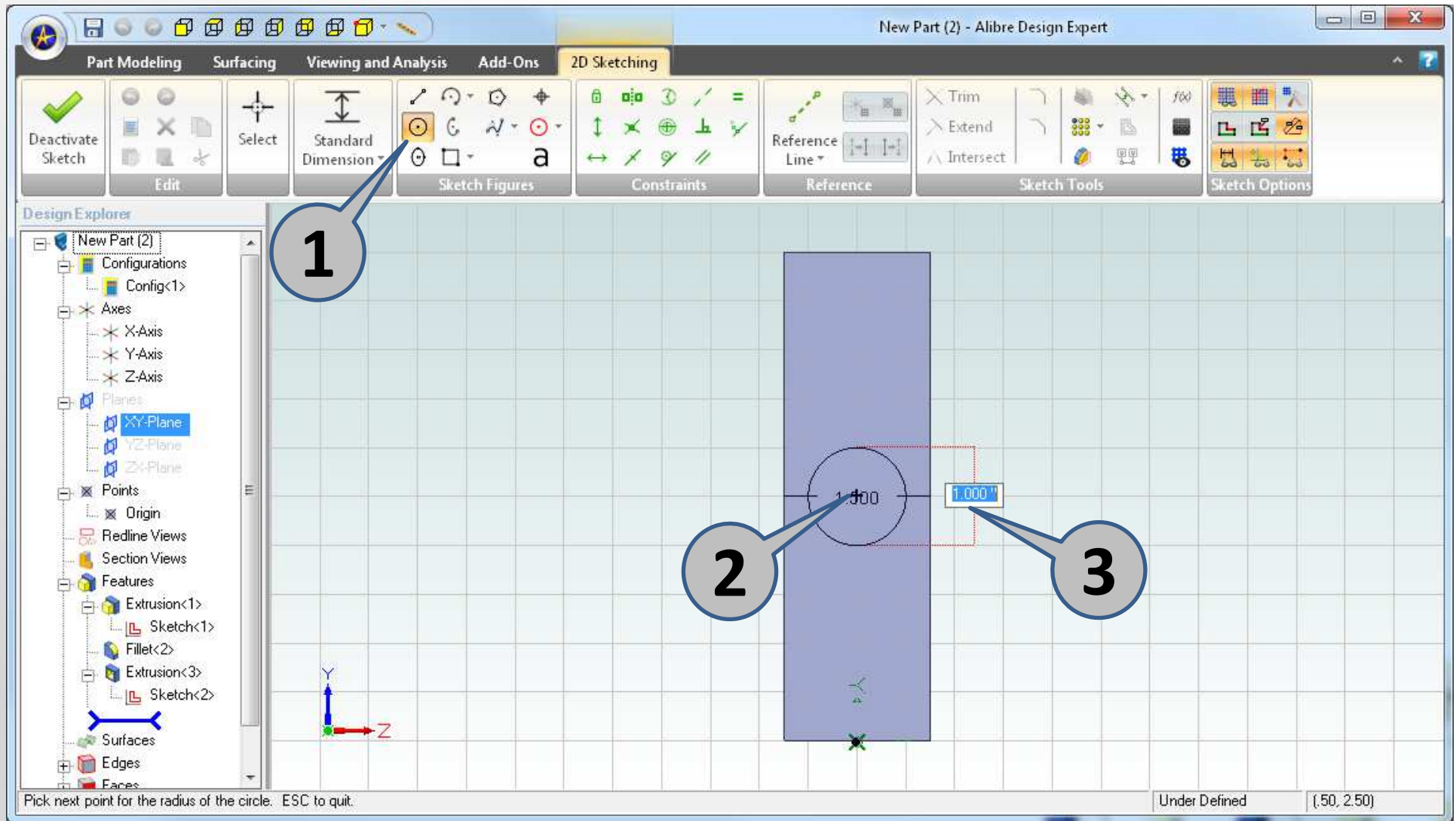




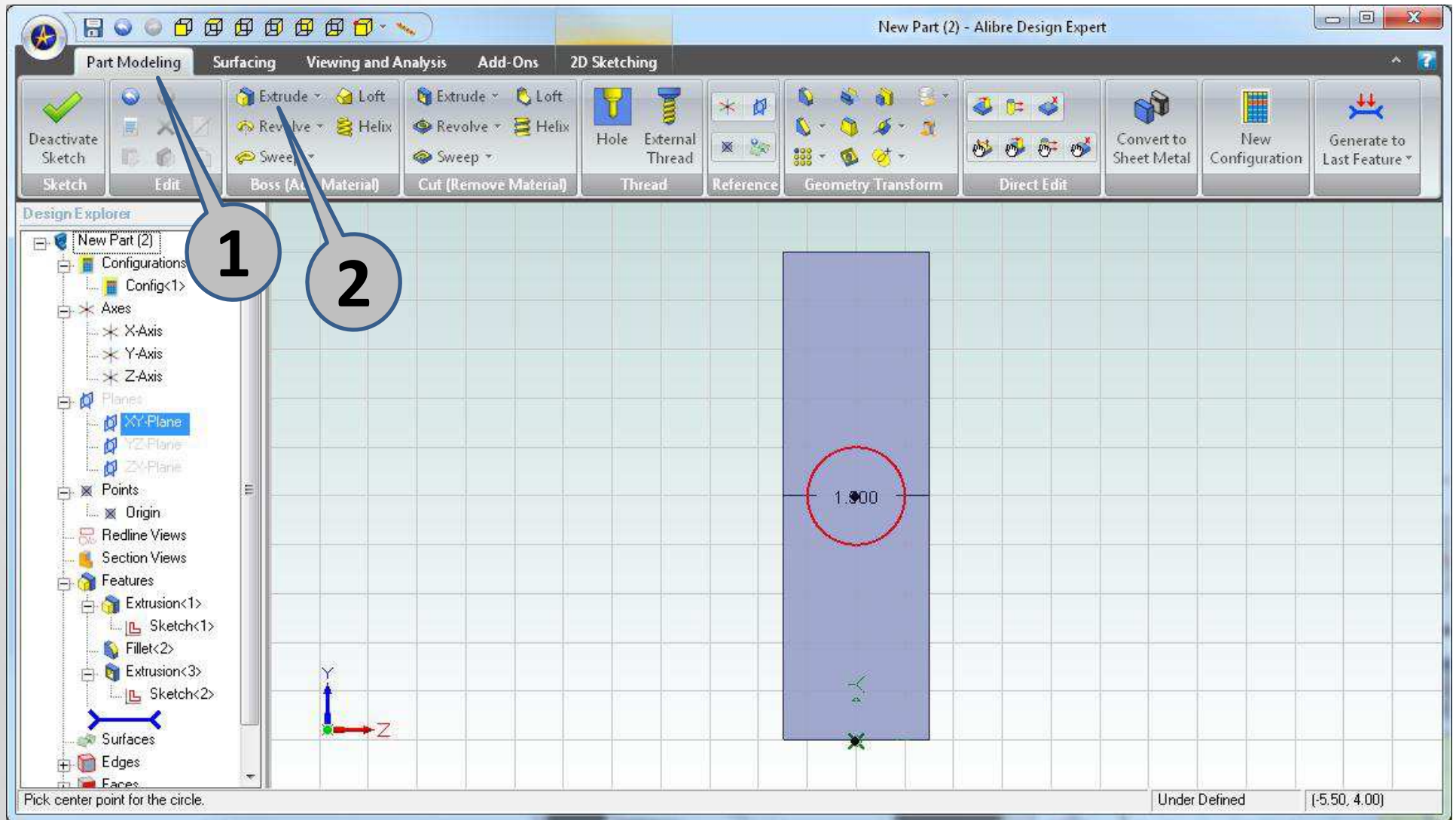
1. Confirm similar results for the view rotation.
2. Click on the **Part Modeling** tab in the ribbon and click on the **Activate 2D Sketch** icon.
3. Click on the face indicated.



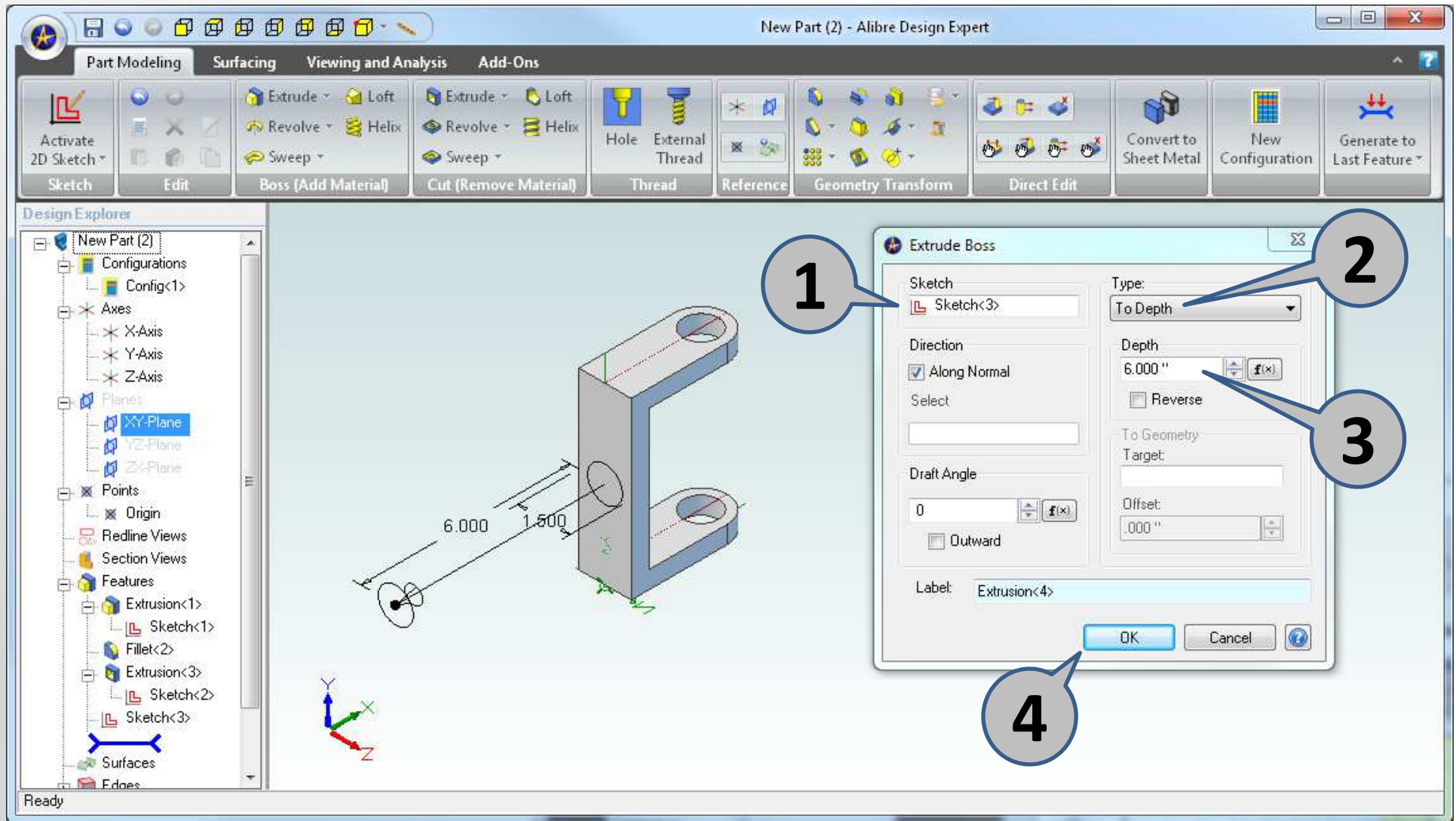
1. The view of the work area should be similar to the screen capture.



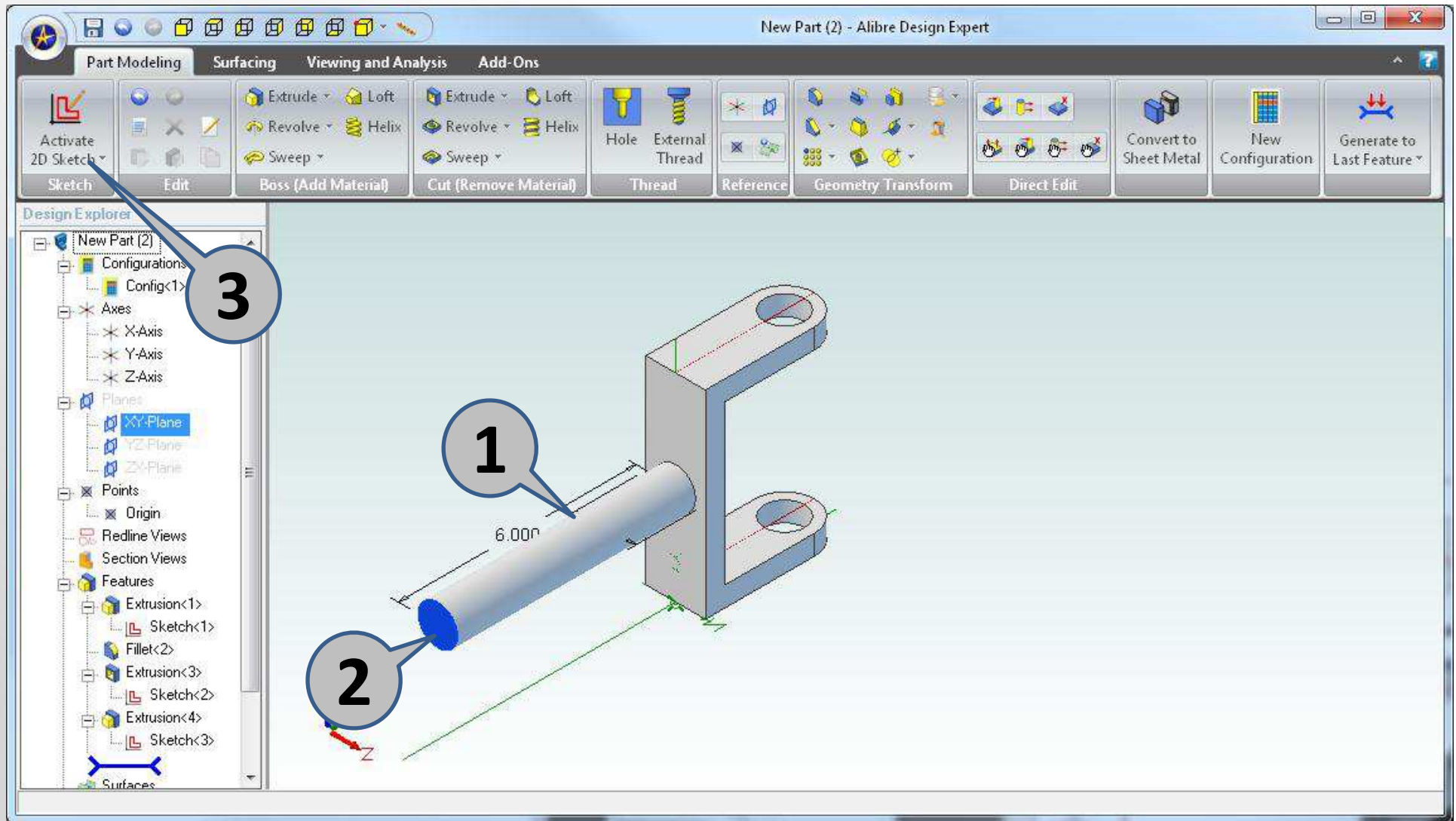
1. Click on the **Circle** tool.
2. Position the mouse cursor 2.50" above the origin. Click and release to create the center of the circle.
3. Move the mouse cursor one grid square to the right; this will create a Ø1.000" circle. Click and release to complete the circle.



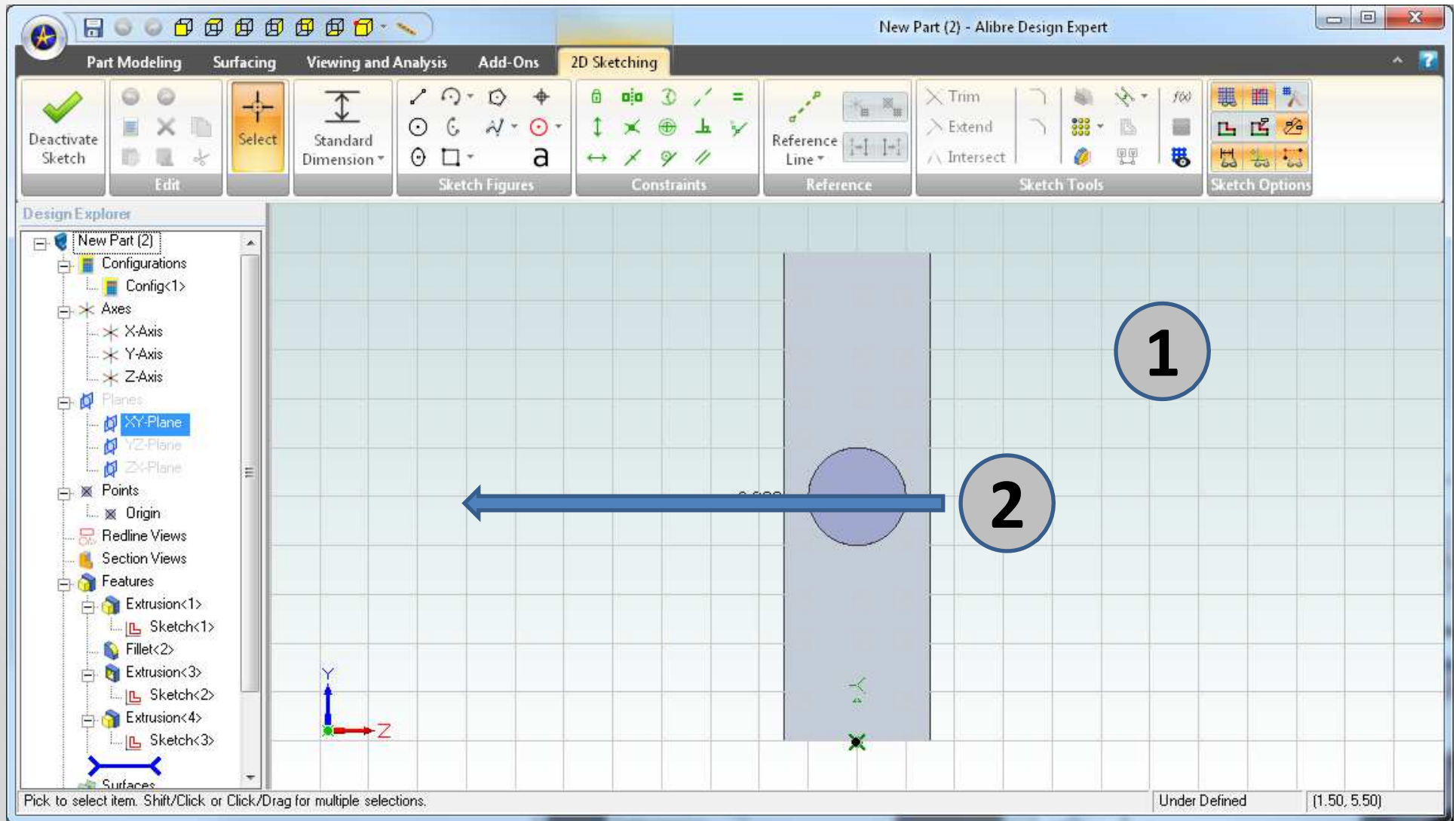
1. Click on the **Part Modeling** tab in the ribbon.
2. Click on the **Extrude Boss** icon.



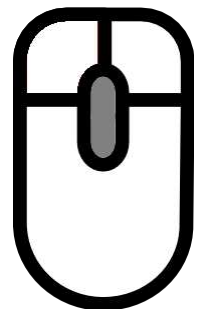
1. Confirm that **Sketch<3>** is the **Sketch** to be extruded.
2. Confirm that **To Depth** is the **Type** of extrusion.
3. Enter a value of **6.0 inches** in the **Depth** entry box.
4. Click **OK**.

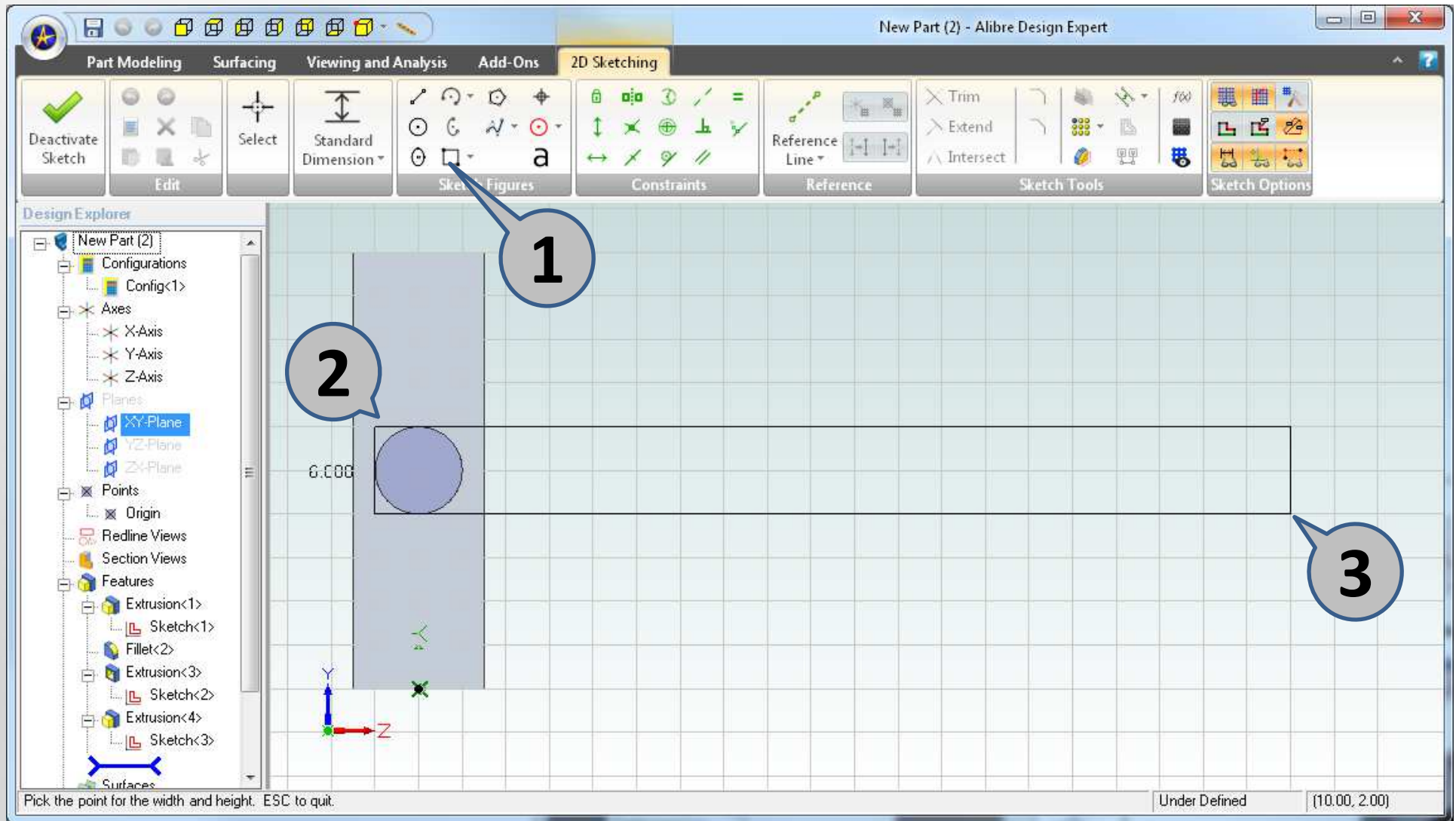


1. Confirm similar results for the **Extrude Boss** operation.
2. Click on the indicated face.
3. Click **Activate 2D Sketch**.

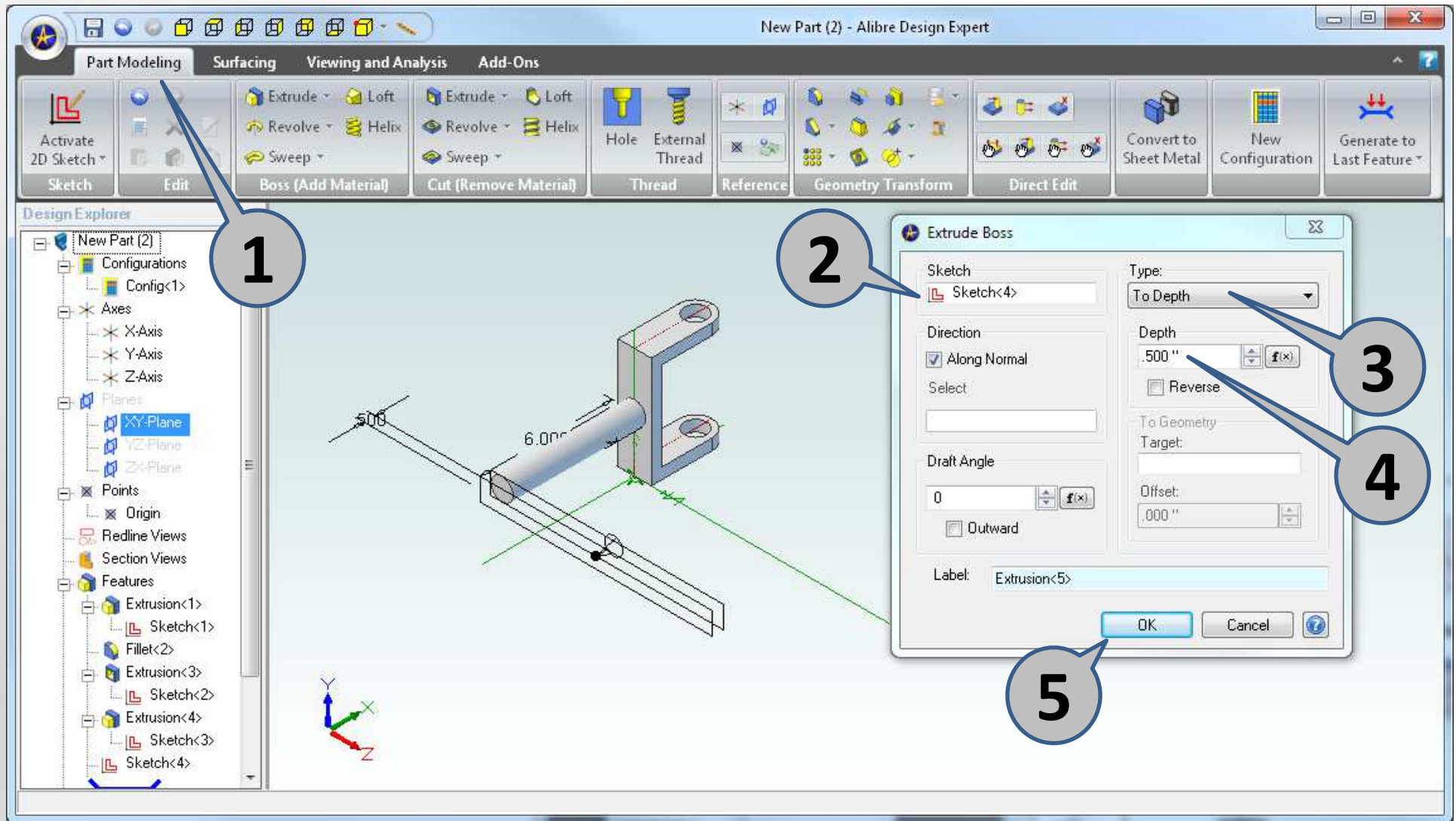


1. The view of the work area should be similar to the above image.
2. Position the mouse near the center of the screen, hold down the **Middle Mouse** button, and drag the mouse to the left.

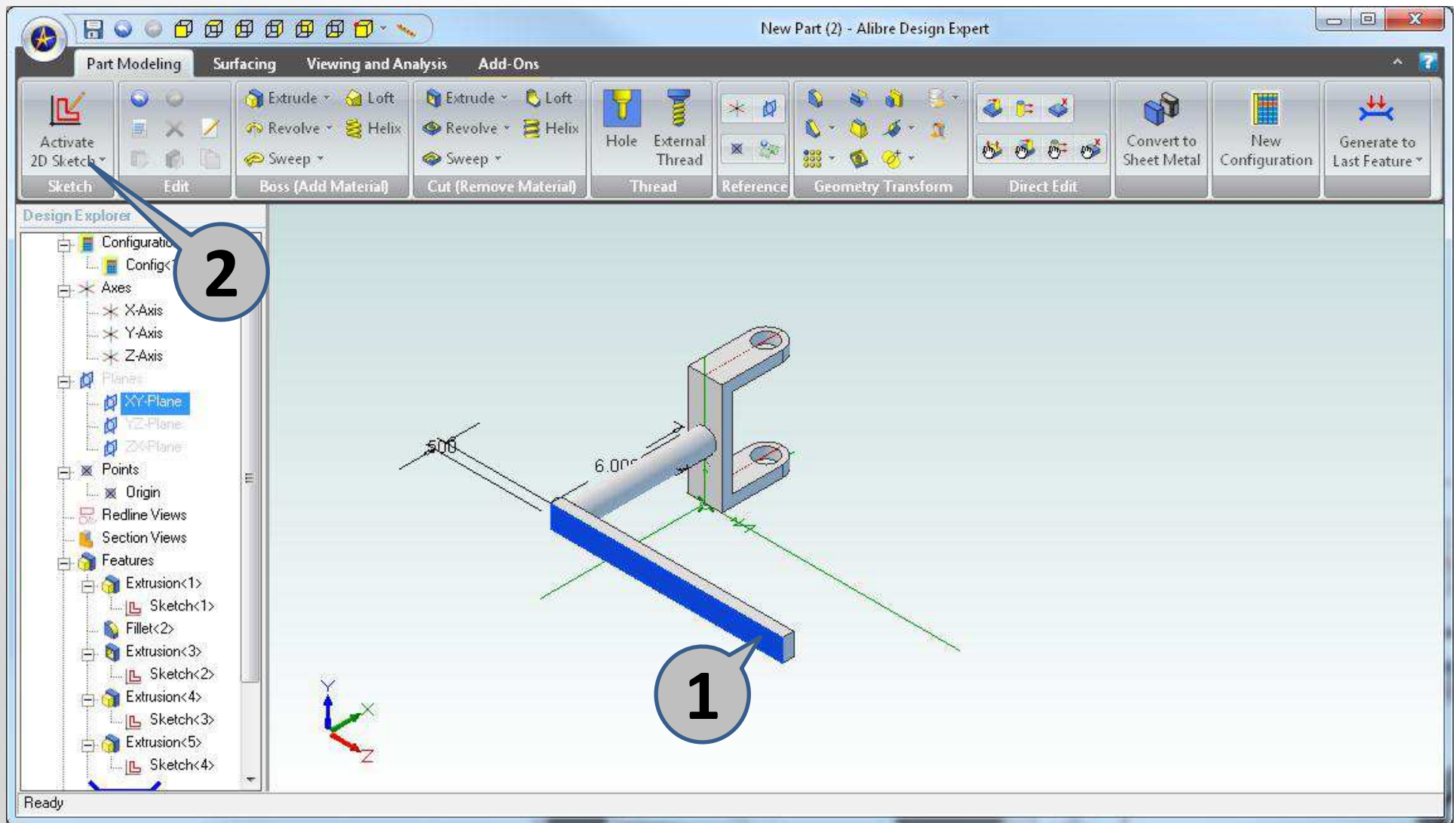




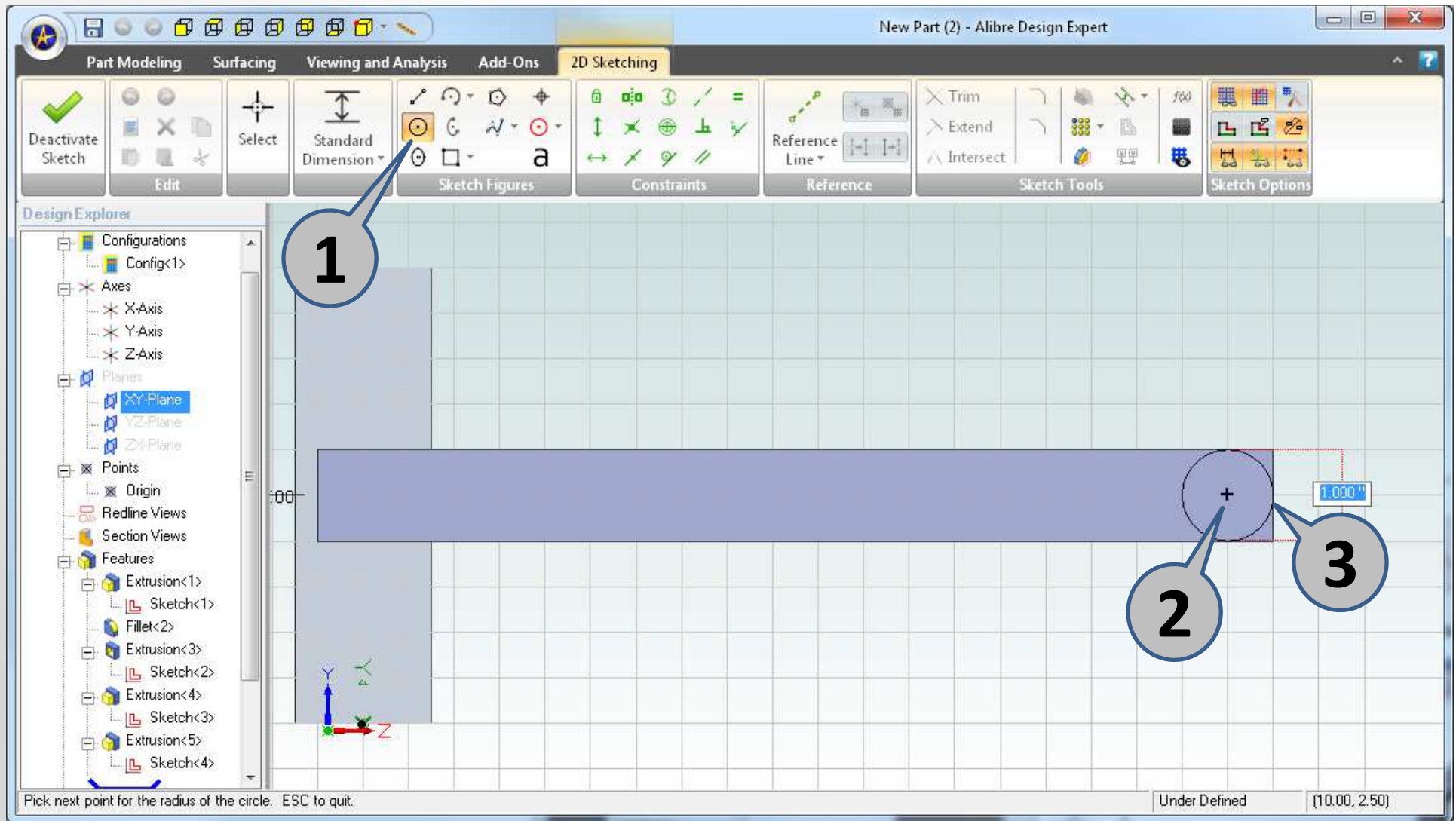
1. Click on the **Rectangle by Two Corners** tool.
2. Position the mouse cursor as shown. Click and release to create the first corner of the rectangle.
3. Move the mouse cursor to the right and upwards until the measurement indicator attached to the cursor displays **Width: 10.000** and **Height: 1.000**. Click and release to complete the rectangle.



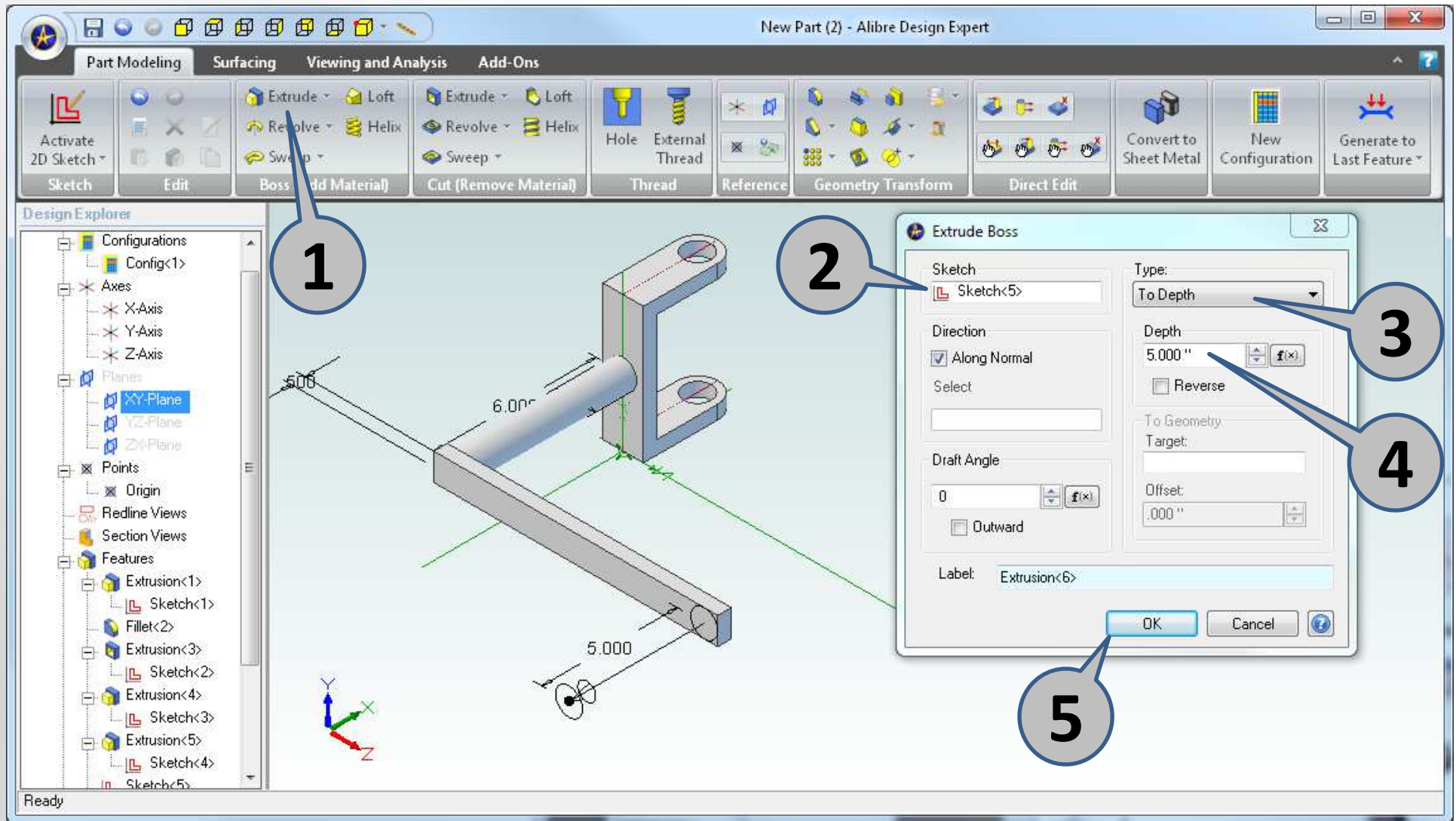
1. Click on the **Part Modeling** tab and click on the **Extrude Boss** icon.
2. Confirm that **Sketch<4>** is the **Sketch** to be extruded.
3. Confirm that **To Depth** is the **Type** of extrusion.
4. Enter a value of **0.5 inches** for the Depth.
5. Click **OK**.



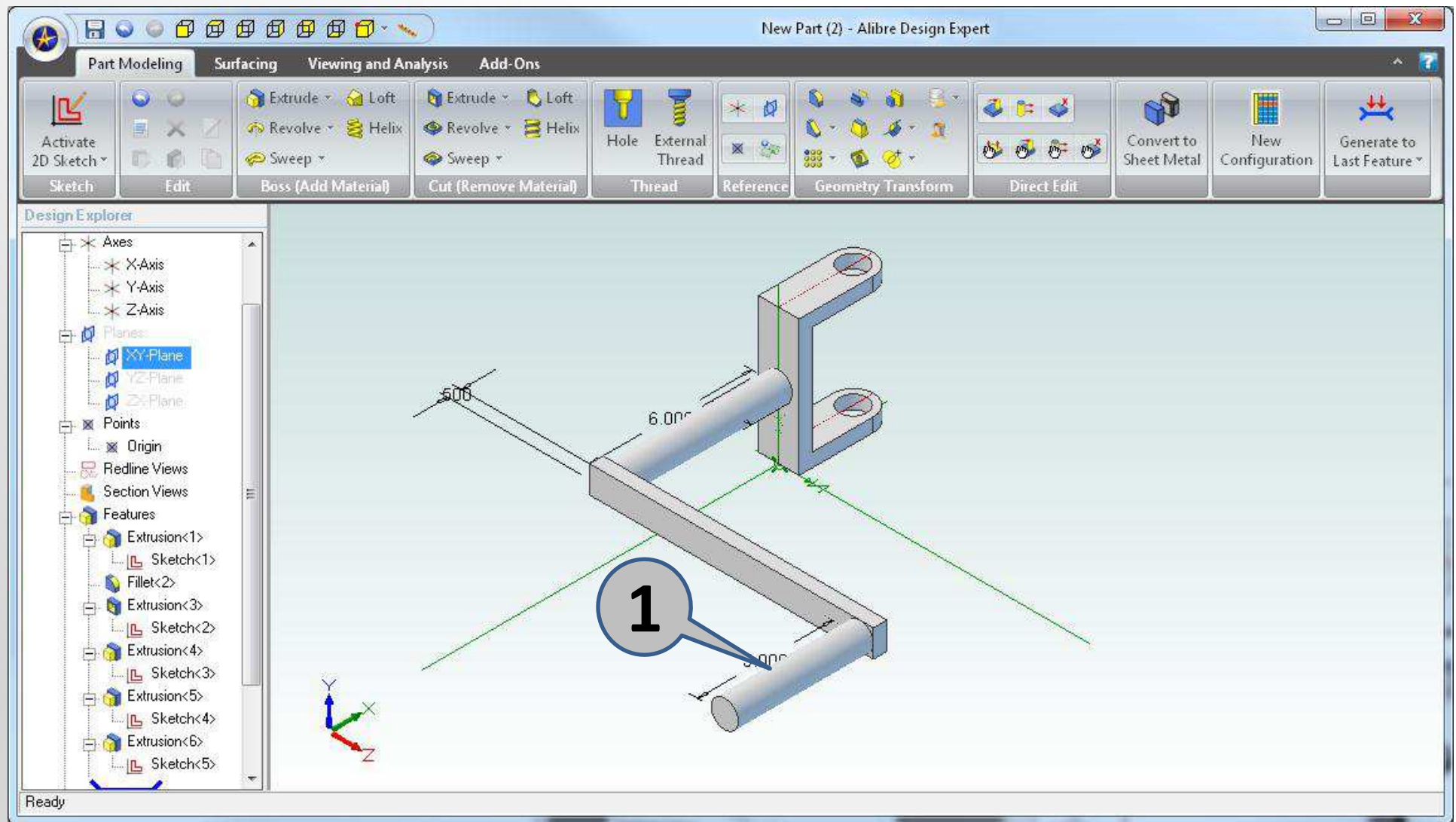
1. Click on the indicated face.
2. Click **Activate 2D Sketch**.



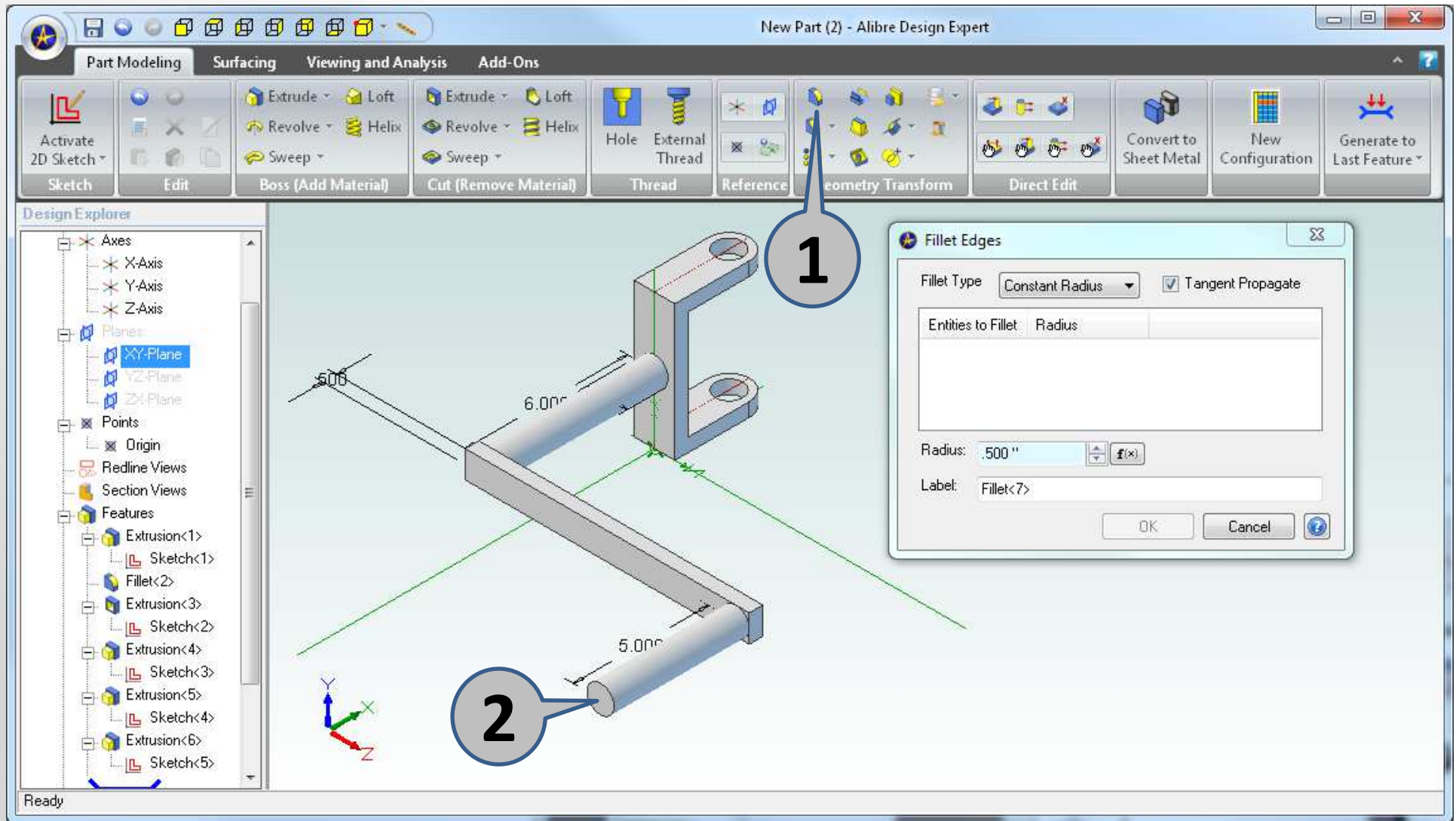
1. Click on the **Circle** tool.
2. Position the mouse cursor as shown. Click and release to create the center of the circle.
3. Move the mouse cursor to the right a distance of one grid square. Click and release to complete the circle.



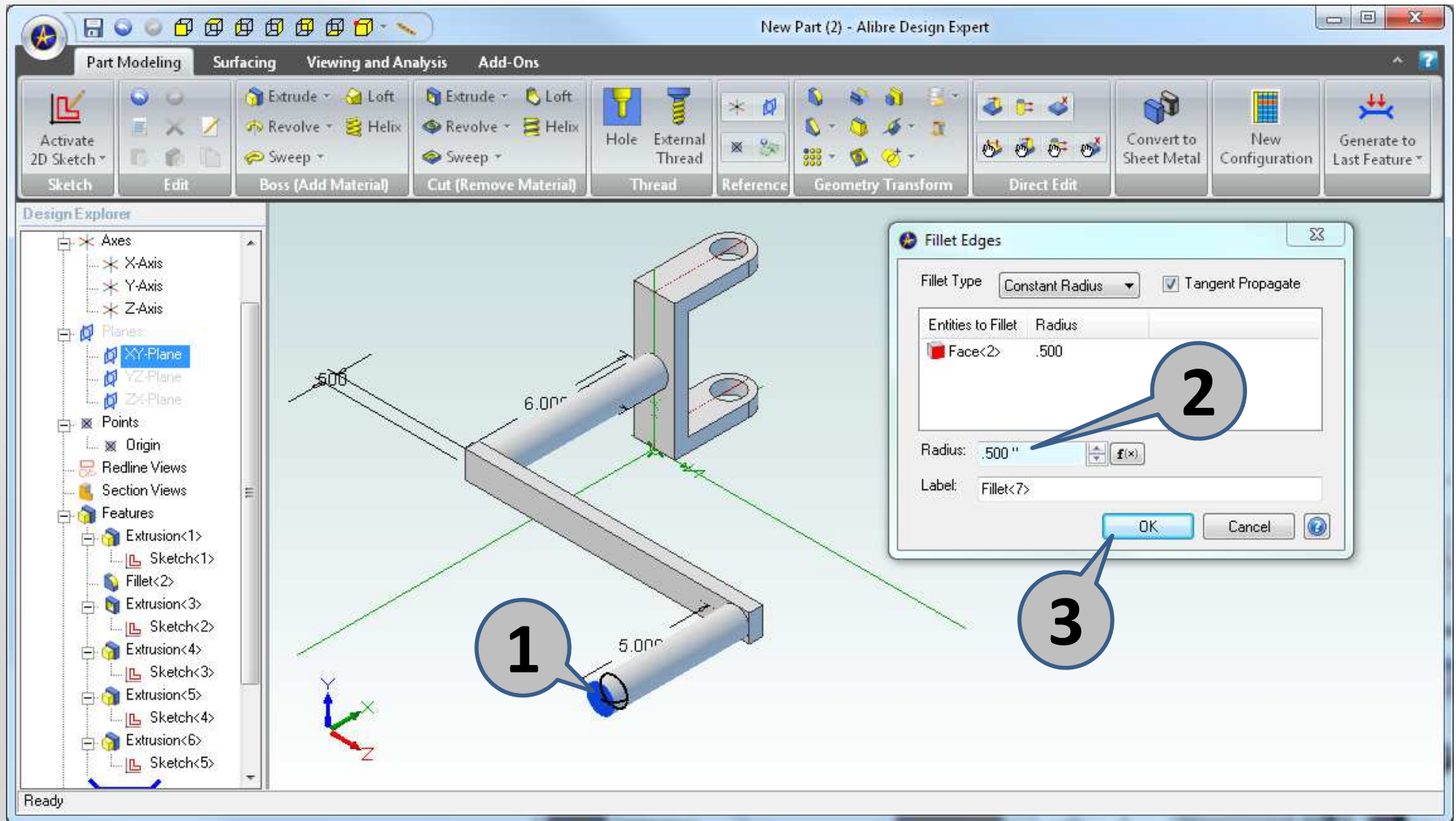
1. Click on the **Part Modeling** tab and click on the **Extrude Boss** icon.
2. Confirm that **Sketch<5>** is the **Sketch** to be extruded.
3. Confirm that **To Depth** is the **Type** of extrusion.
4. Enter a value of **5 inches** for the Depth.
5. Click **OK**.



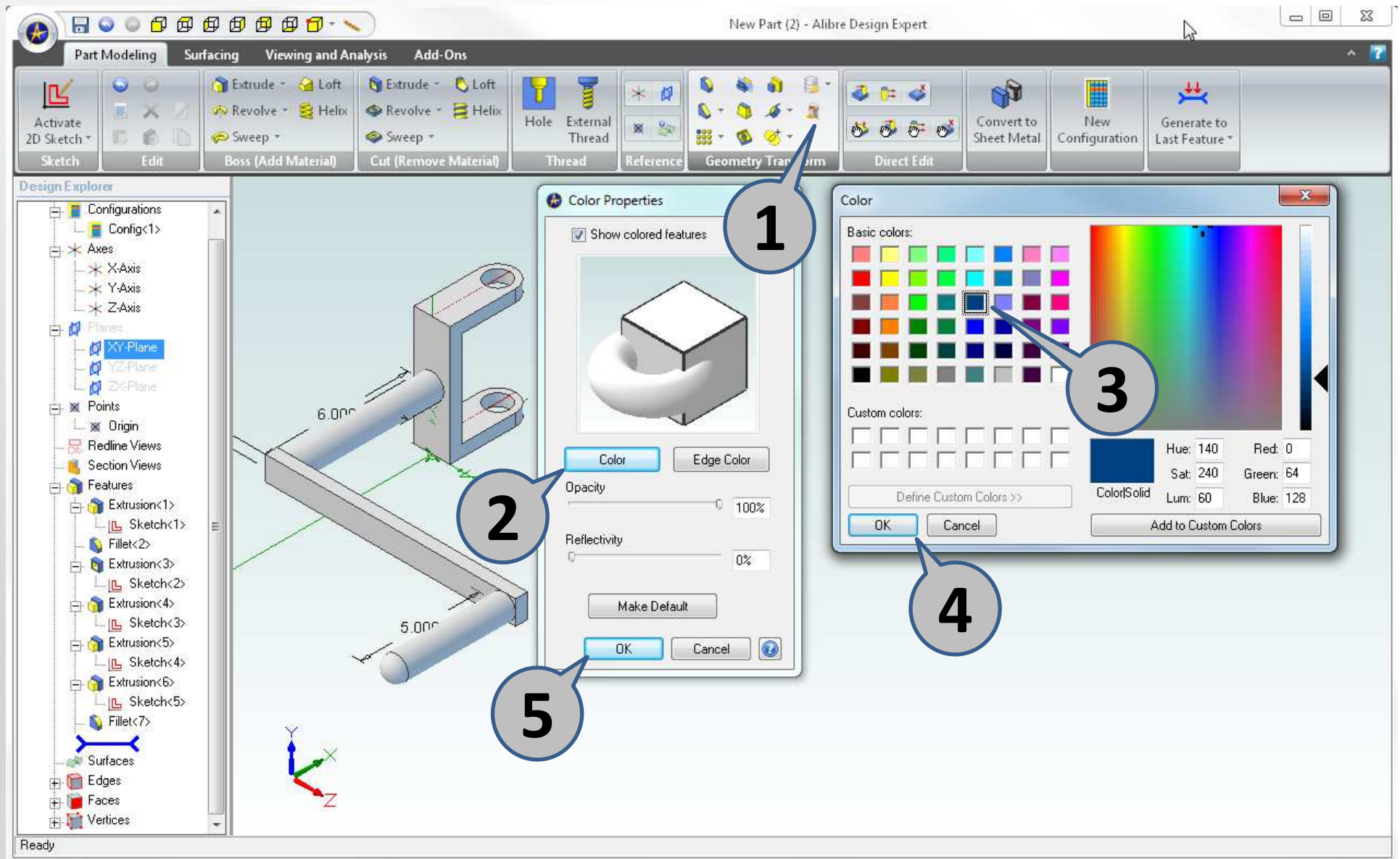
1. Confirm similar results for the **Extrude Boss** operation.



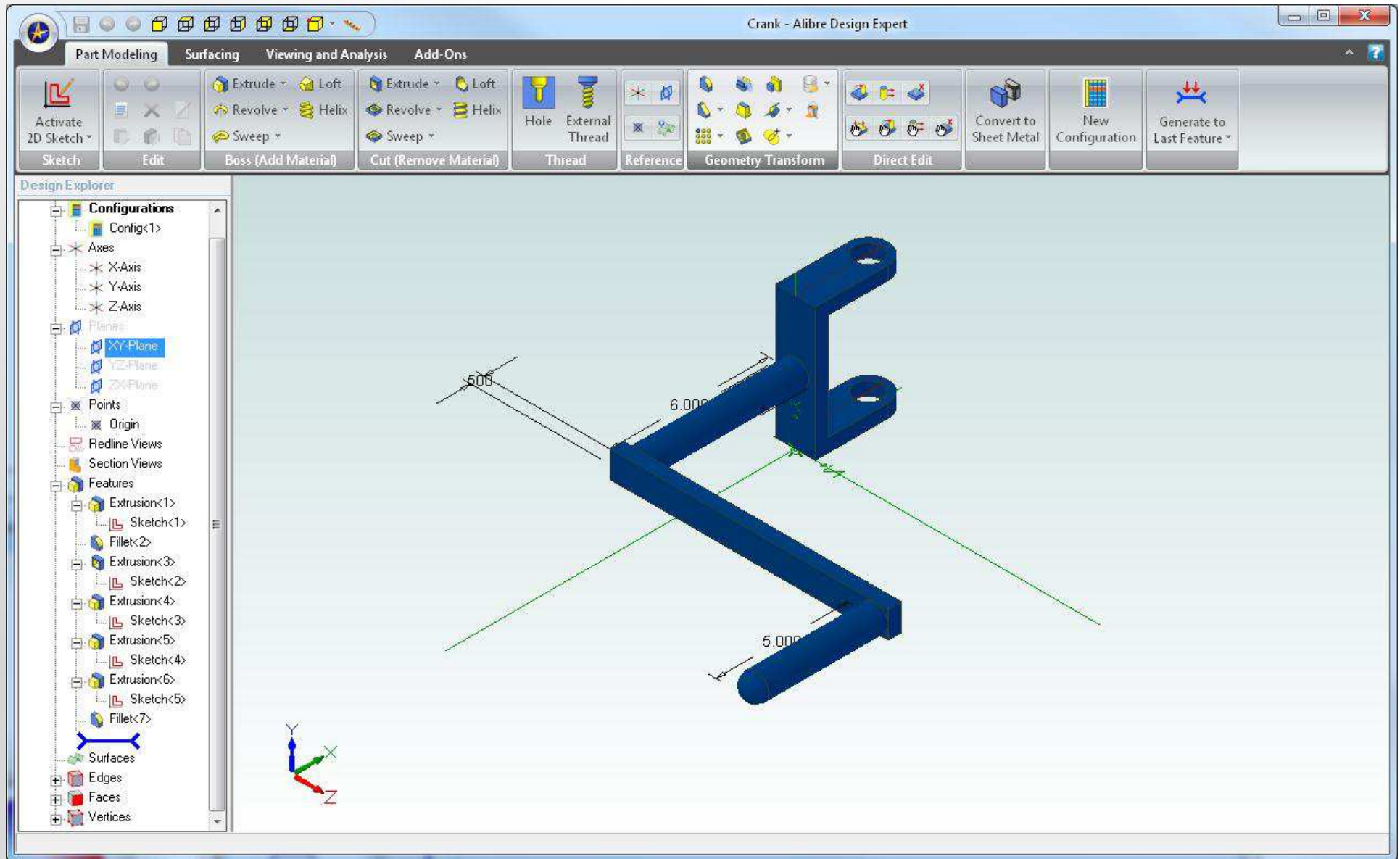
1. Select the **Fillet** icon.
2. Click on the face indicated.



1. Confirm the preview of the **Fillet** operation.
2. Enter a value of **0.5 inches** for the **Radius** in the **Fillet Edges** dialog box.
3. Click **OK**.



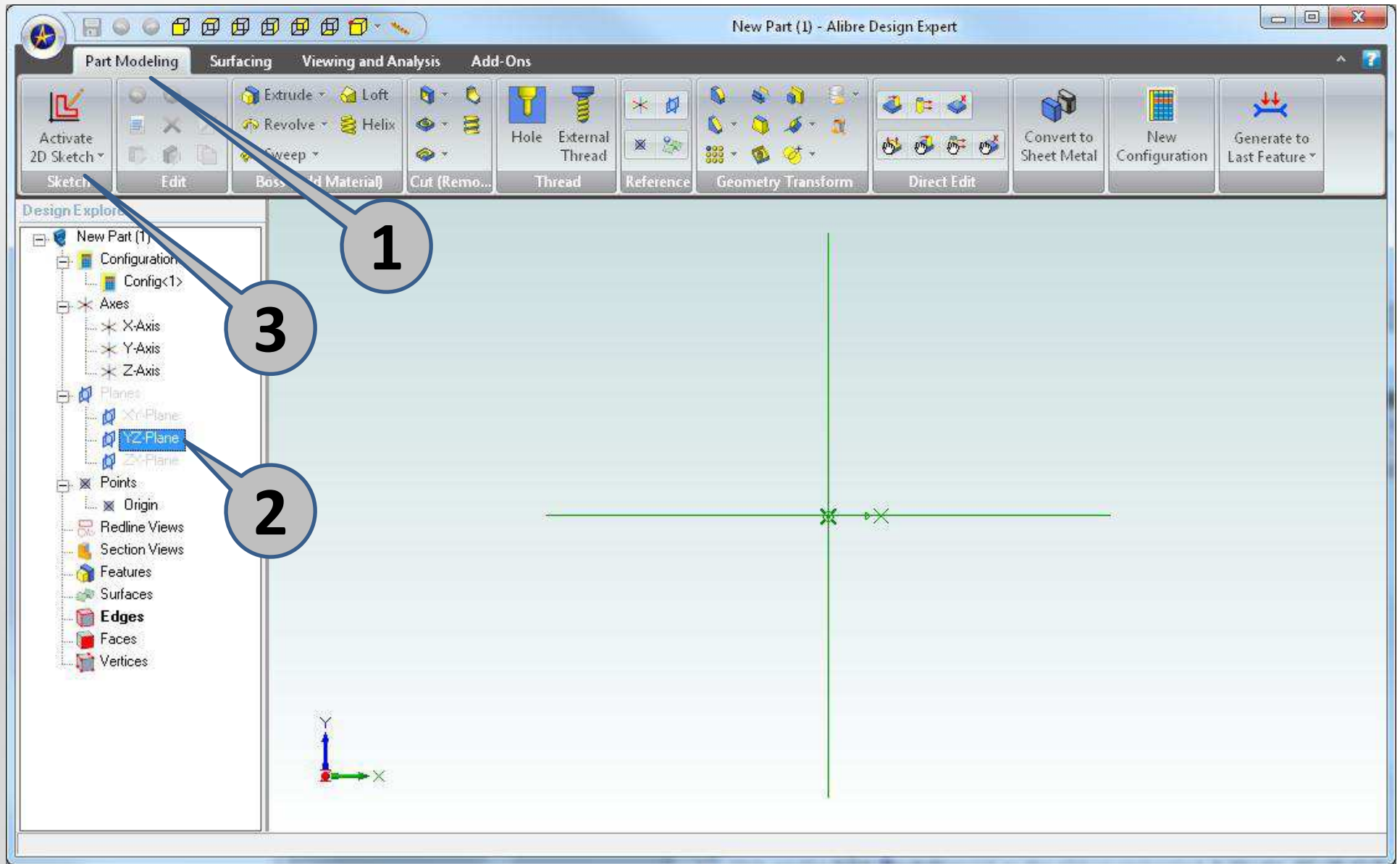
1. Select the **Part Color** icon.
2. Click on the **Color** button in the **Color Properties** dialog box.
3. Select a medium blue color in the **Color** dialog box.
4. Click **OK** in the **Color** dialog box.
5. Click **OK** in the **Color Properties** dialog box.



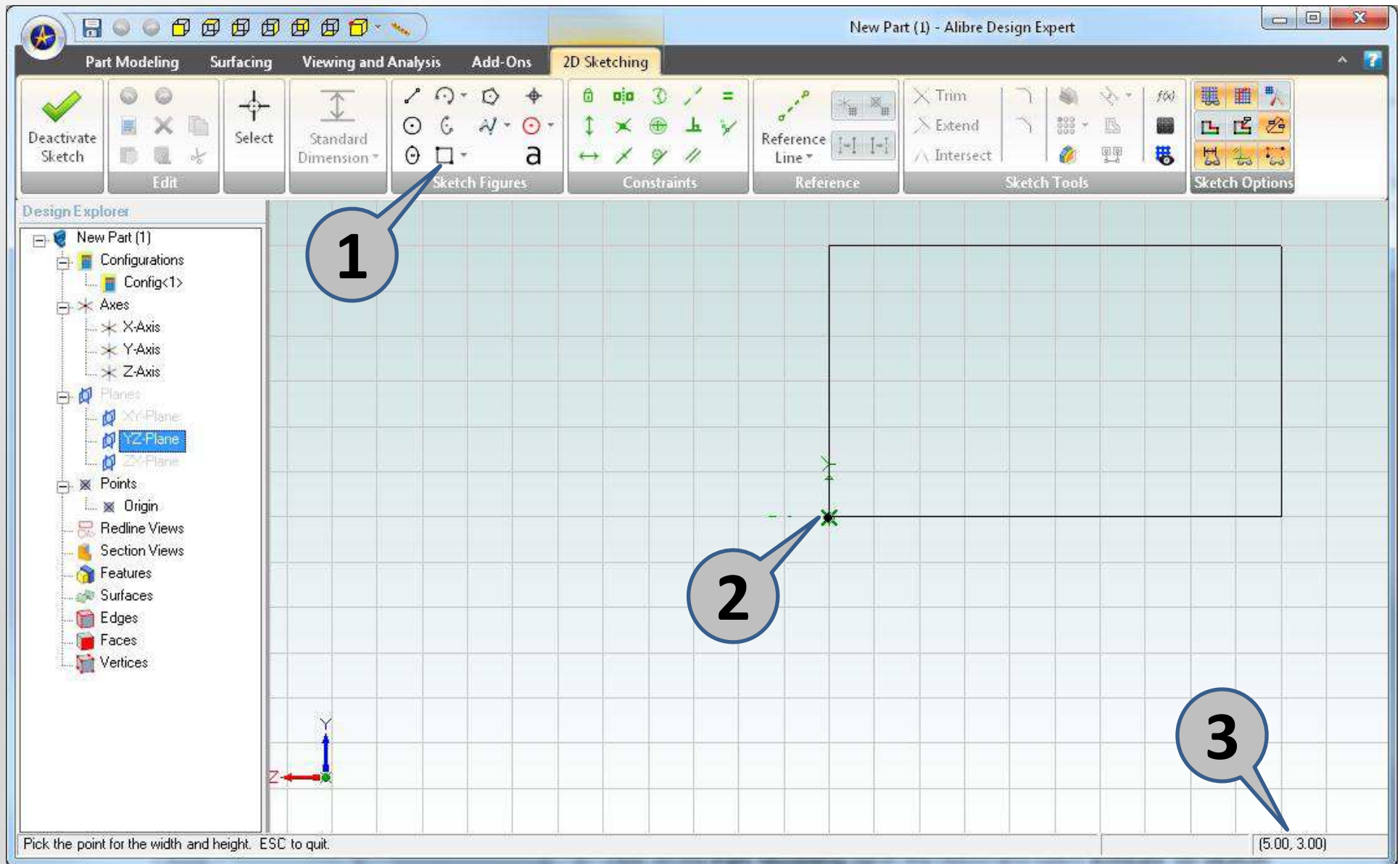
1. Confirm similar results for the color change.
2. Save this Part to the same location as the **Journal** with the name: **Crank** (not shown).
3. Close the **Part** workspace (not shown).



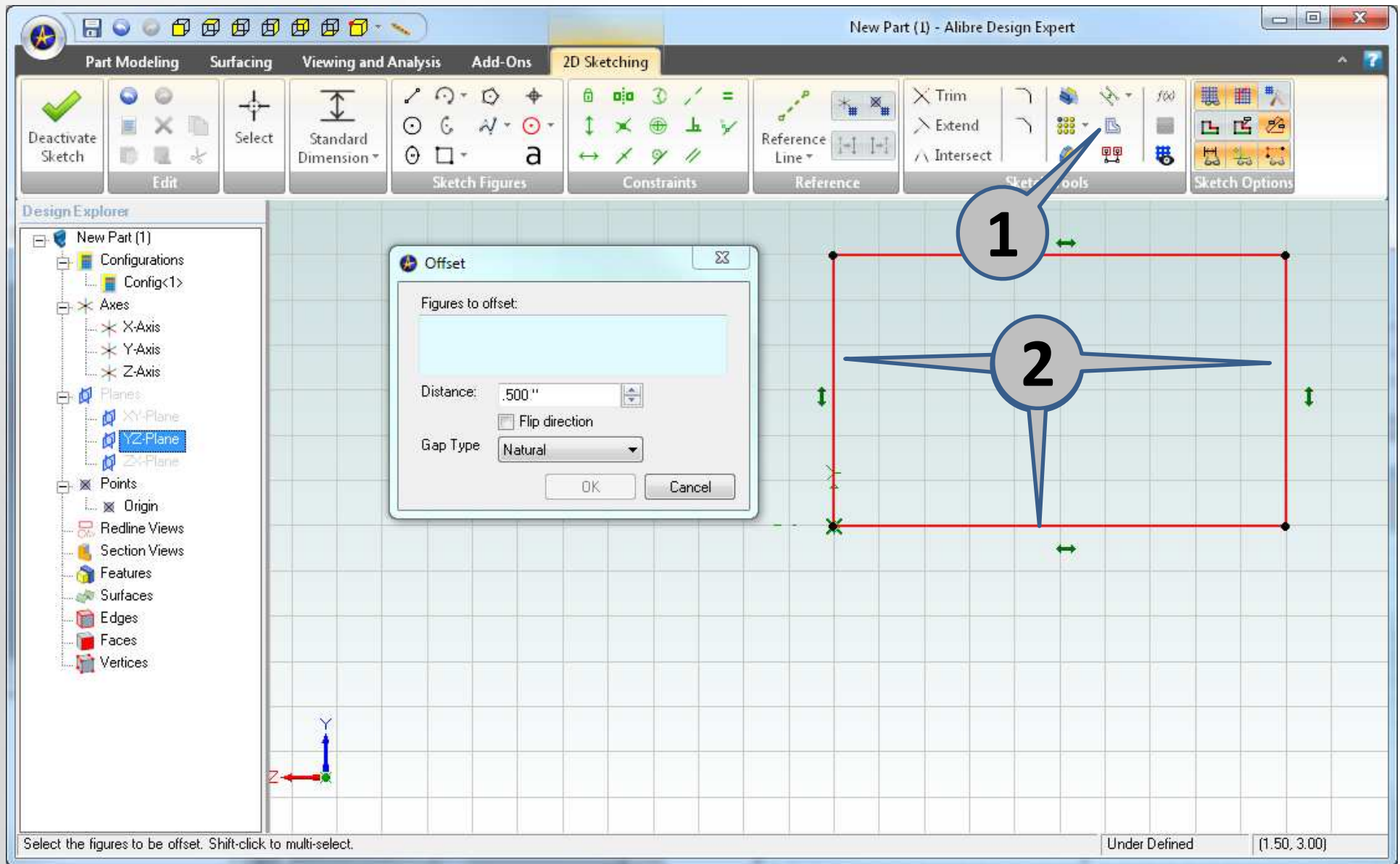
Creating the pulley



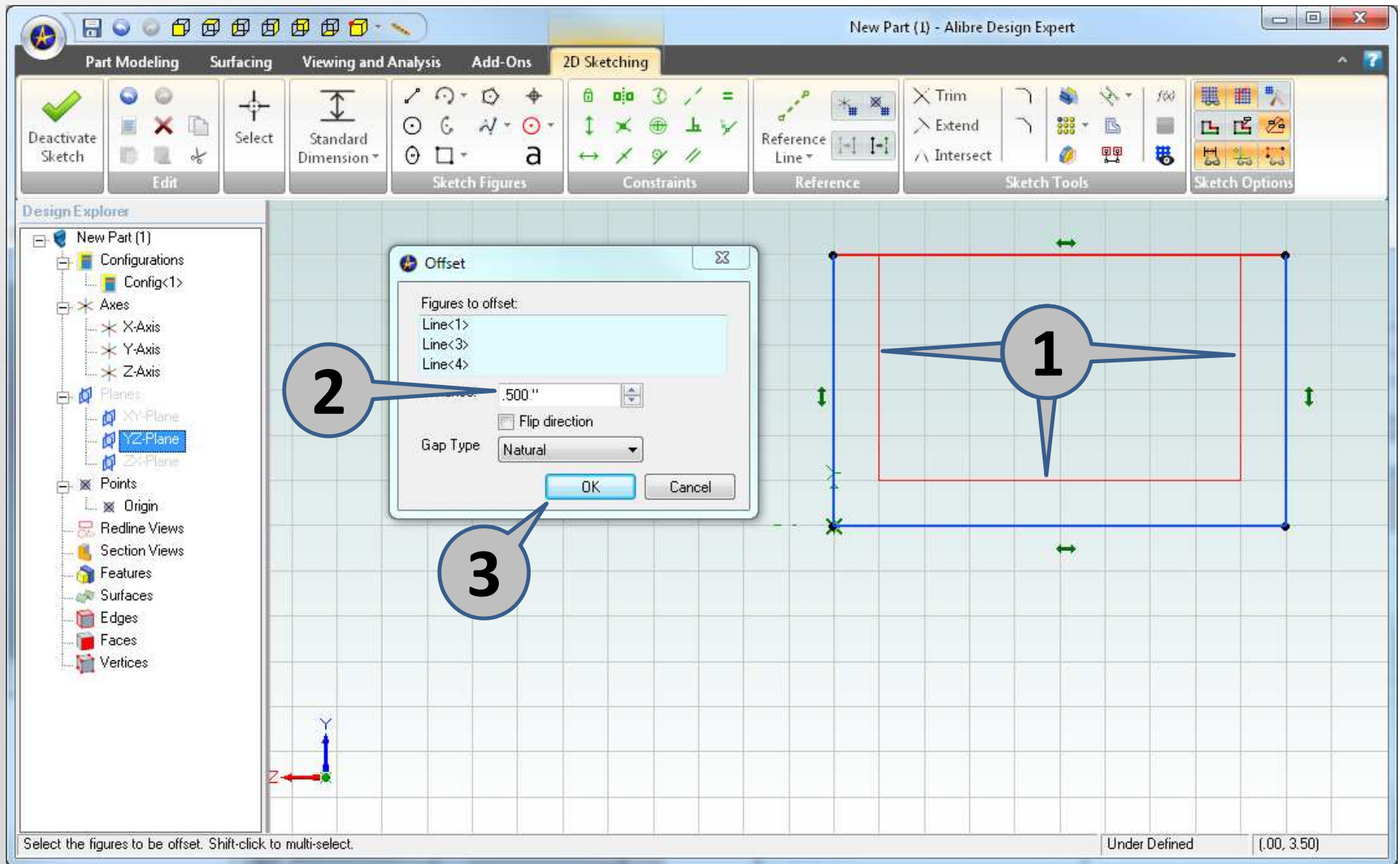
1. Click on the **Part Modeling** tab in the ribbon.
2. Click on the **YZ Plane** in the **Design Explorer**.
3. Click on the **Activate 2D Sketch** icon.



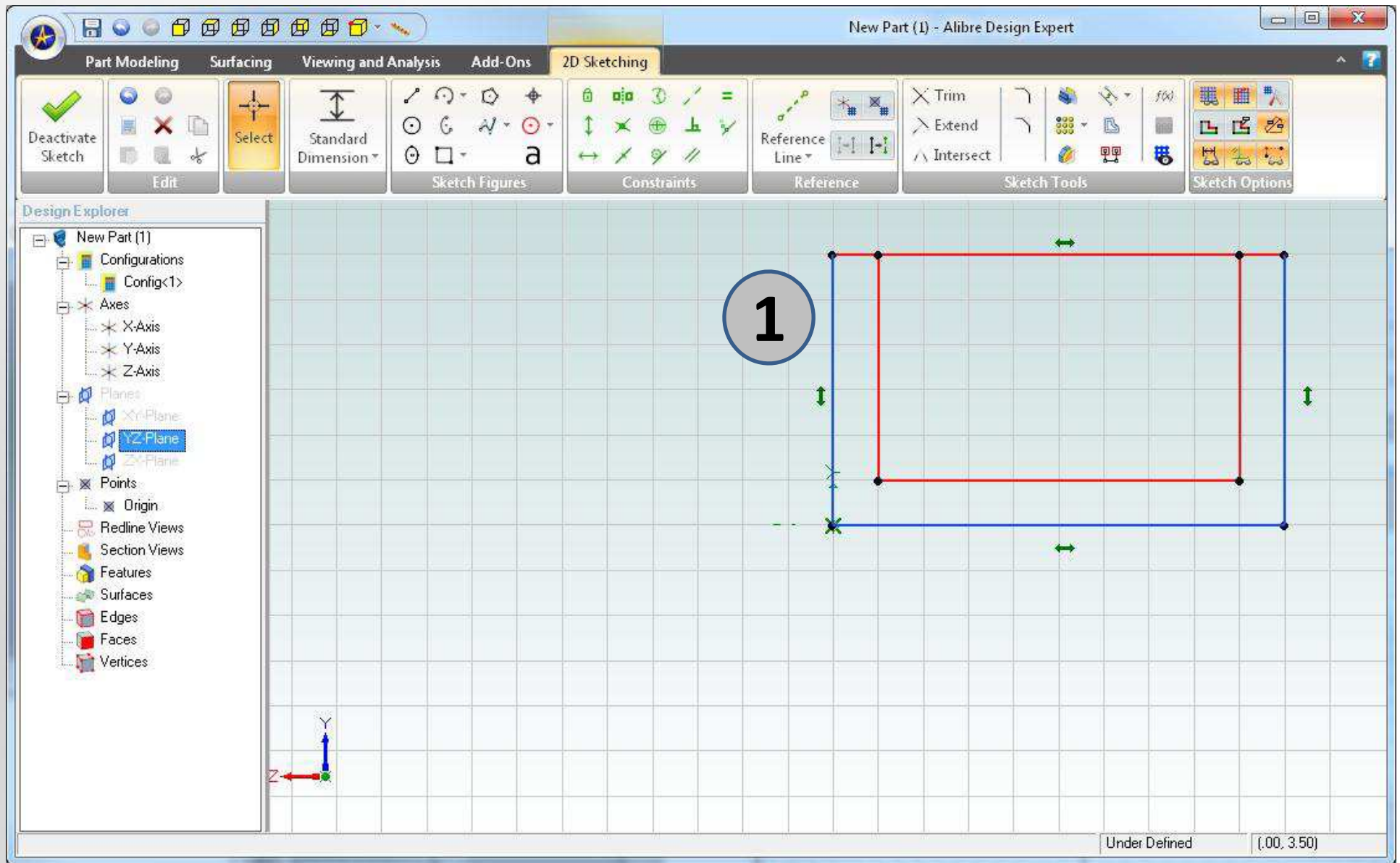
1. Click on the **Rectangle by Two Corners** tool.
2. Position the mouse cursor over the **Origin point**. *Click and release* to create the first corner of the rectangle.
3. Move the mouse cursor upwards and to the right until the measurement indicator attached to the cursor displays: **Width: 5.000** and **Height: 3.500**. *Click and release* to complete the rectangle.



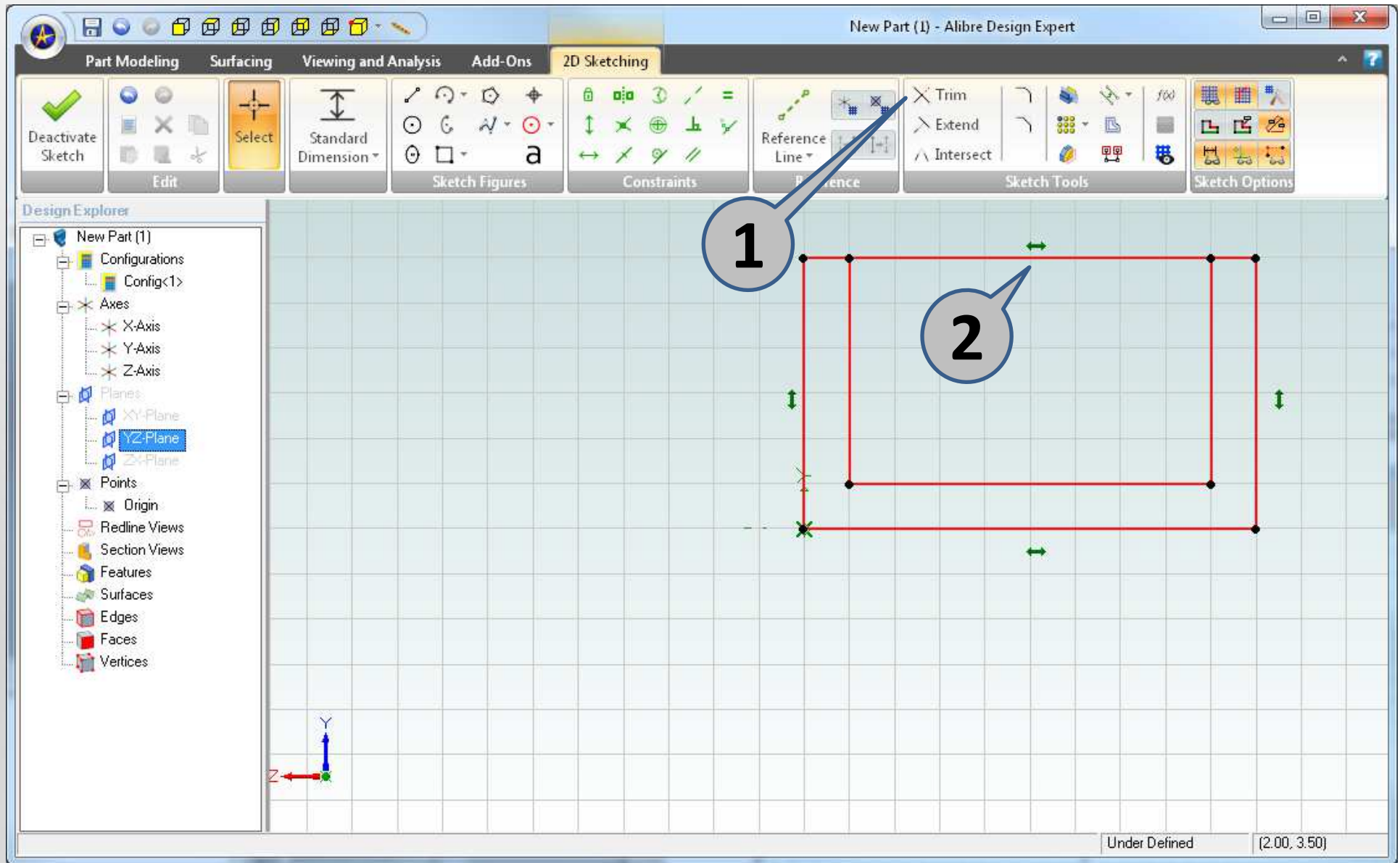
1. Select the **Offset** command.
2. Click to select the right , left and bottom segments of the rectangle. The segments will change color upon selection.



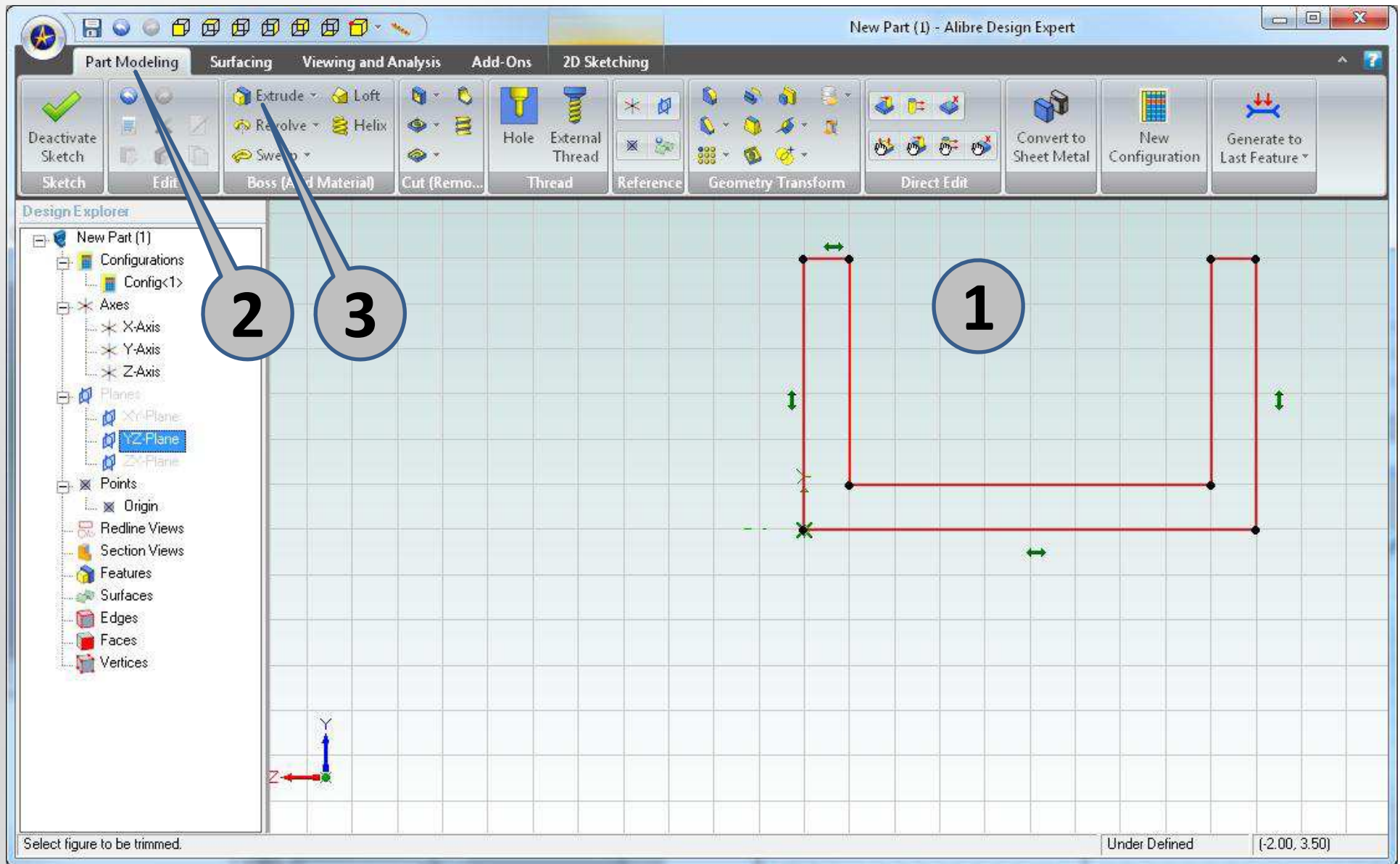
1. Confirm the preview of the **Offset** results.
2. Confirm the **Distance:** value in the **Offset** dialog box is **0.5 inch**.
3. Click **OK**.



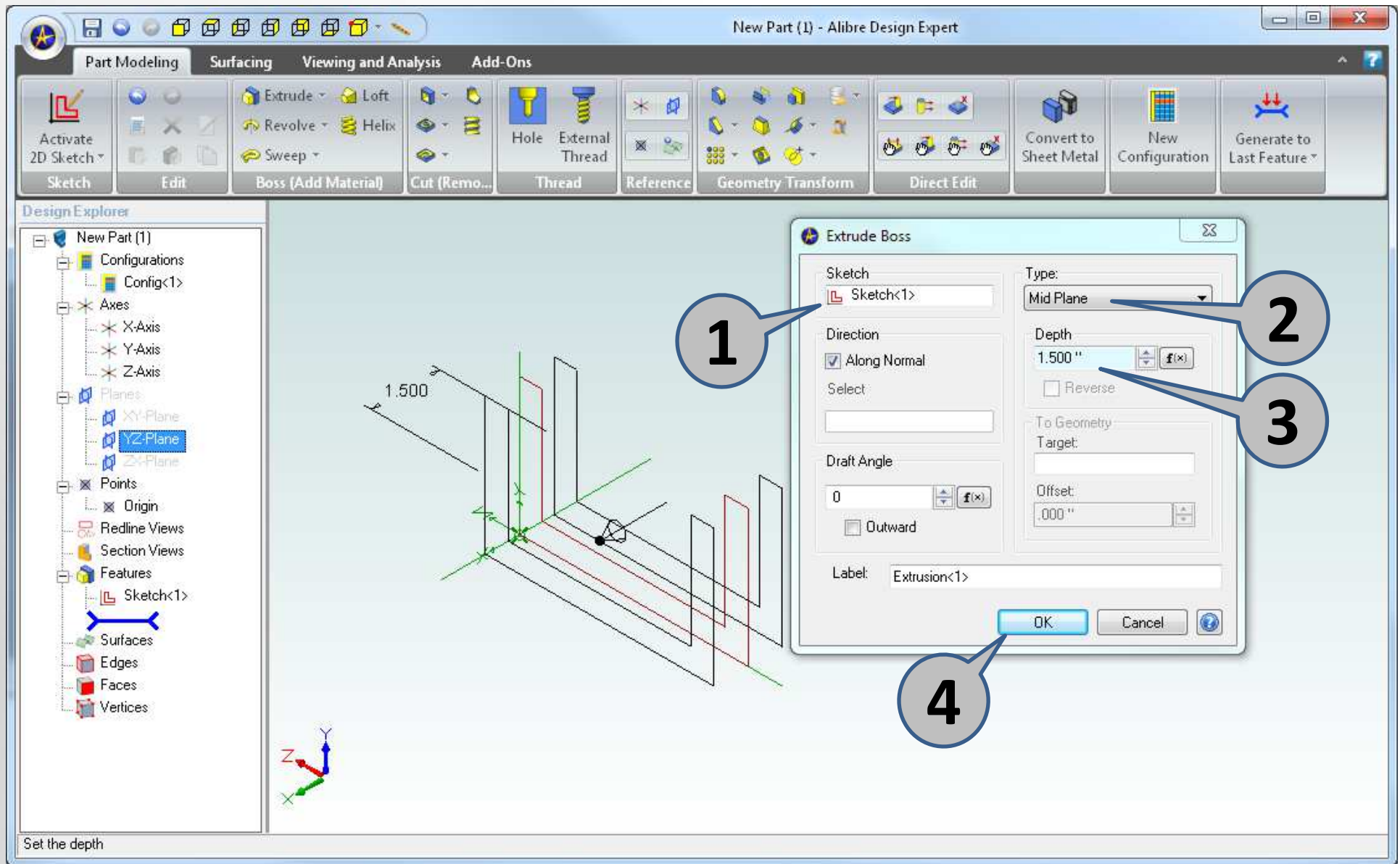
1. Confirm the result of the **Offset** operation.



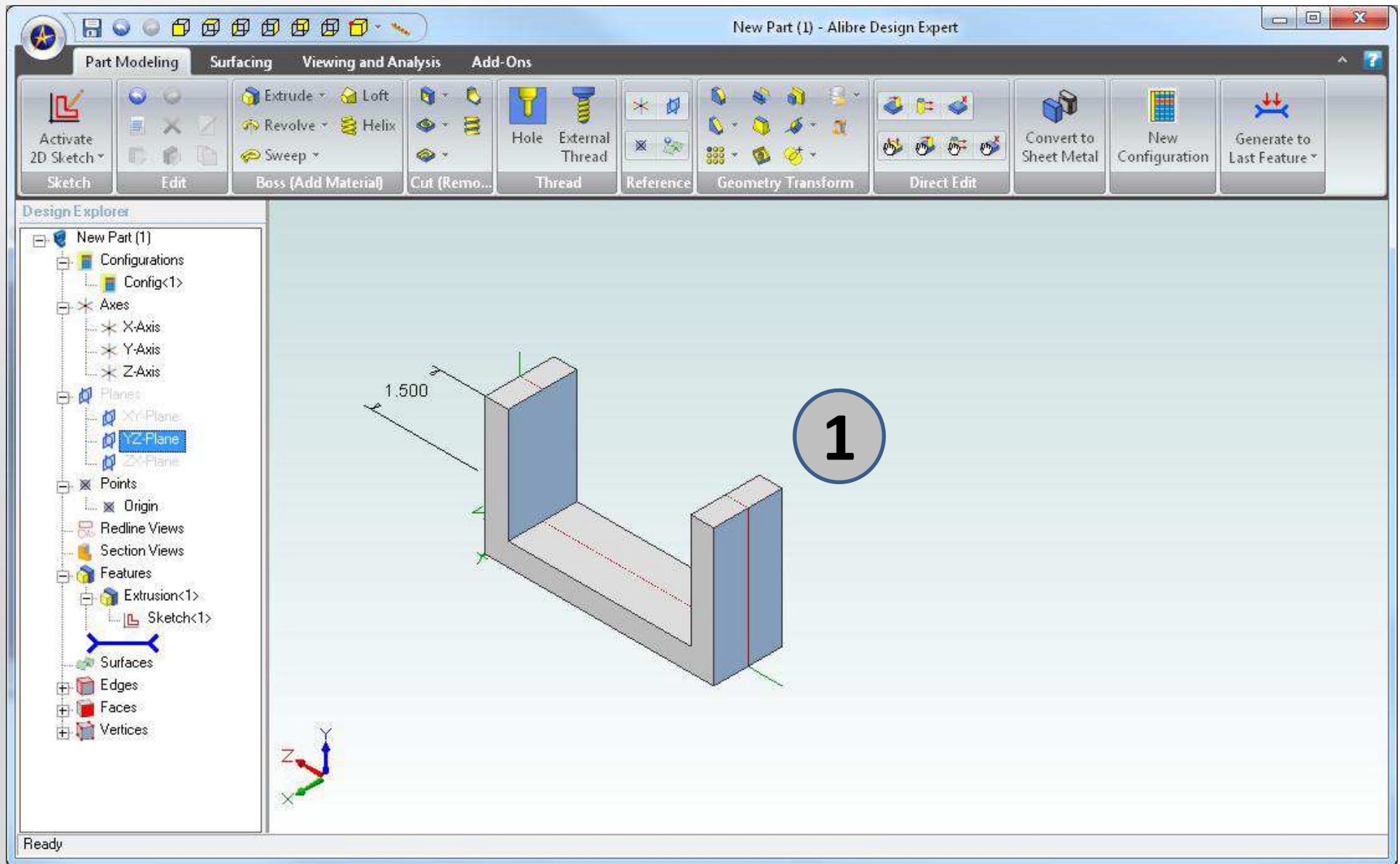
1. Select the **Trim** command.
2. Click on the line segment indicated.



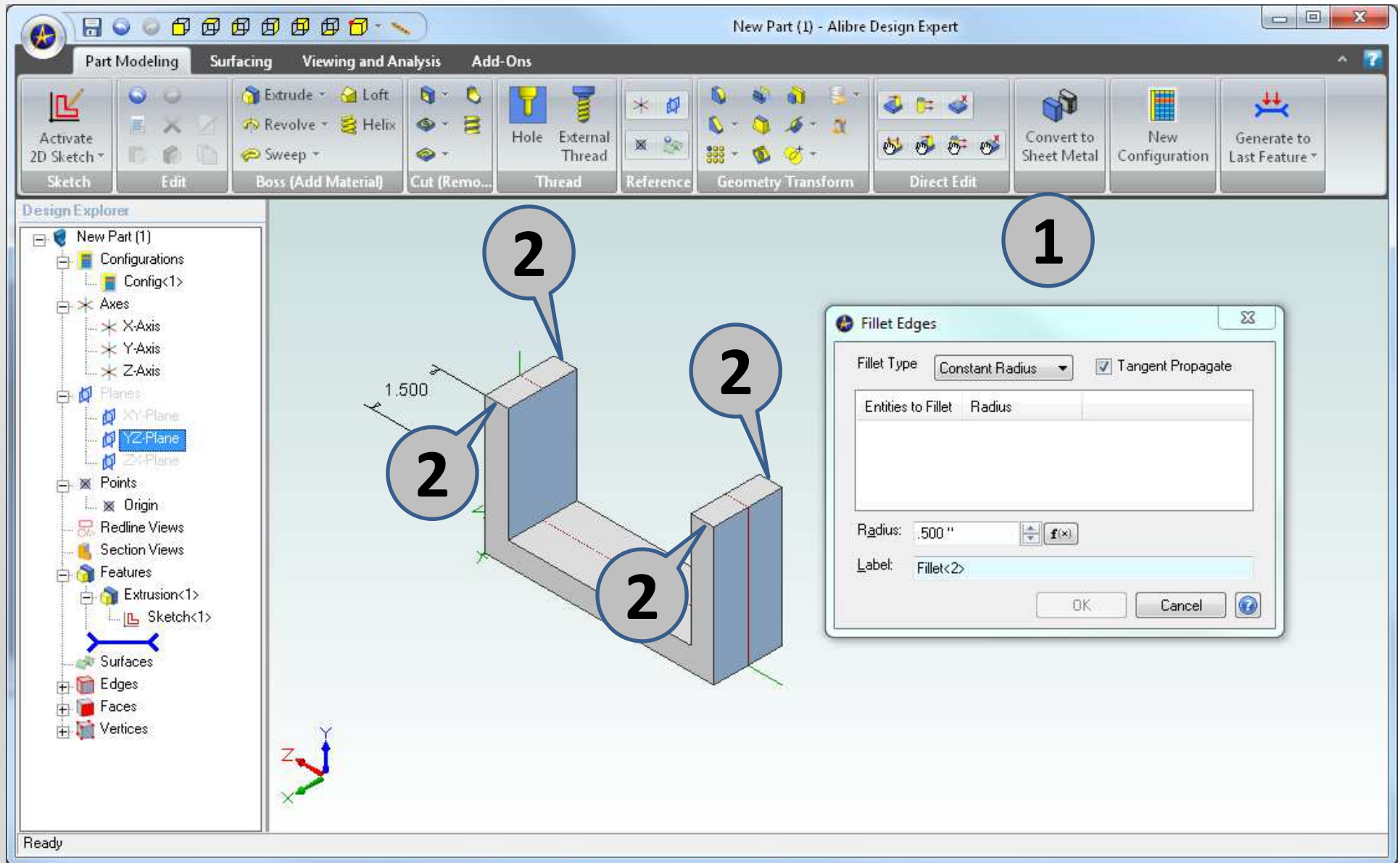
1. Confirm the result of the **Trim** operation.
2. Click on the **Part Modeling** tab in the ribbon (or click on the **Deactivate Sketch** icon).
3. Click on the **Extrude Boss** icon in the upper-right corner of the screen.



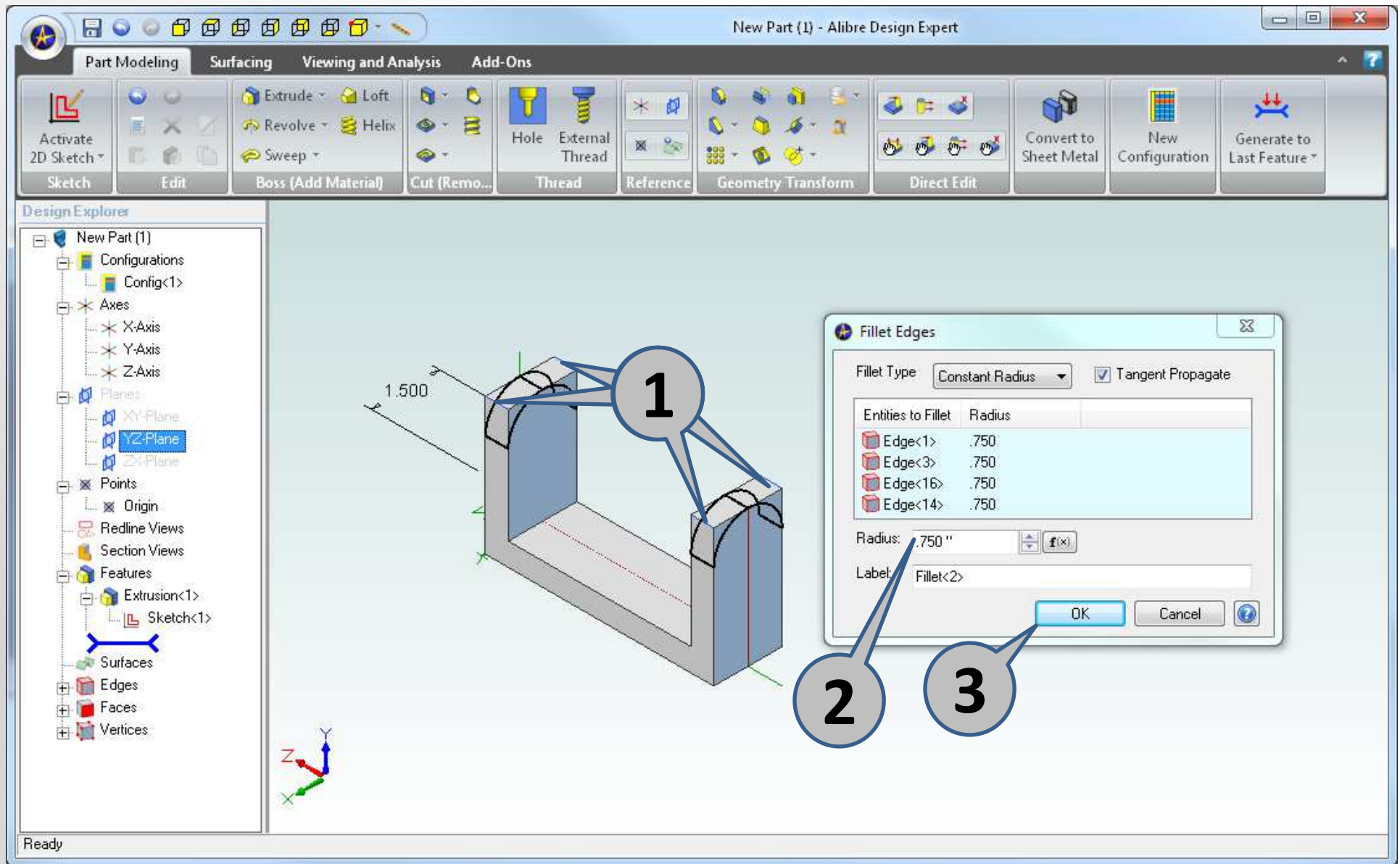
1. Confirm that **Sketch<1>** is the **Sketch** to be extruded.
2. Select **Mid Plane** as the **Type** of extrusion.
3. Enter a value of **1.5 inches** for the **Depth**.
4. Click **OK**.



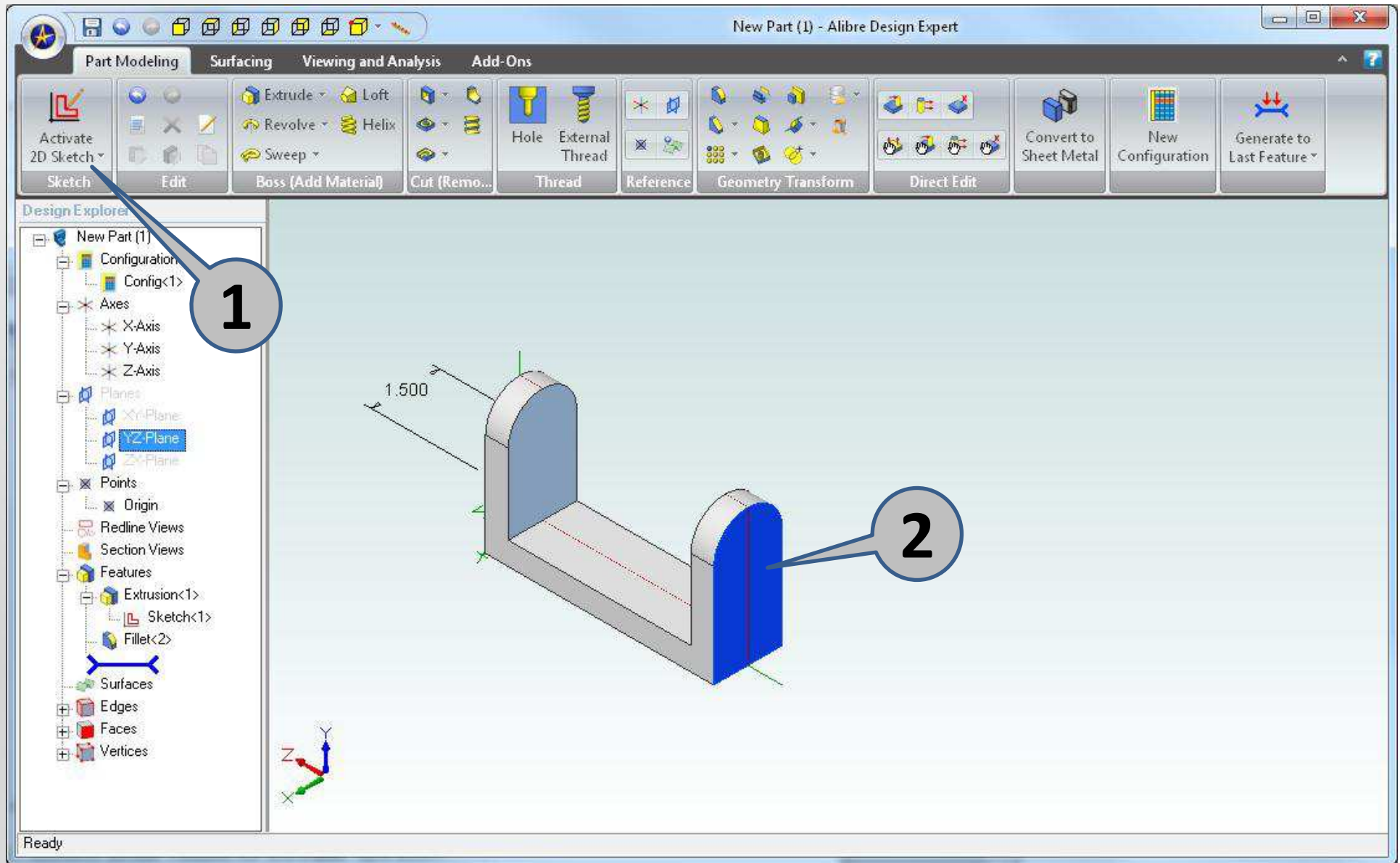
1. Confirm similar results for the **Extrude Boss** operation.



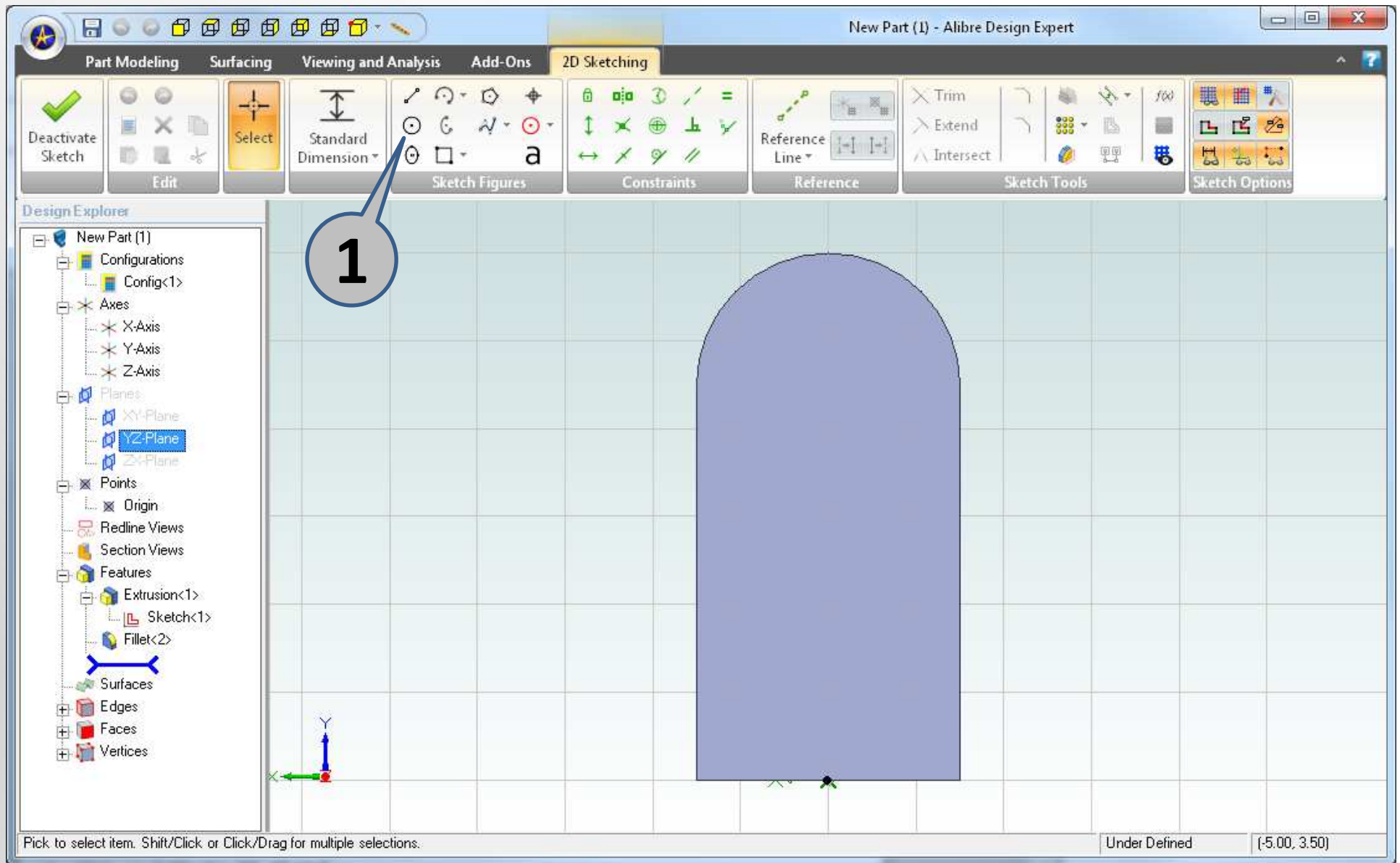
1. Click on the **Fillet** tool.
2. Move the **Fillet Edges** dialog box such that the entire part is visible.
3. Click on the four edges indicated.



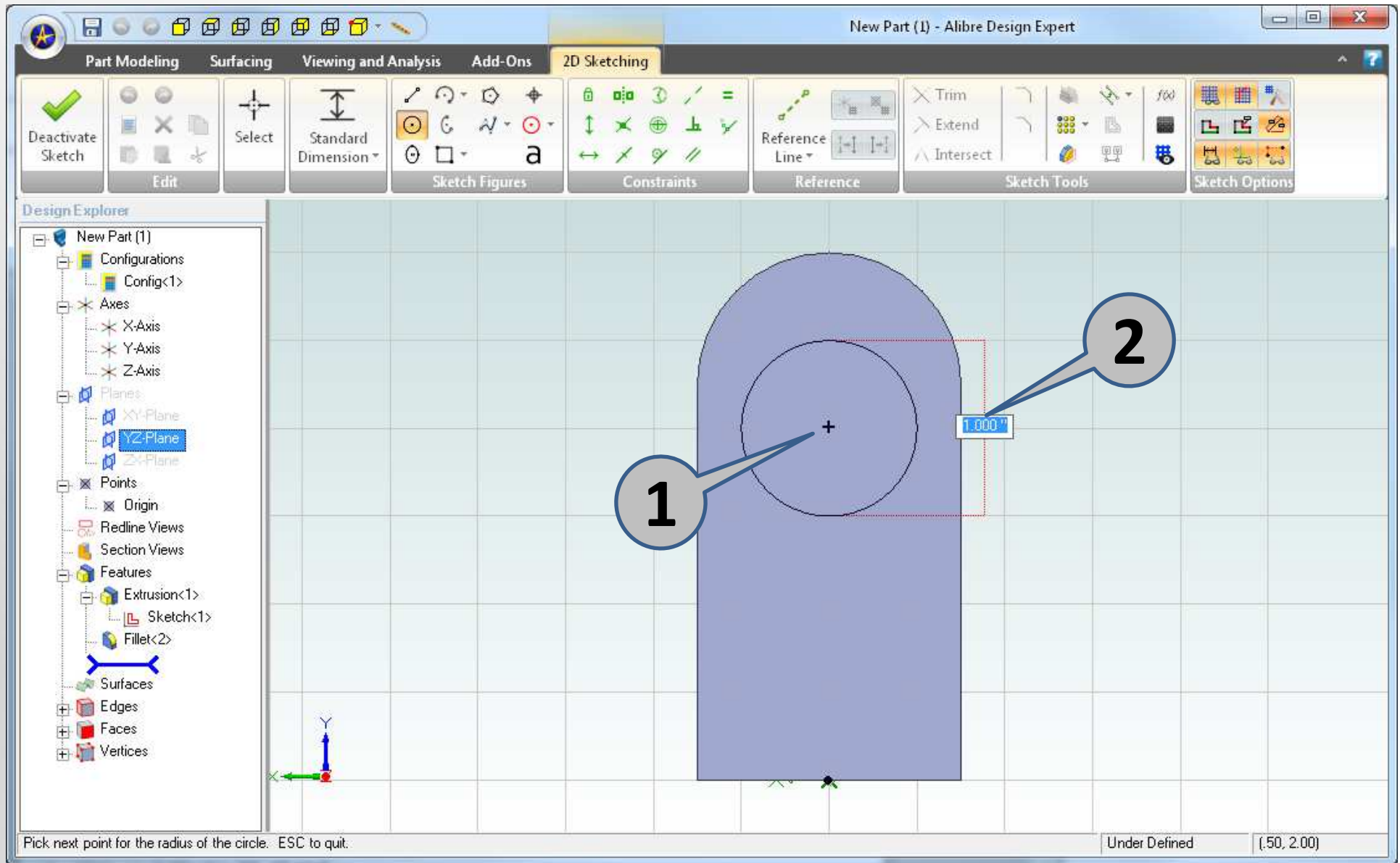
1. A preview of the **Fillet** operation is displayed.
2. In the **Fillet Edges** dialog box, change the **Radius:** value to **0.75 inch**.
3. Click **OK**.



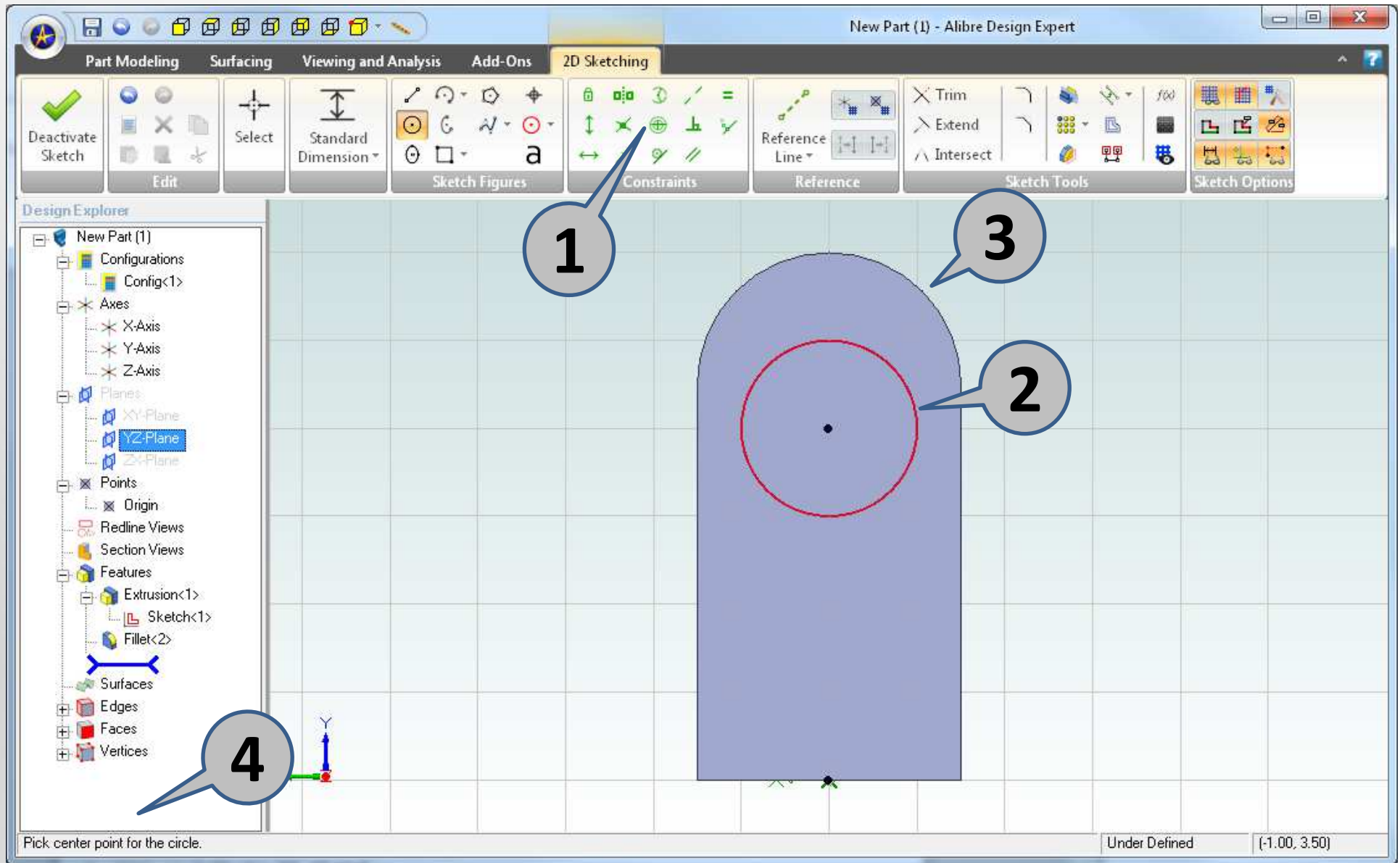
1. Click on the face indicated.
2. Click on **Activate 2D Sketch**.



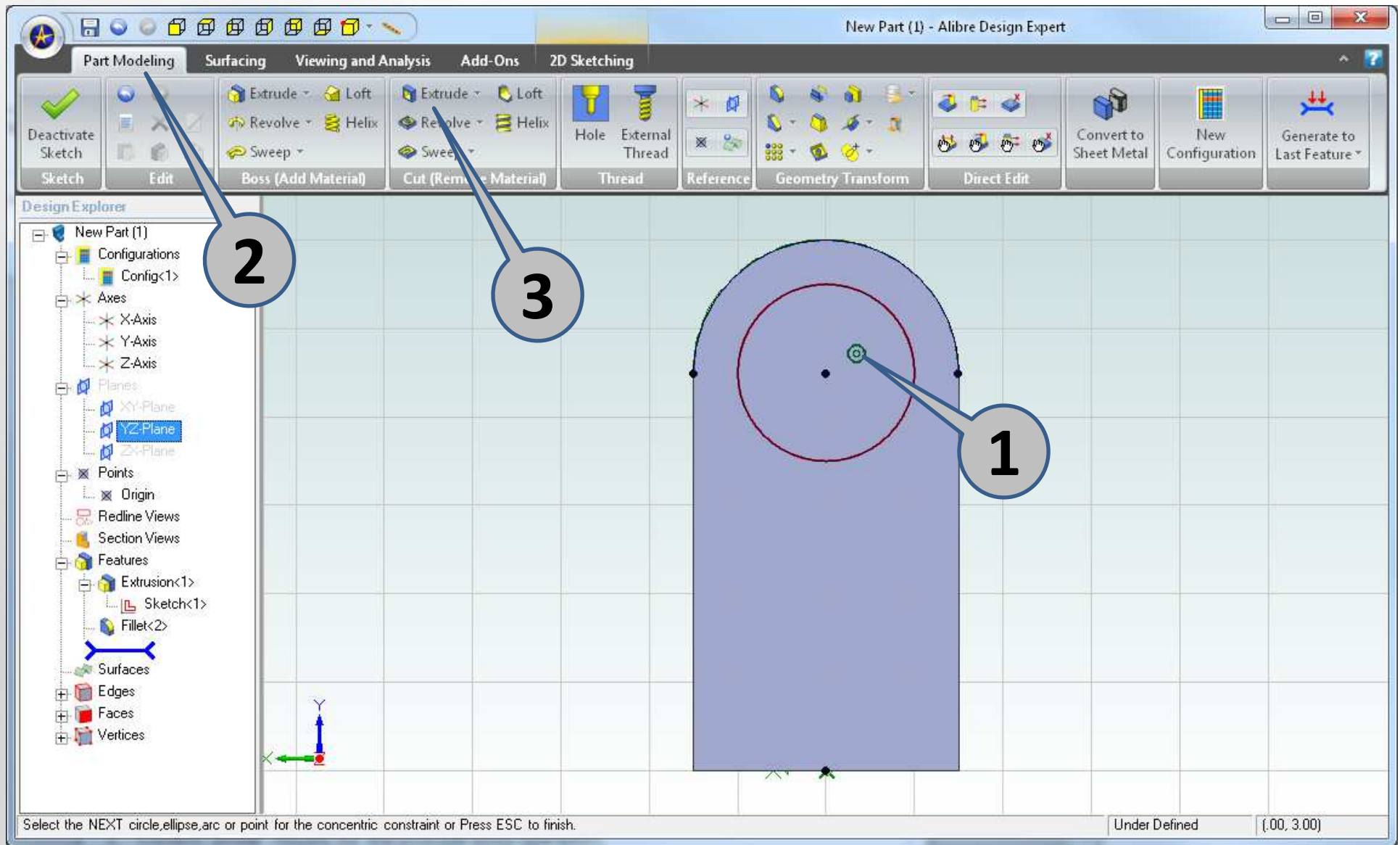
1. Click on the **Circle** tool.



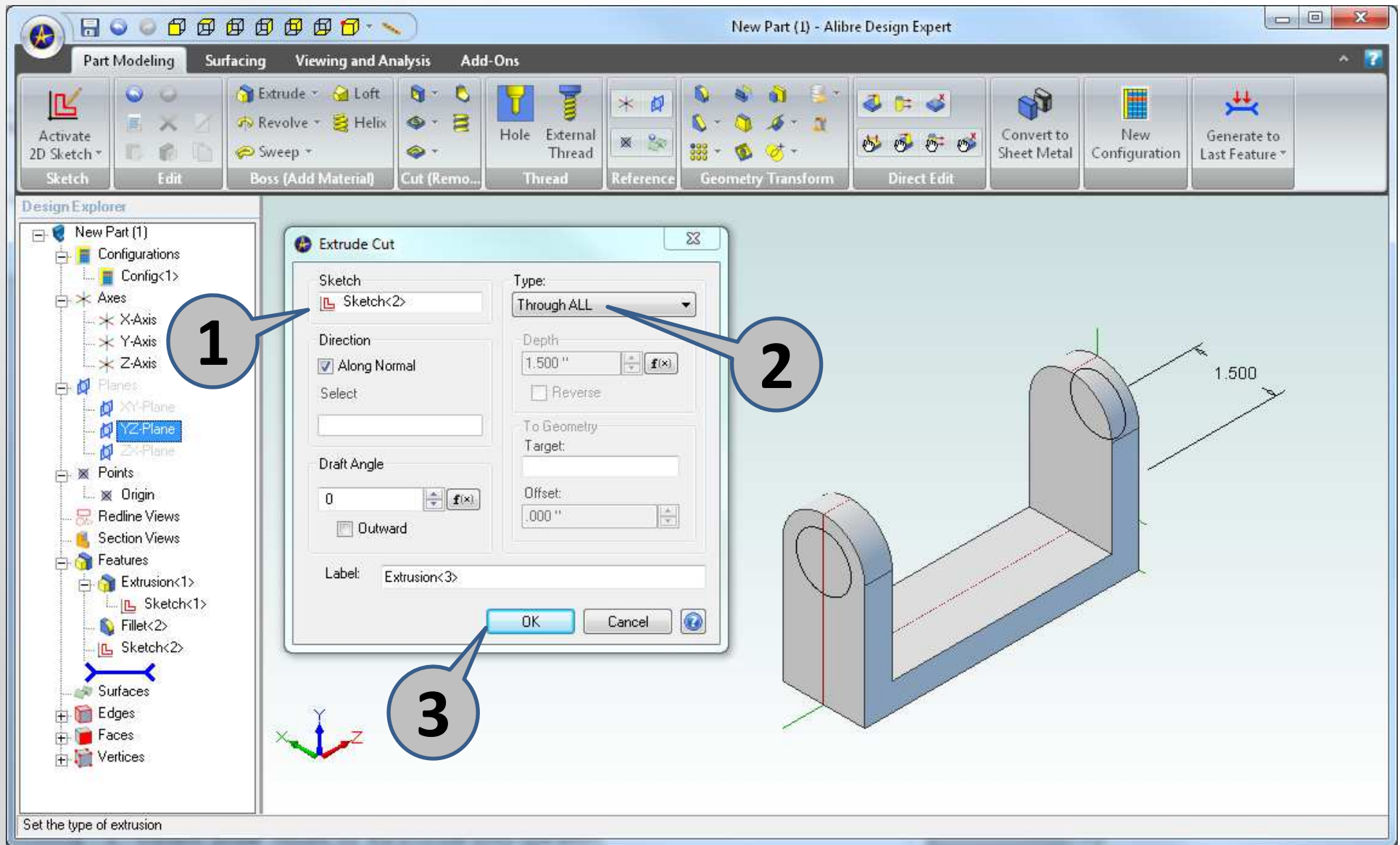
1. Move the mouse cursor to the position shown. *Click and release* to create the center point of the circle.
2. Move the mouse cursor to the right a distance of one grid square. *Click and release* to complete the circle.



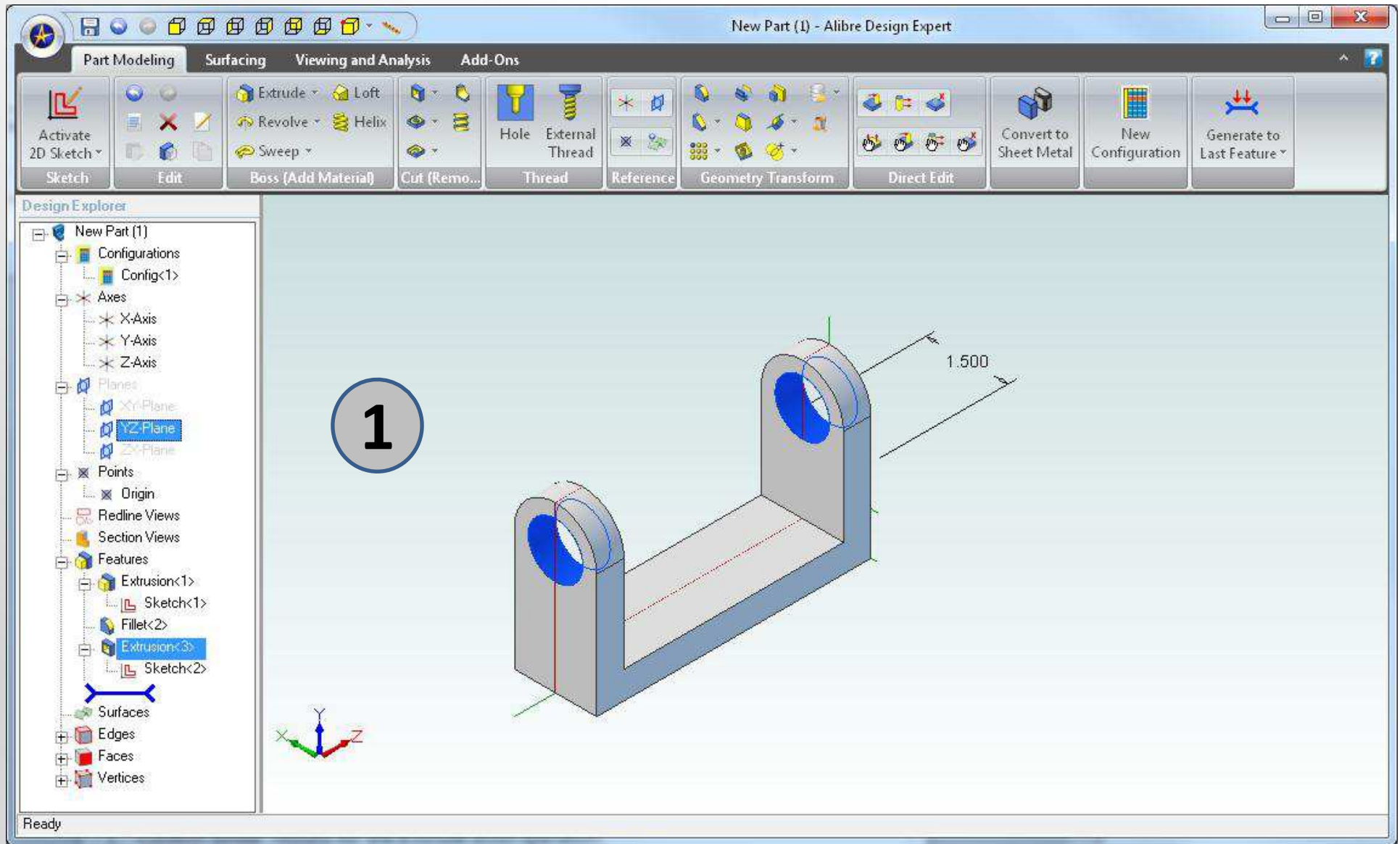
1. Select the **Concentric** option.
2. Click on the perimeter of the circle.
3. Click on the circular edge indicated.
4. Alibre Design will often tell you what it expects.



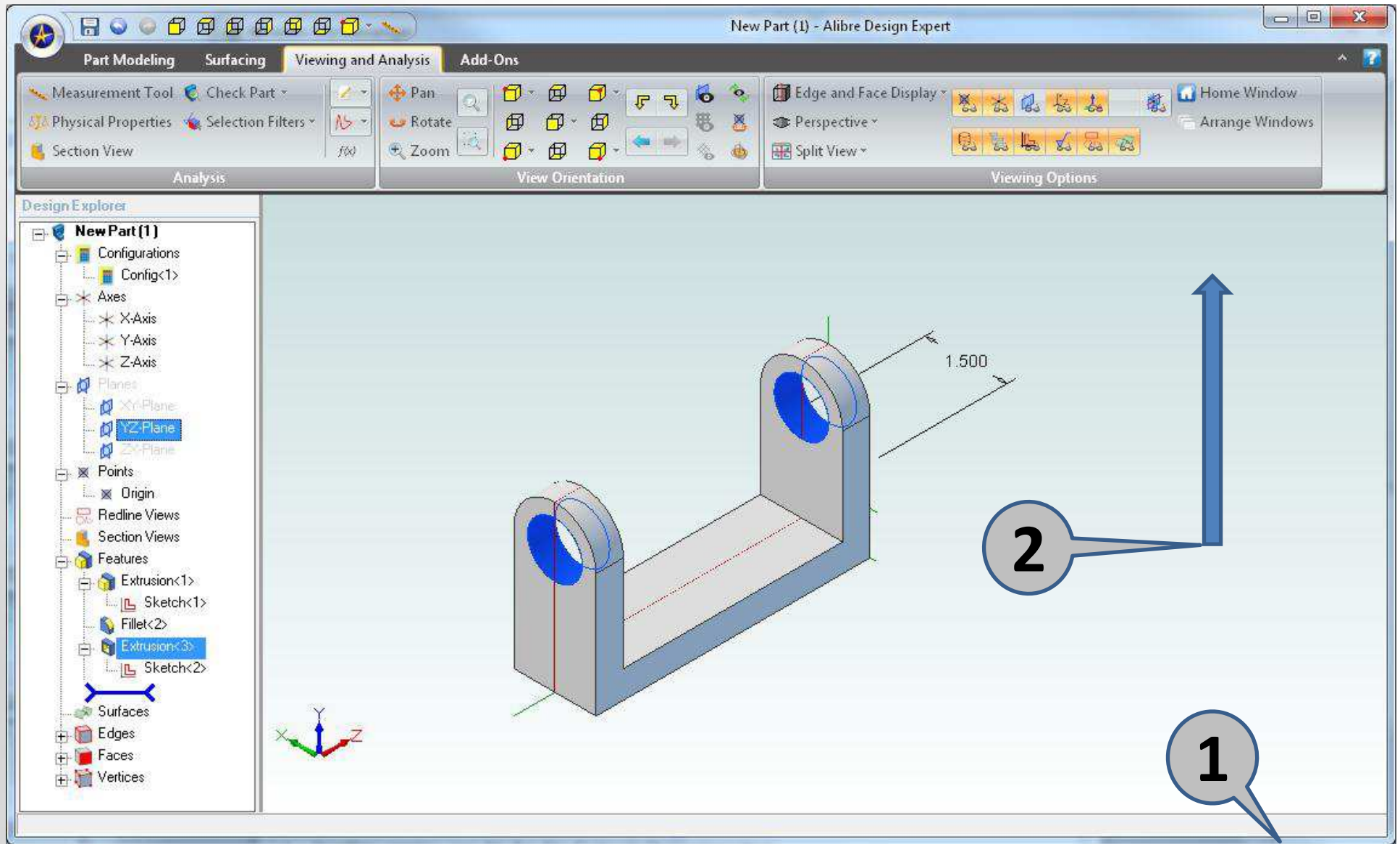
1. Confirm similar results for the **Concentric** constraint. Take note of the **Concentric** constraint symbol that appears near the circle's center.
2. Click on the **Part Modeling** tab.
3. Click on **Extrude Cut**.



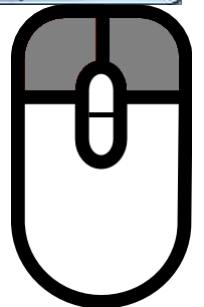
1. In the **Extrude Cut** dialog box, confirm that the **Sketch** entry box displays **Sketch<2>**.
2. Select **Through All** from the **Type** drop down menu.
3. Click **OK**.

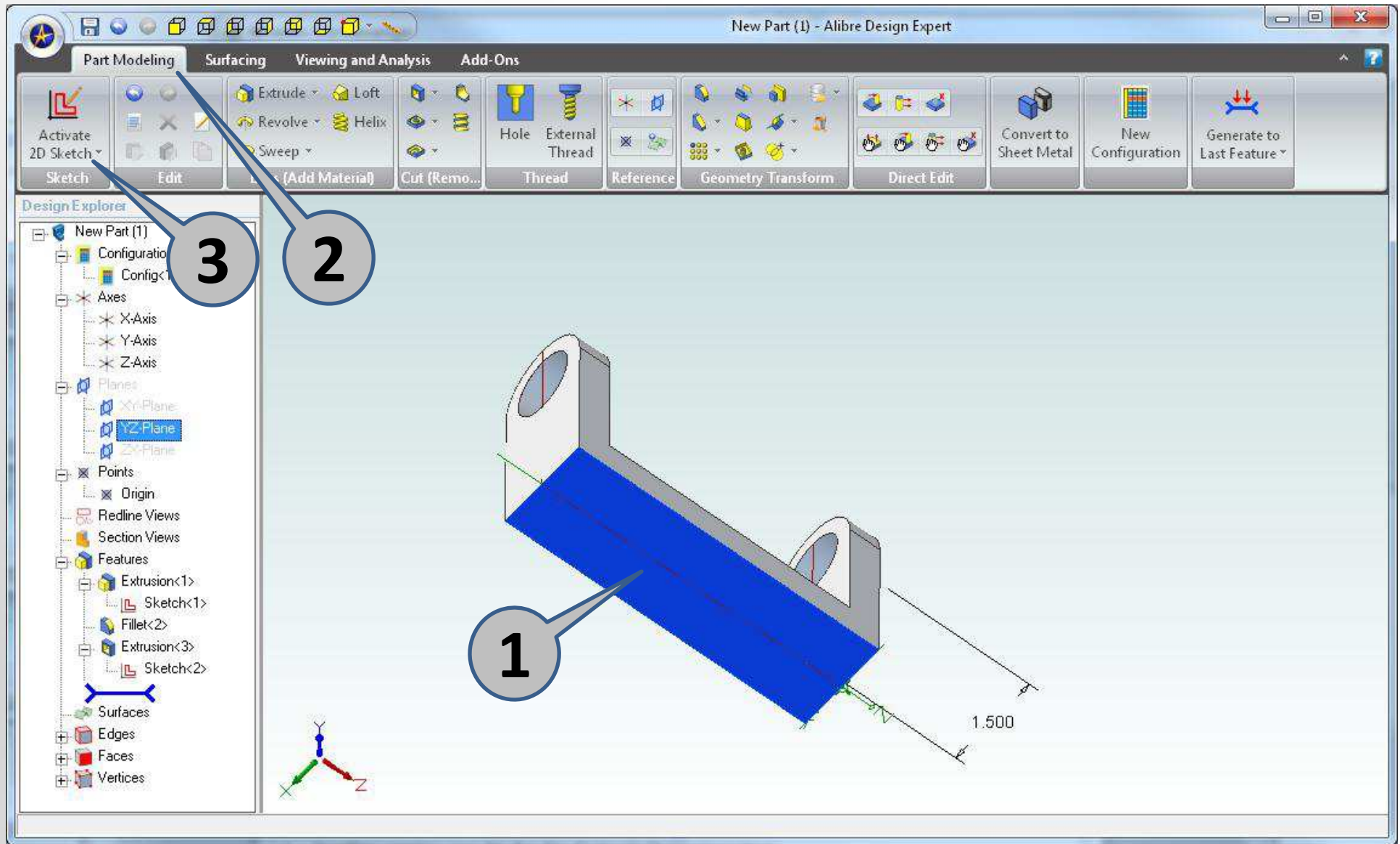


1. Confirm similar results for the **Extrude Cut** operation.

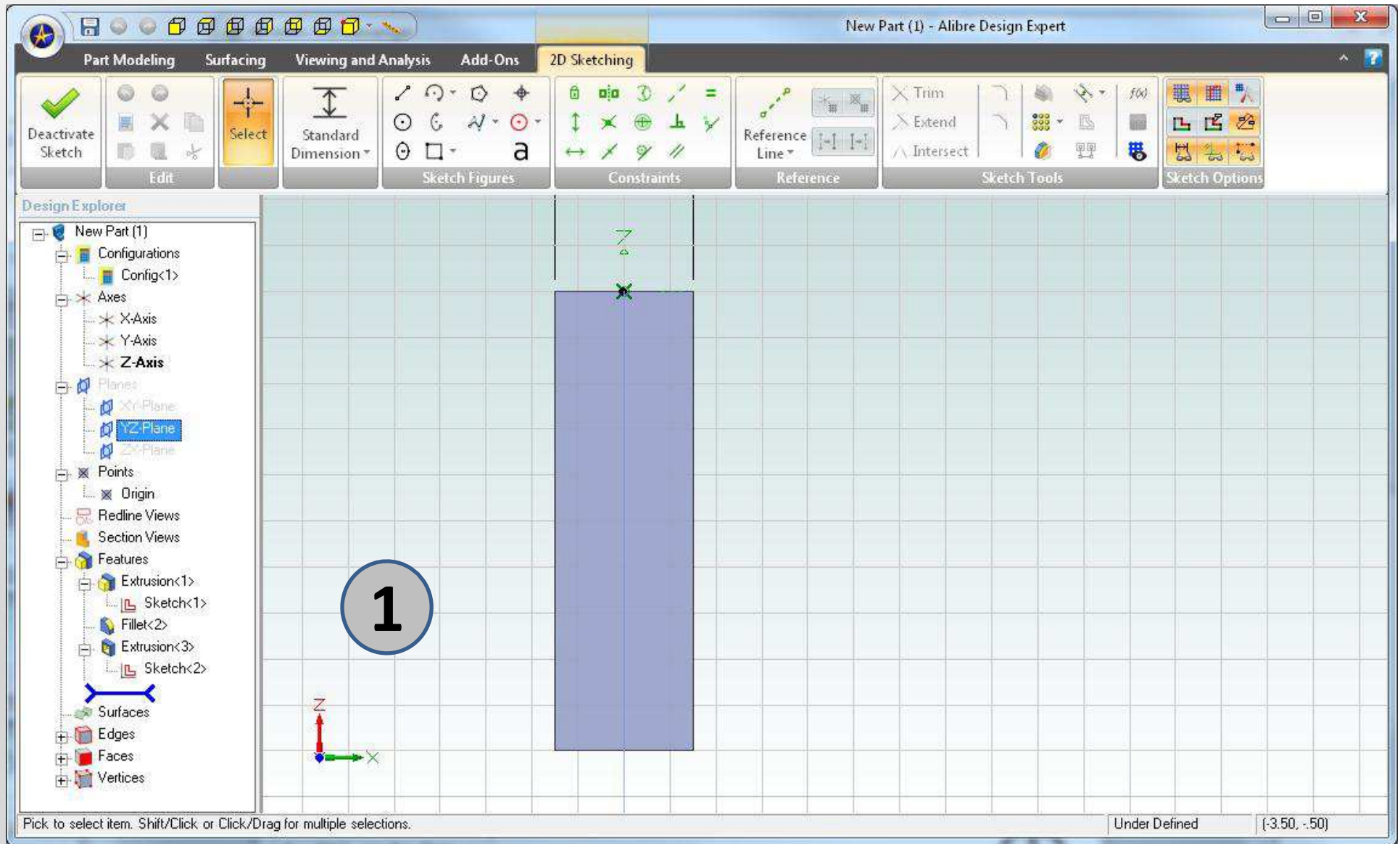


1. Rotate the model by holding down both of the mouse buttons and dragging.
2. *Click and hold* both mouse buttons while moving the mouse cursor upward (approximately as shown). Release the mouse buttons.

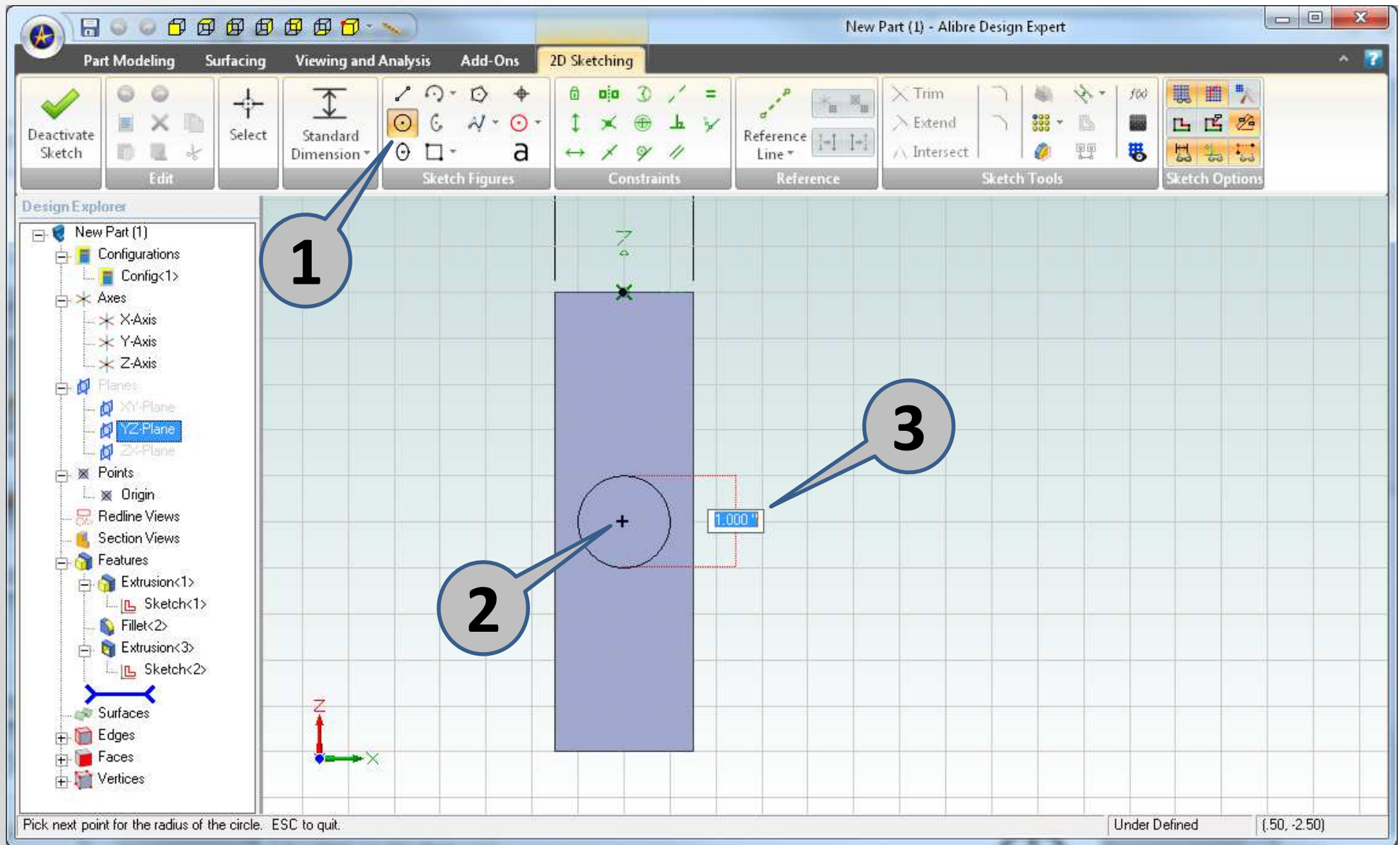




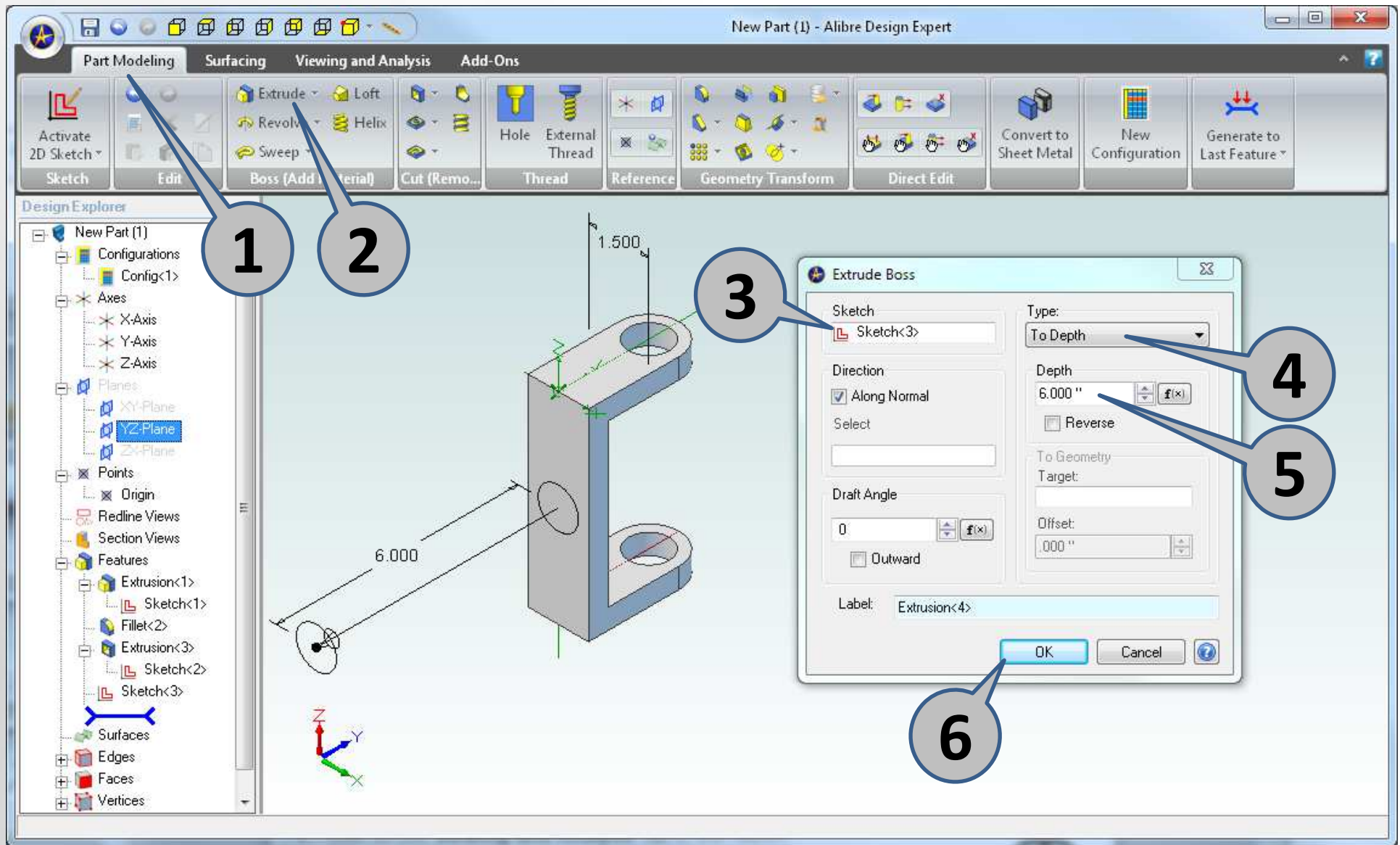
1. Click on the face indicated.
2. Click in the **Part Modeling** tab.
3. Click on **Activate 2D Sketch**.



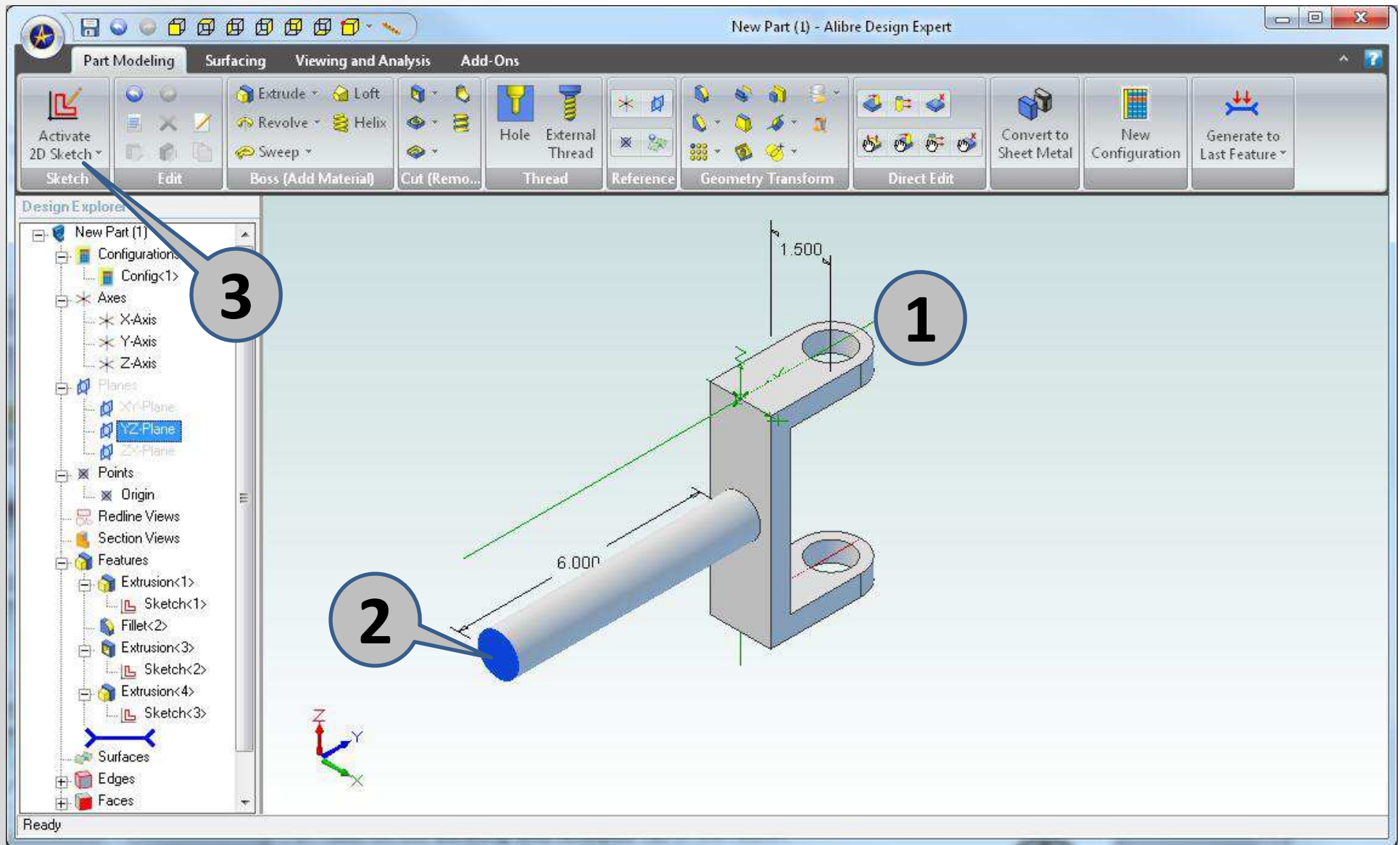
1. The view of the work area should be similar to the screen capture.



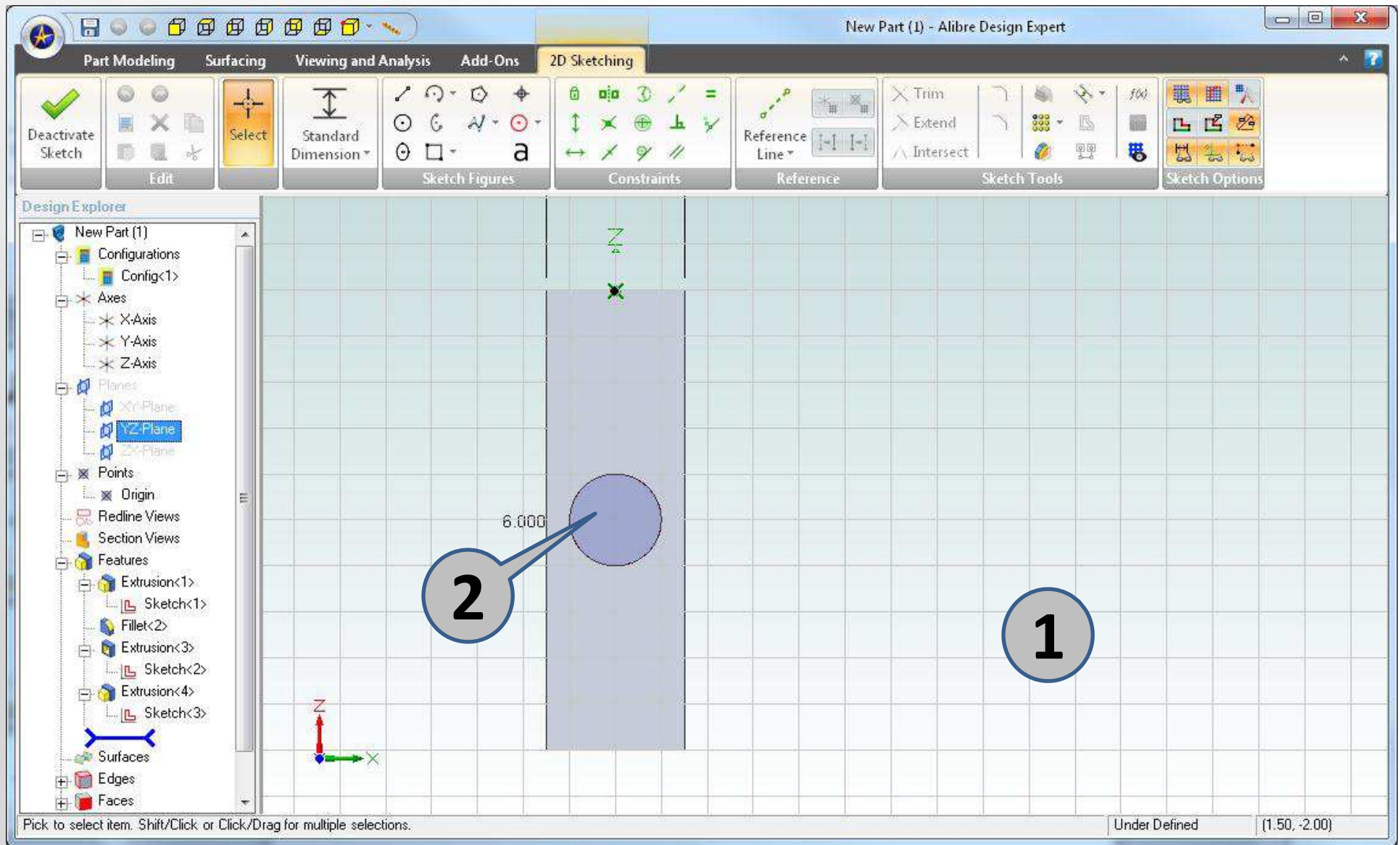
1. Click on the **Circle** tool.
2. Position the mouse cursor as shown. Count the grid squares to confirm the location as the center of the rectangular face in the background. *Click and release* to create the center of the circle.
3. Move the mouse cursor one grid square to the right. *Click and release* to complete the circle.



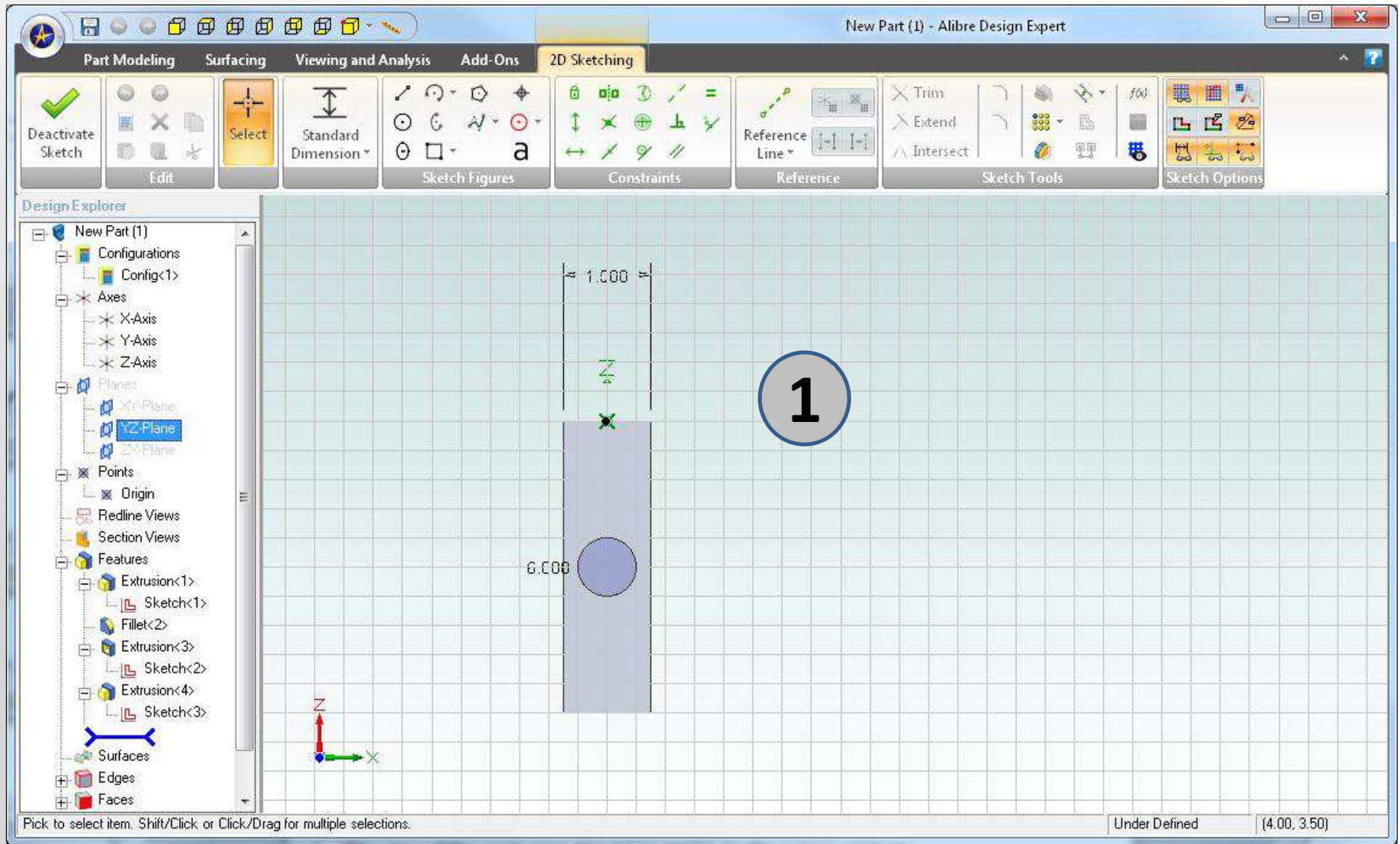
1. Click on the **Part Modeling** tab in the ribbon.
2. Click on the **Extrude Boss** icon.
3. Confirm that **Sketch<3>** is the **Sketch** to be extruded.
4. Confirm that **To Depth** is the **Type** of extrusion.
5. Enter a value of **6.0 inches** for the **Depth**.
6. Click **OK**.



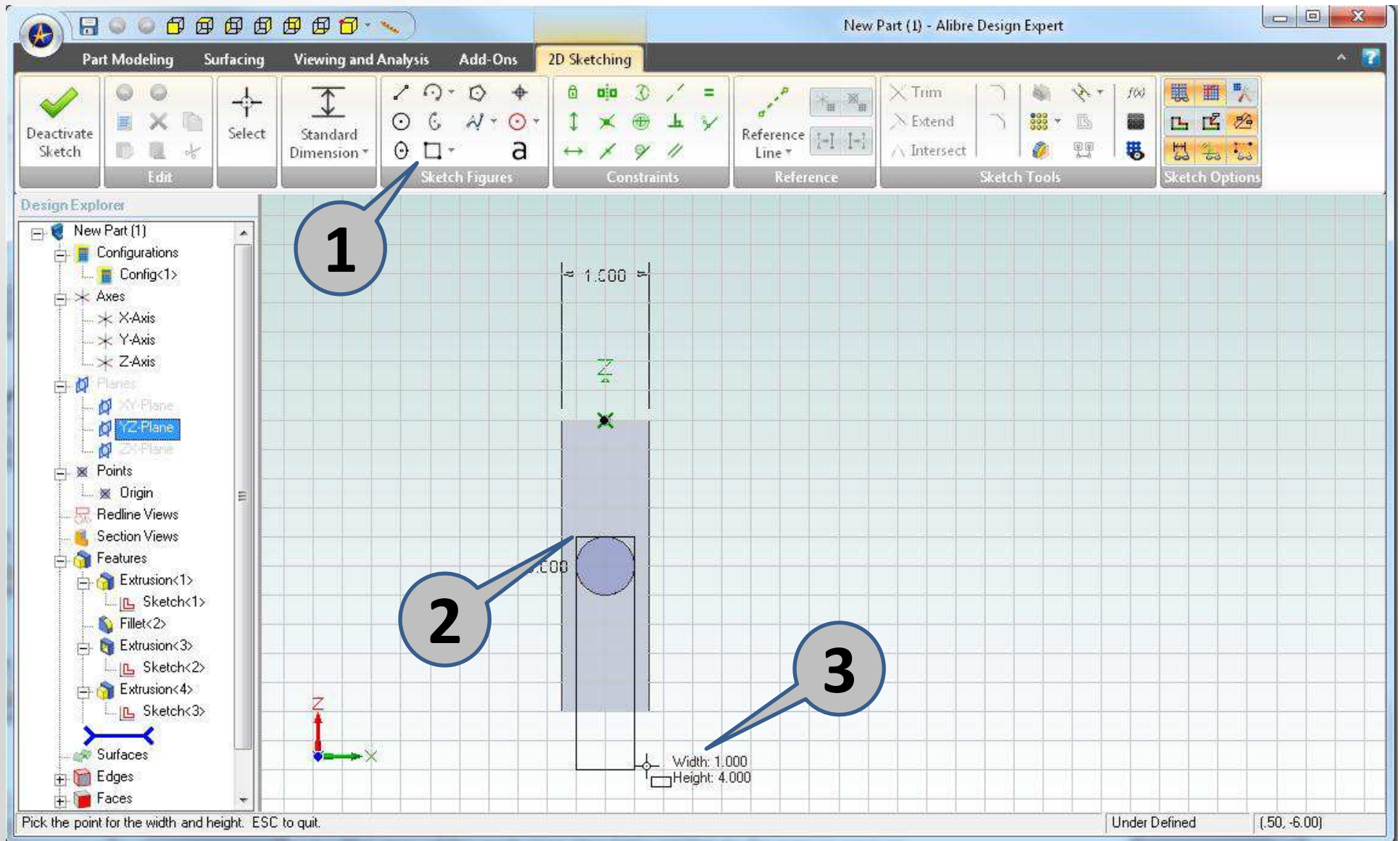
1. Confirm similar results for the **Extrude Boss** operation.
2. Click on the face indicated.
3. Click on **Activate 2D Sketch**.



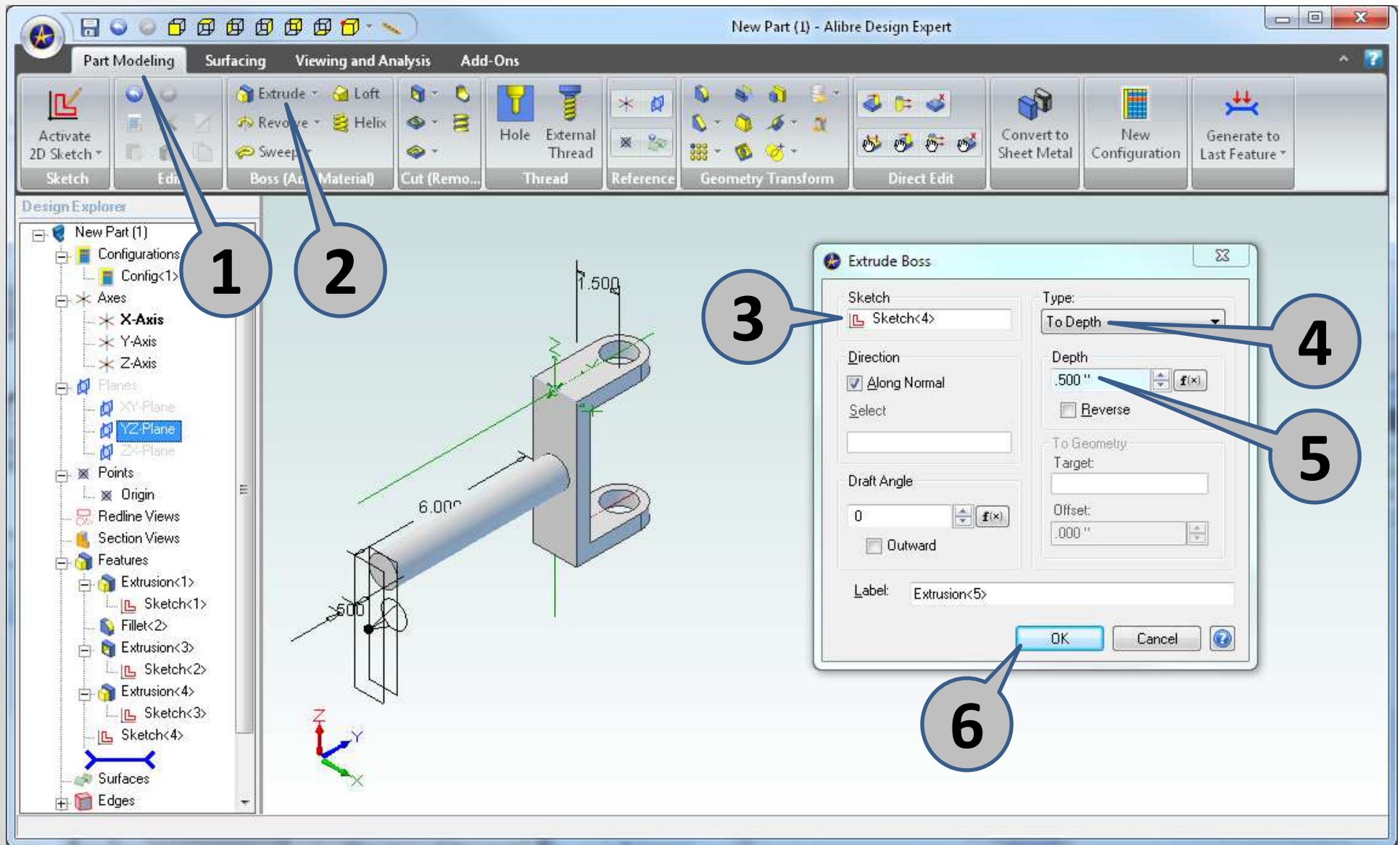
1. The view of the work area should be similar to the screen capture.
2. Position the mouse cursor over the visible, circular face. Roll the mouse wheel backwards to zoom out until the visible geometry is roughly half of its original size.



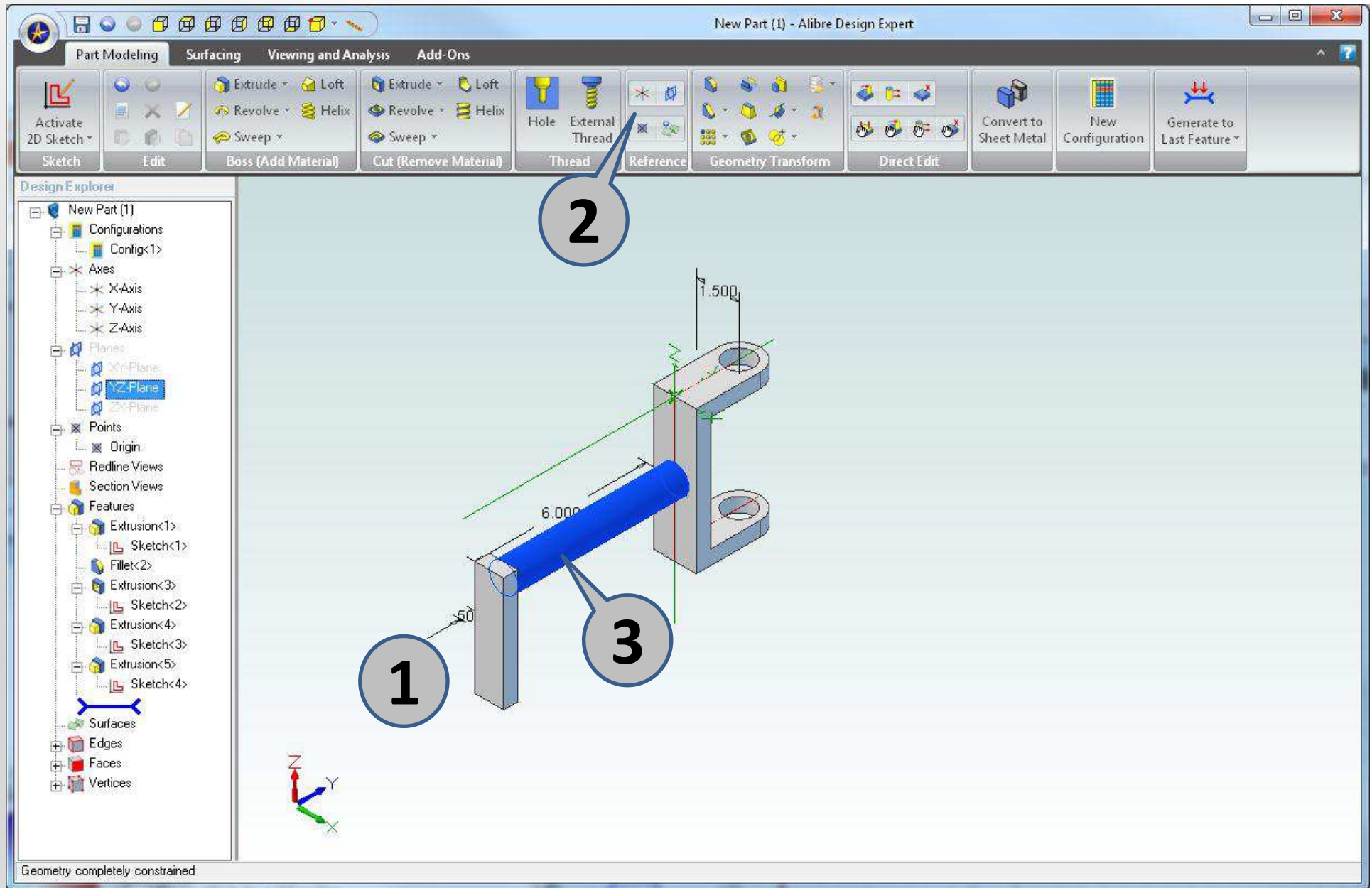
1. The view of the work area should be similar to the screen capture.



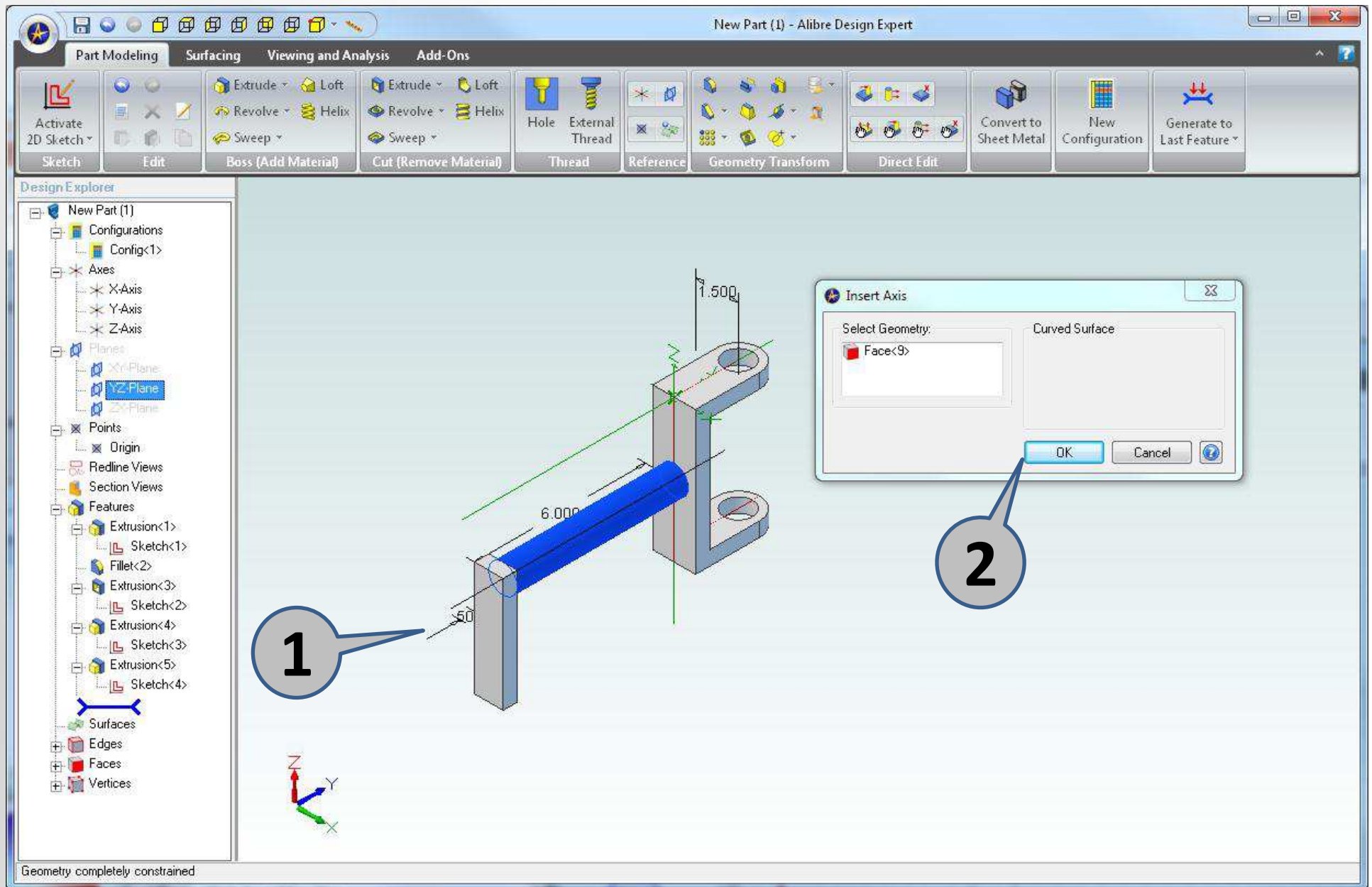
1. Click on the **Rectangle by Two Corners** tool.
2. Position the mouse cursor as shown. *Click and release* to create the first corner of the rectangle.
3. Move the mouse cursor downwards and to the right until the measurement indicator attached to the cursor displays **Width: 1.000** and **Height: 4.000**. *Click and release* to complete the rectangle.



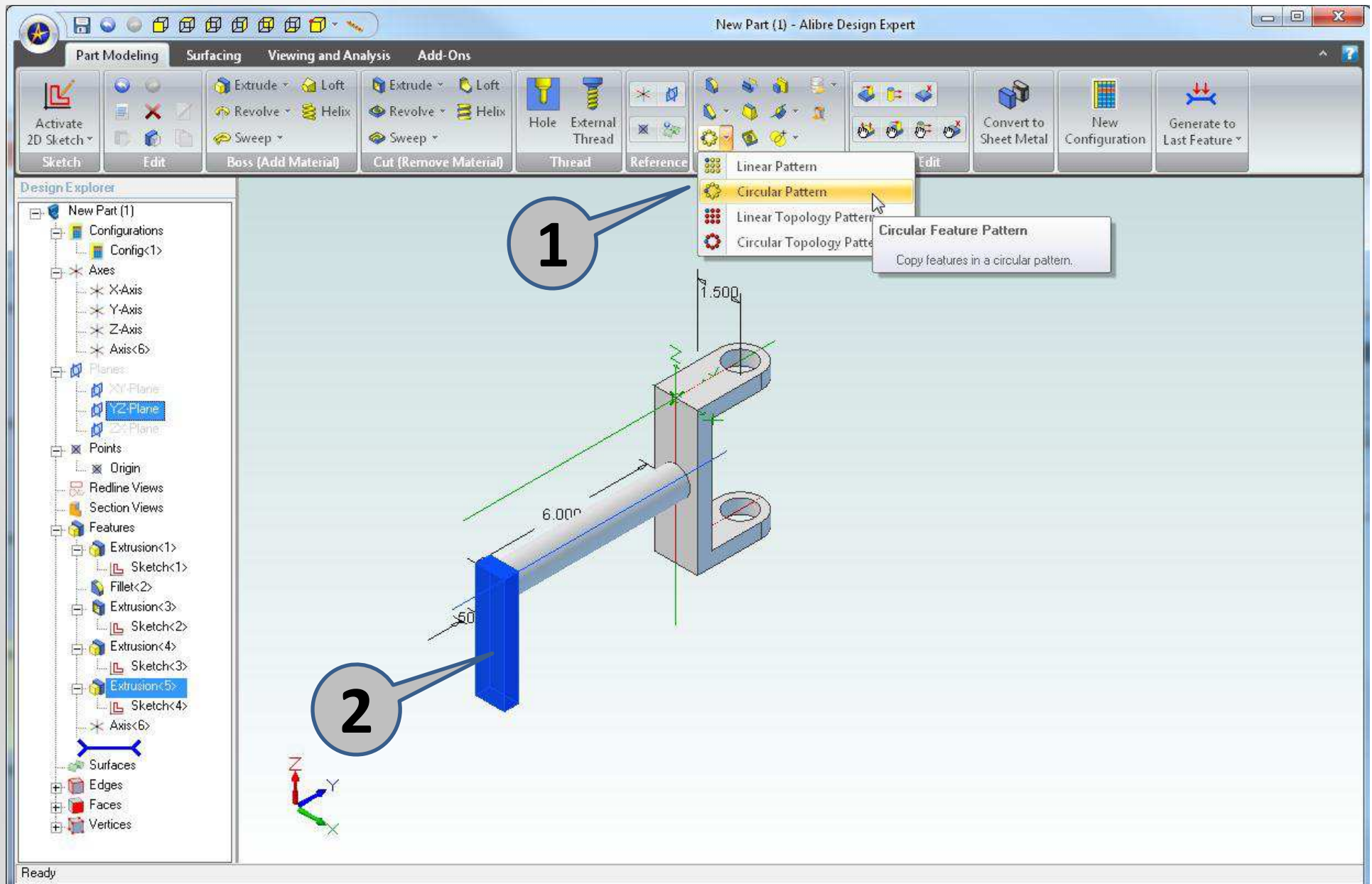
1. Click on the **Part Modeling** tab in the ribbon.
2. Click on **Extrude Boss**.
3. Confirm that **Sketch<4>** is the **Sketch** to be extruded.
4. Confirm that **To Depth** is the **Type** of extrusion.
5. Enter a value of **0.5 inch** for the **Depth**.
6. Click **OK**.



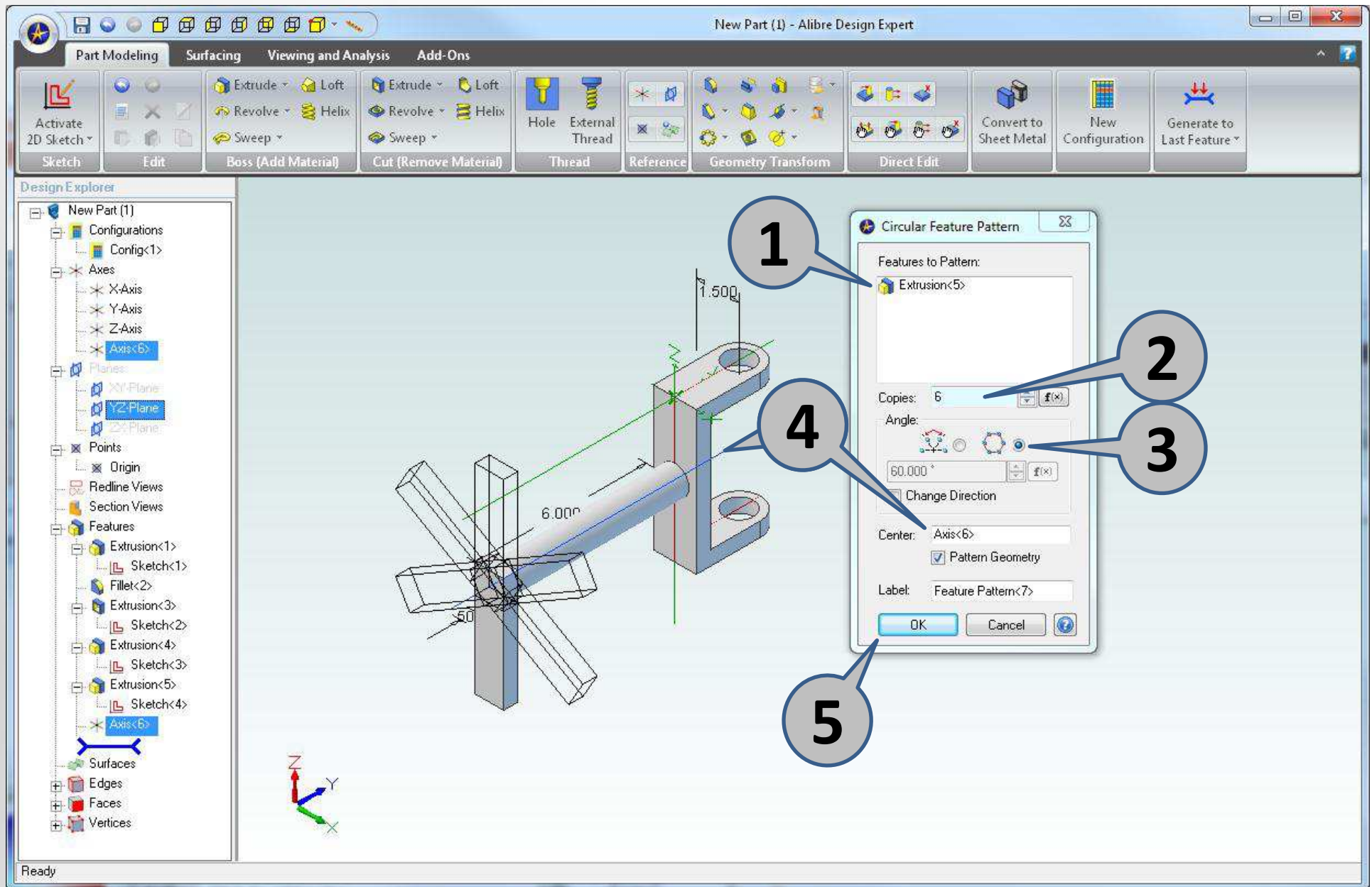
1. Confirm similar results for the **Extrude Boss**.
2. Select the **Insert Reference Axis** icon.
3. Click on the face indicated in the screen capture: the cylindrical face of the shaft.



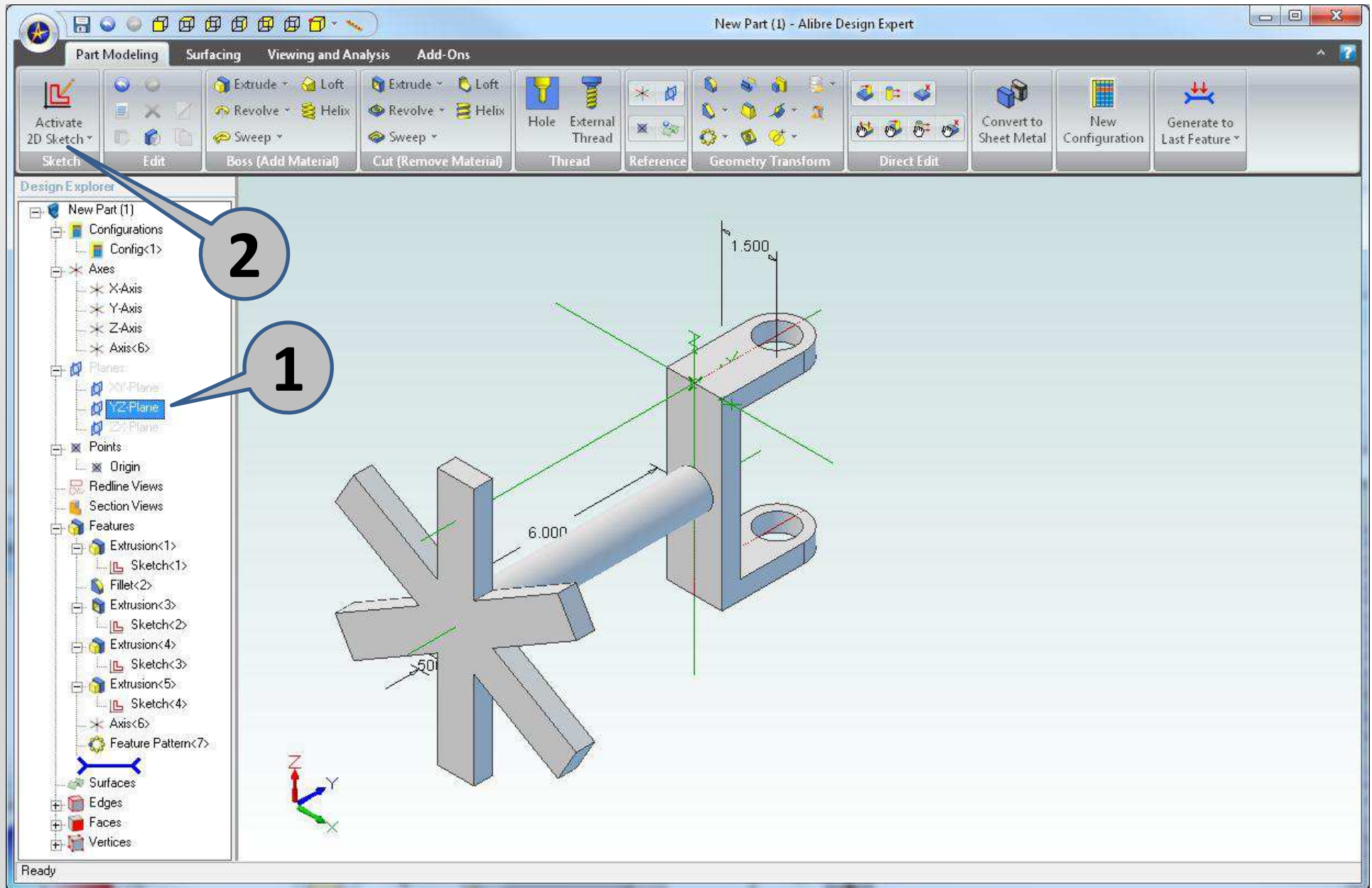
1. A preview of the **Axis** is displayed.
2. Click **OK**.



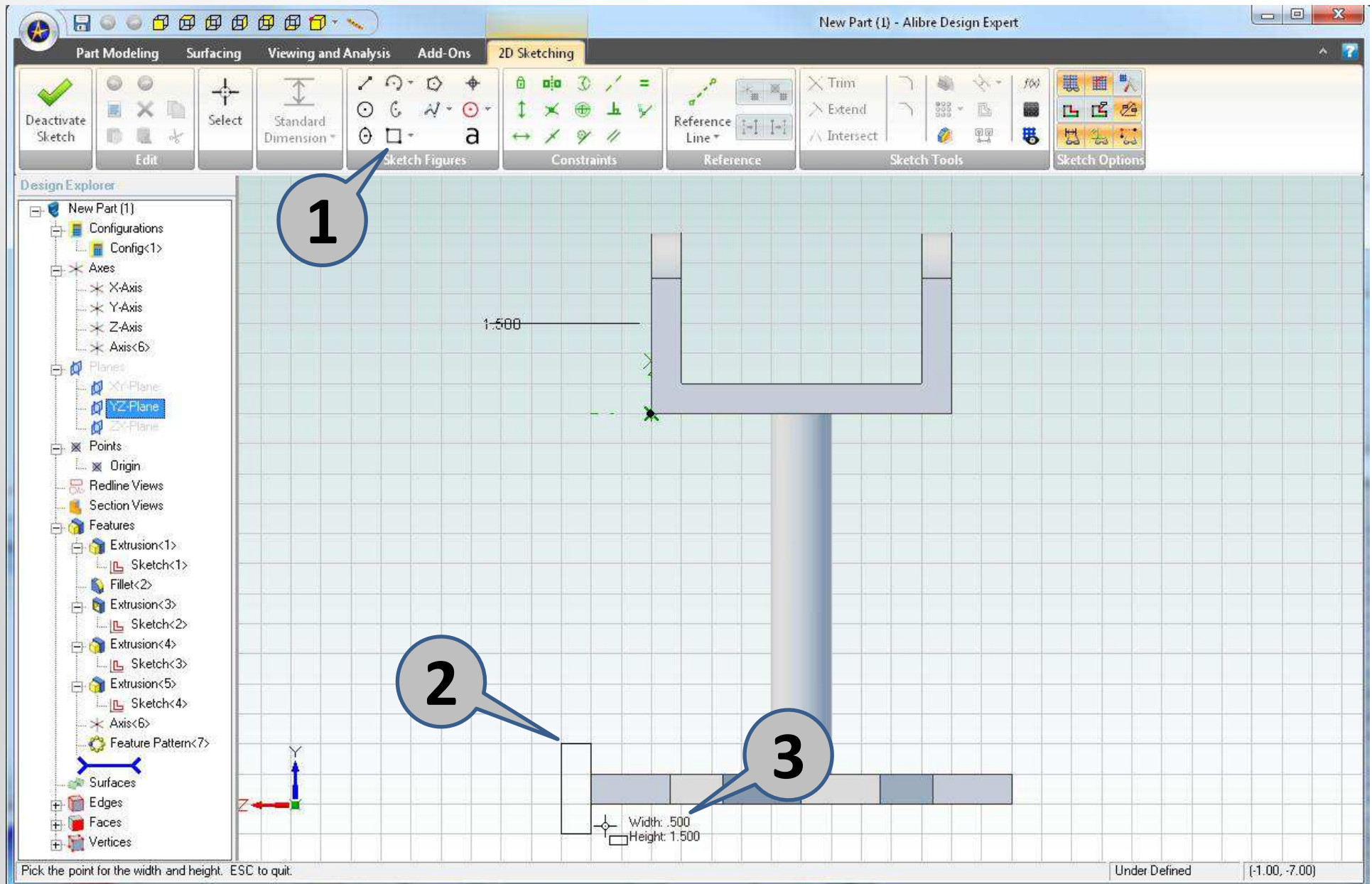
1. Click on the **Feature Pattern** sub-menu and select the **Circular Pattern** option.
2. Click on the face indicated.



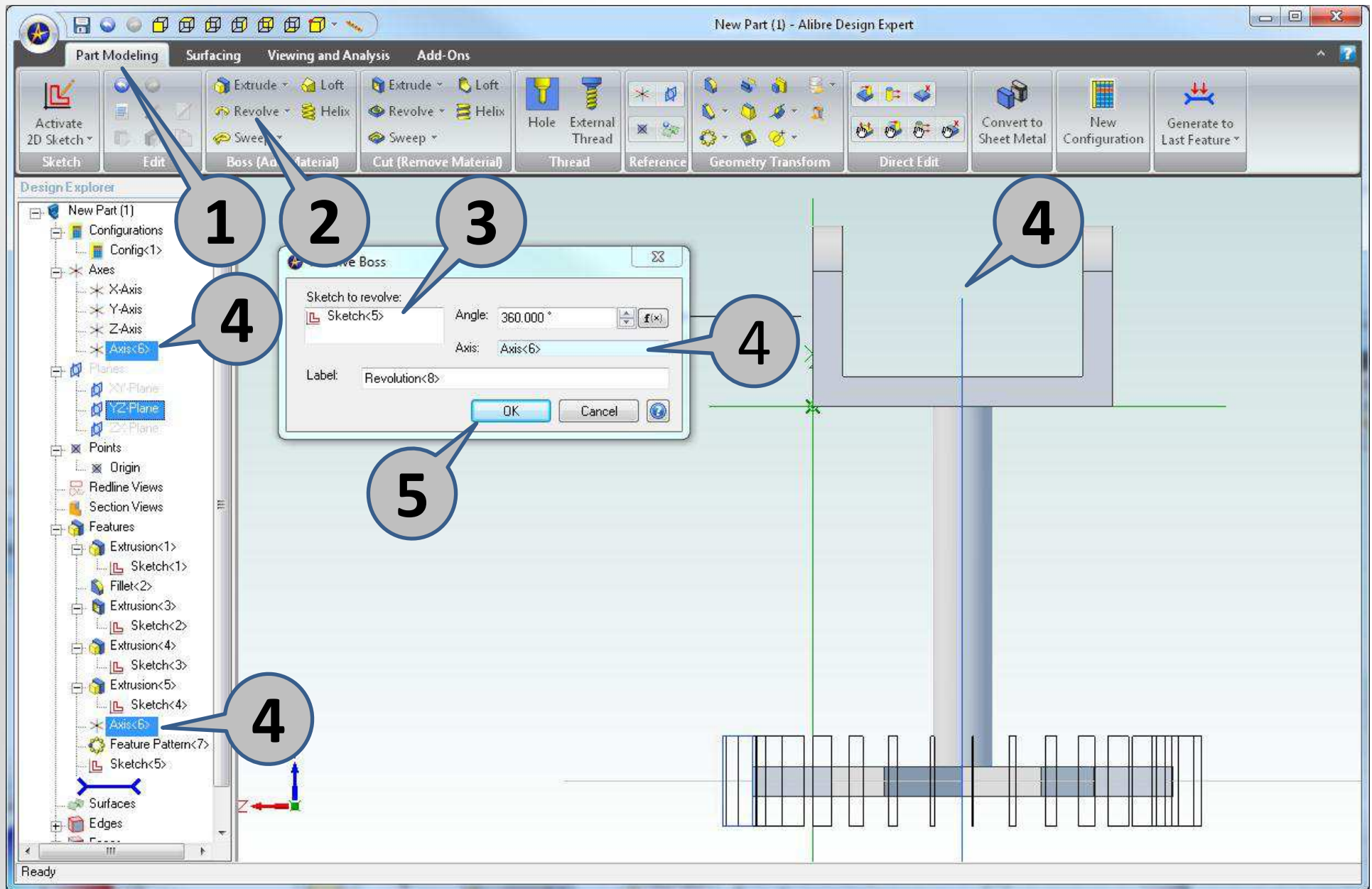
1. Confirm that **Extrusion<5>** appears in the **Features to Pattern:** entry box.
2. In the **Copies:** entry box, enter **6**.
3. Select the **Complete Revolution** option.
4. Click on **Axis 6** in the **Design Explorer**.
5. Click **OK**.



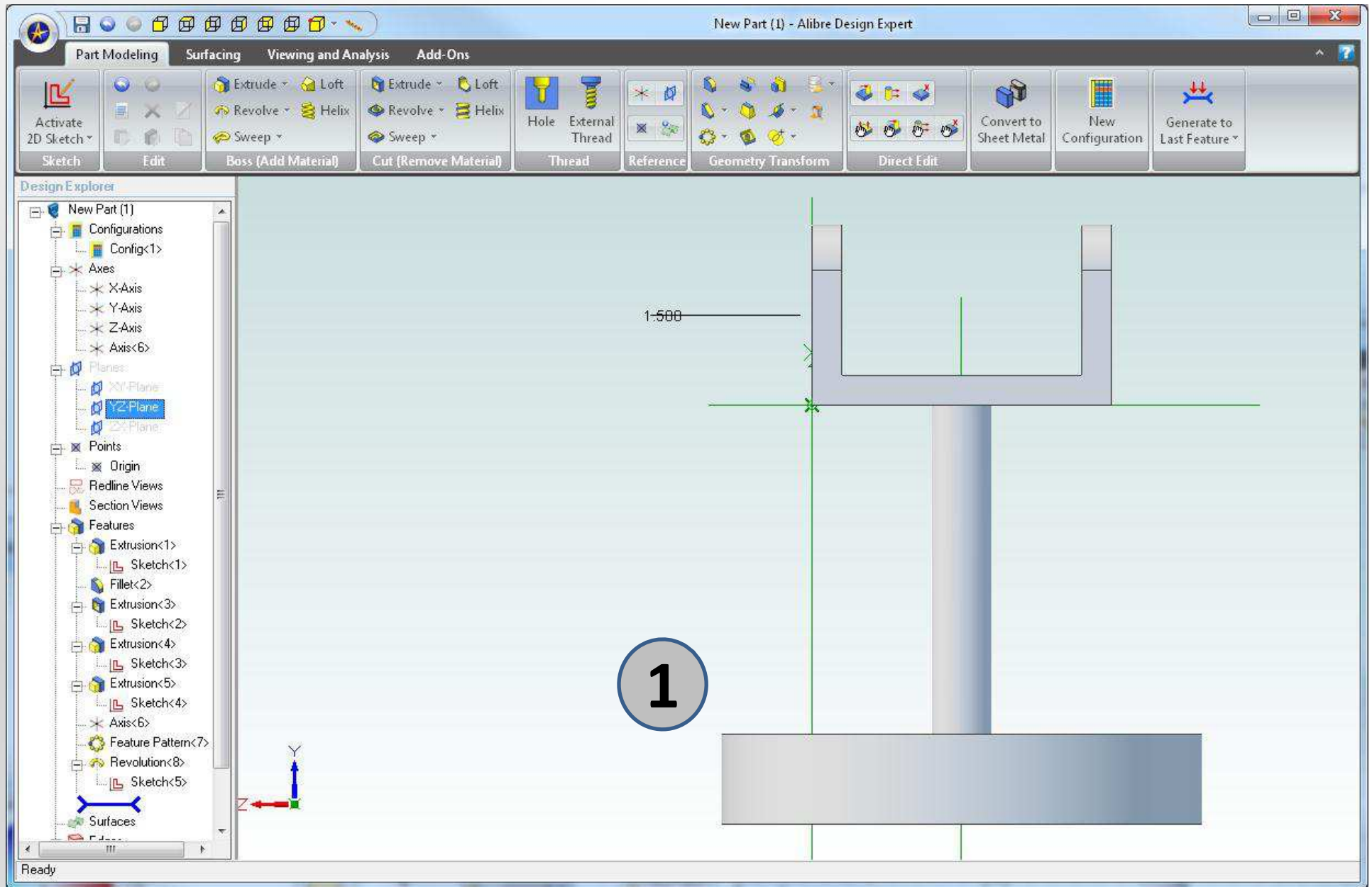
1. Click on the **YZ Plane** in the **Design Explorer**.
2. Click on **Activate 2D Sketch**.



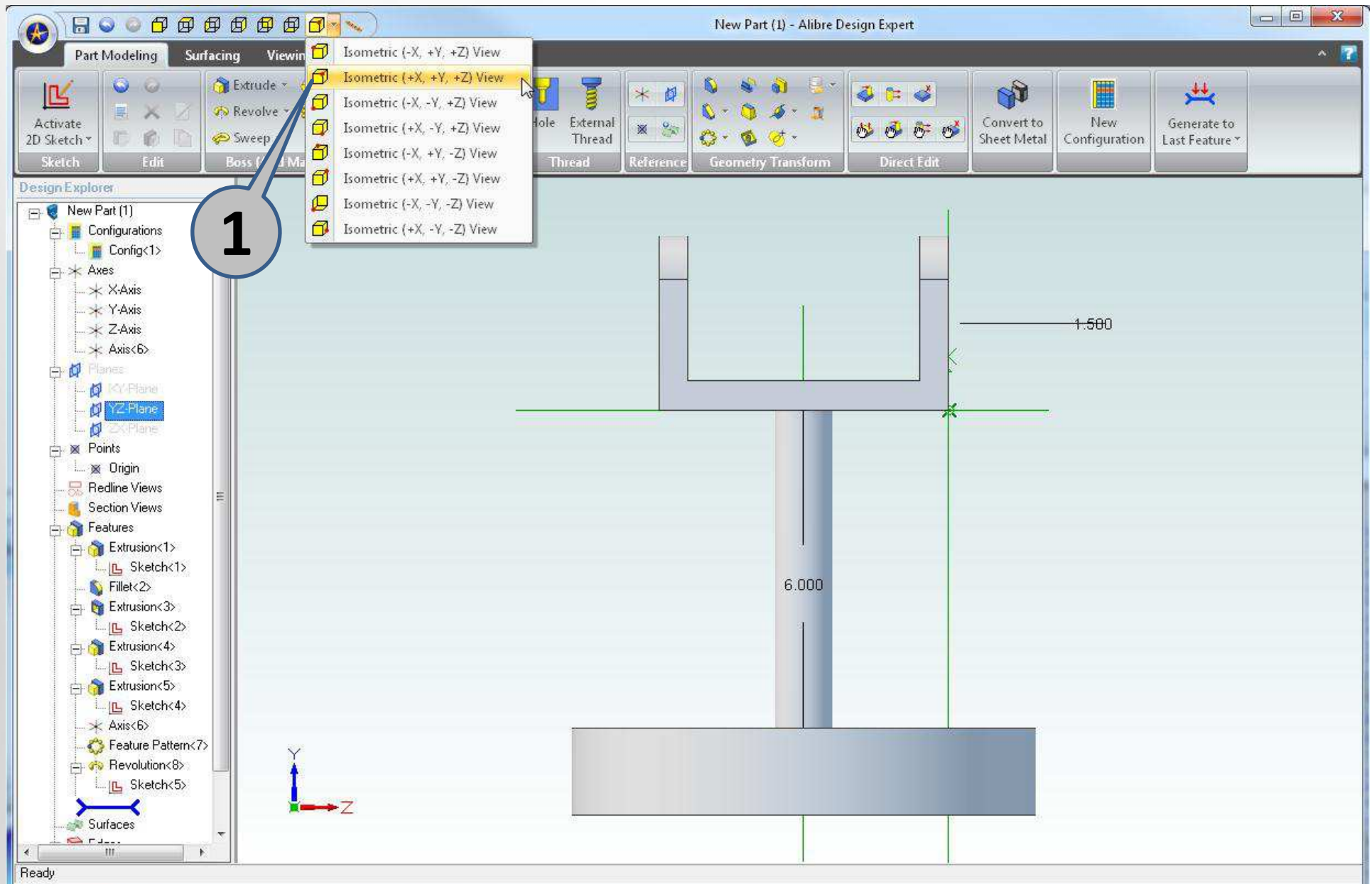
1. Click on the **Rectangle by Two Corners** tool.
2. Position the mouse cursor as shown. *Click and release* to create the first corner.
3. Move the mouse to the position indicated. *Click and release* to complete the rectangle.



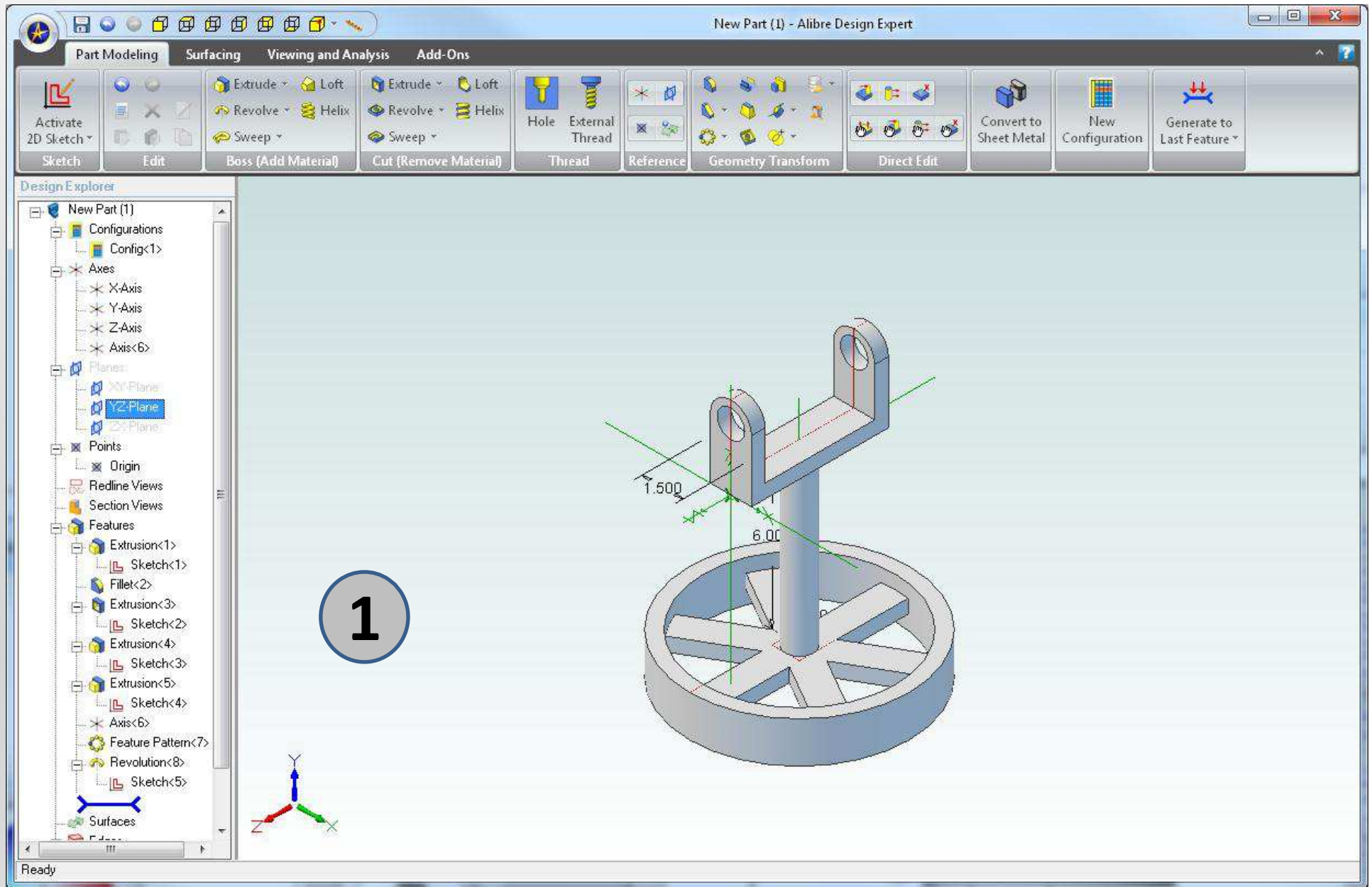
1. Click on the **Part Modeling** tab in the ribbon.
2. Click on the **Revolve Boss** tool.
3. Confirm that **Sketch<5>** appears in the **Sketch to revolve:** entry box.
4. Click in the **Axis** entry box and select **Axis 6**.
5. Click **OK**.



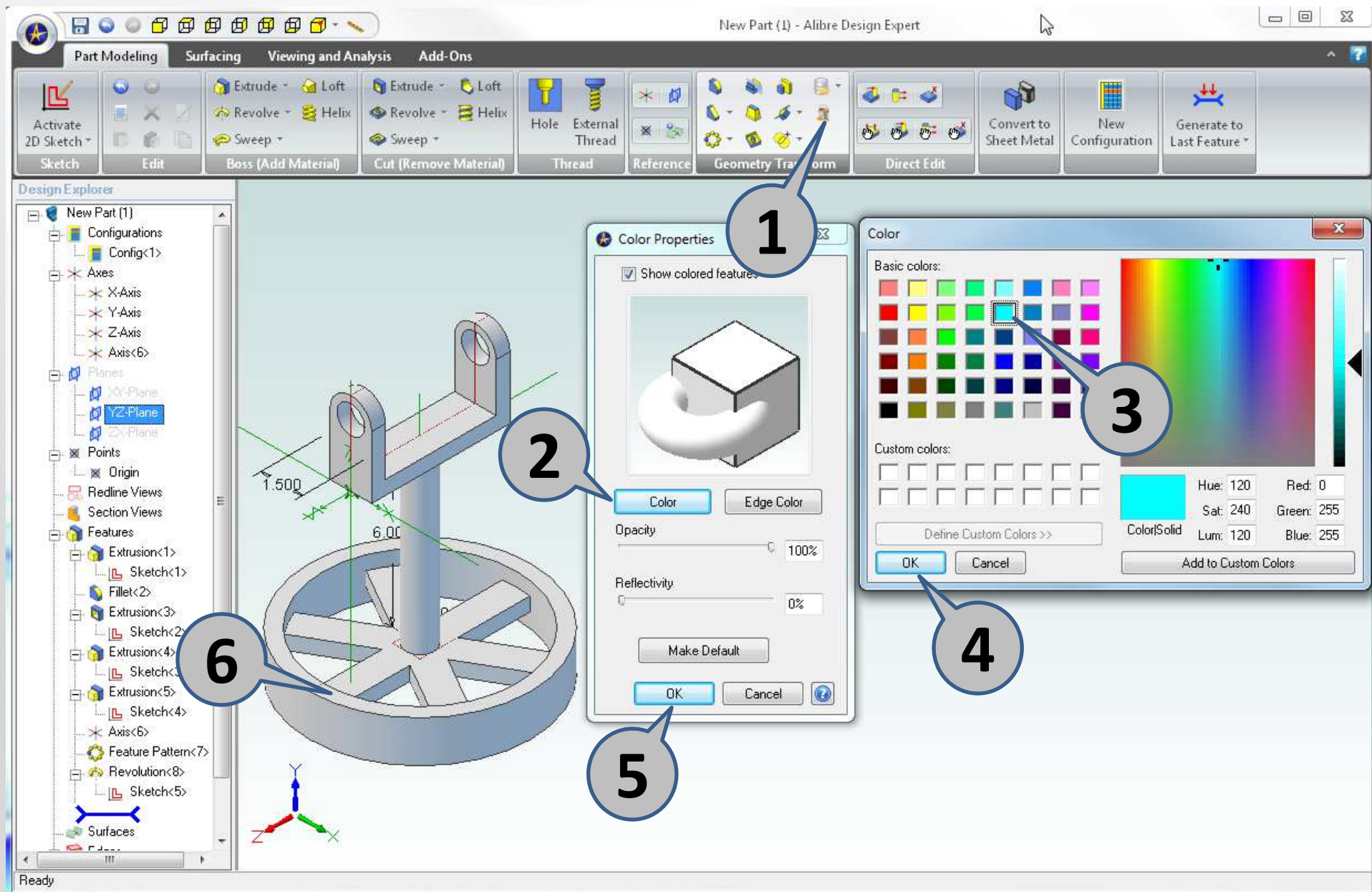
1. Confirm similar results for the **Revolve Boss** operation.



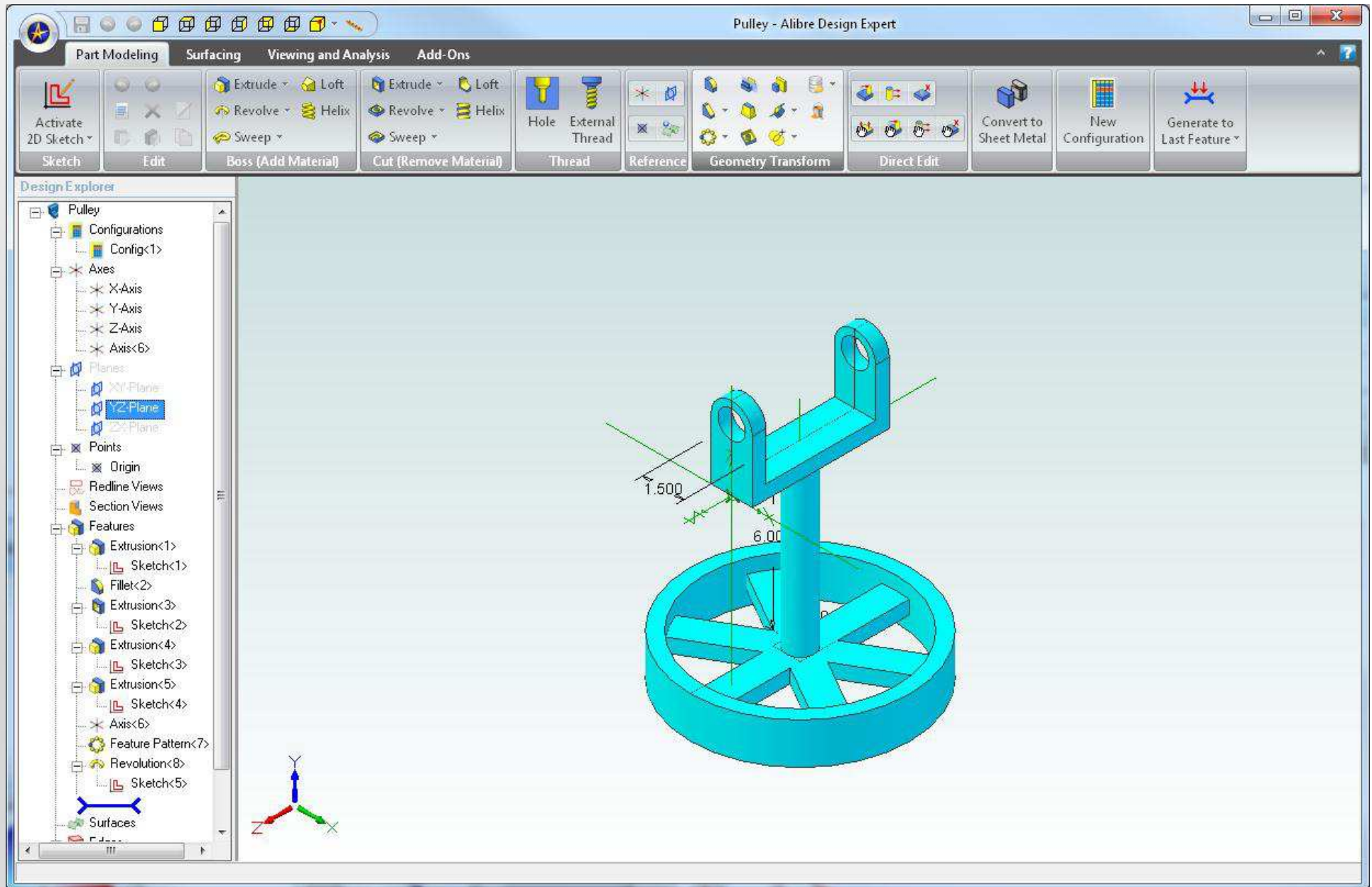
1. From the Quick Start menu select Isometric (+X, +Y, +Z) View.



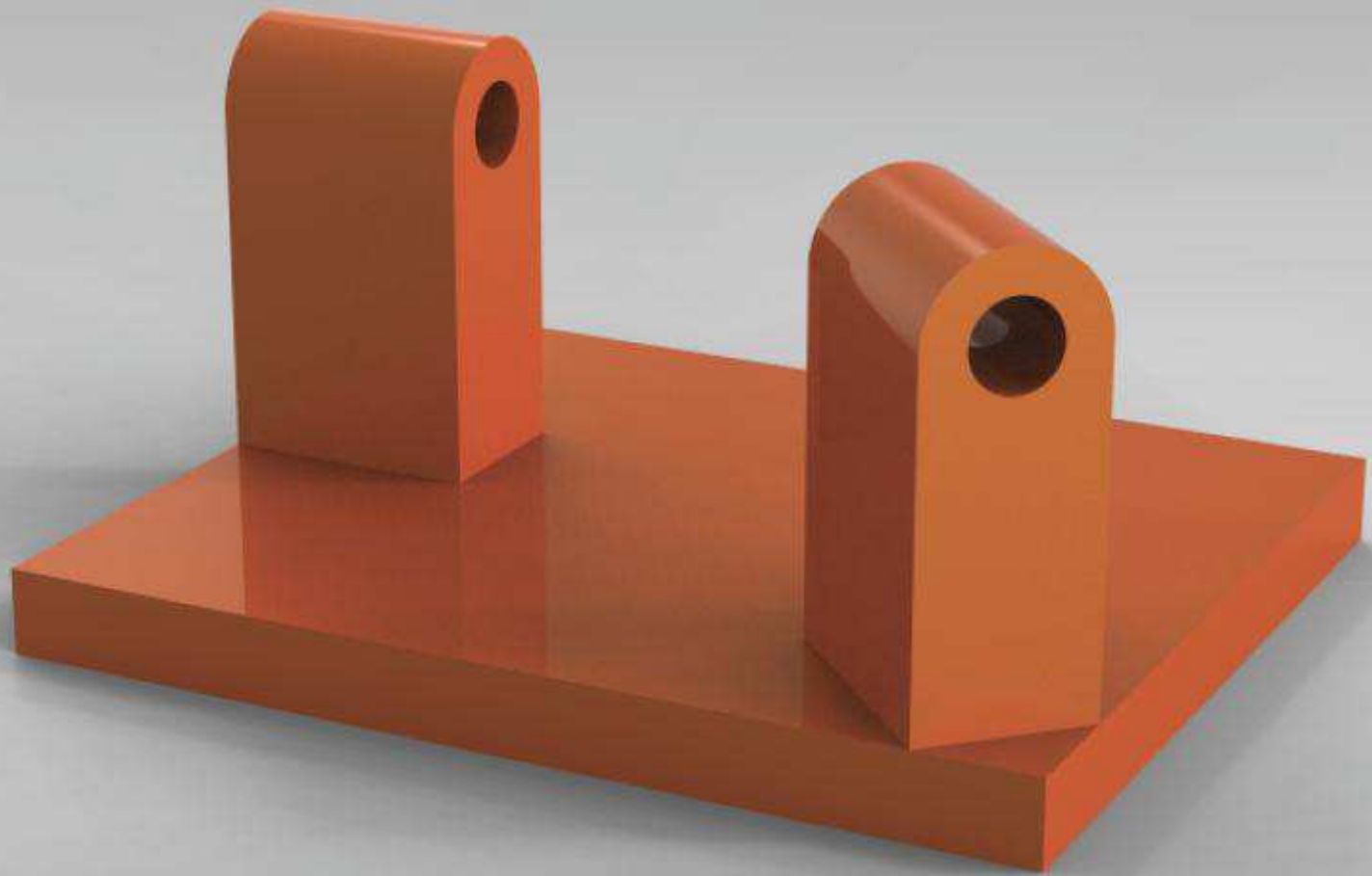
1. Confirm similar results for the view re-orientation.



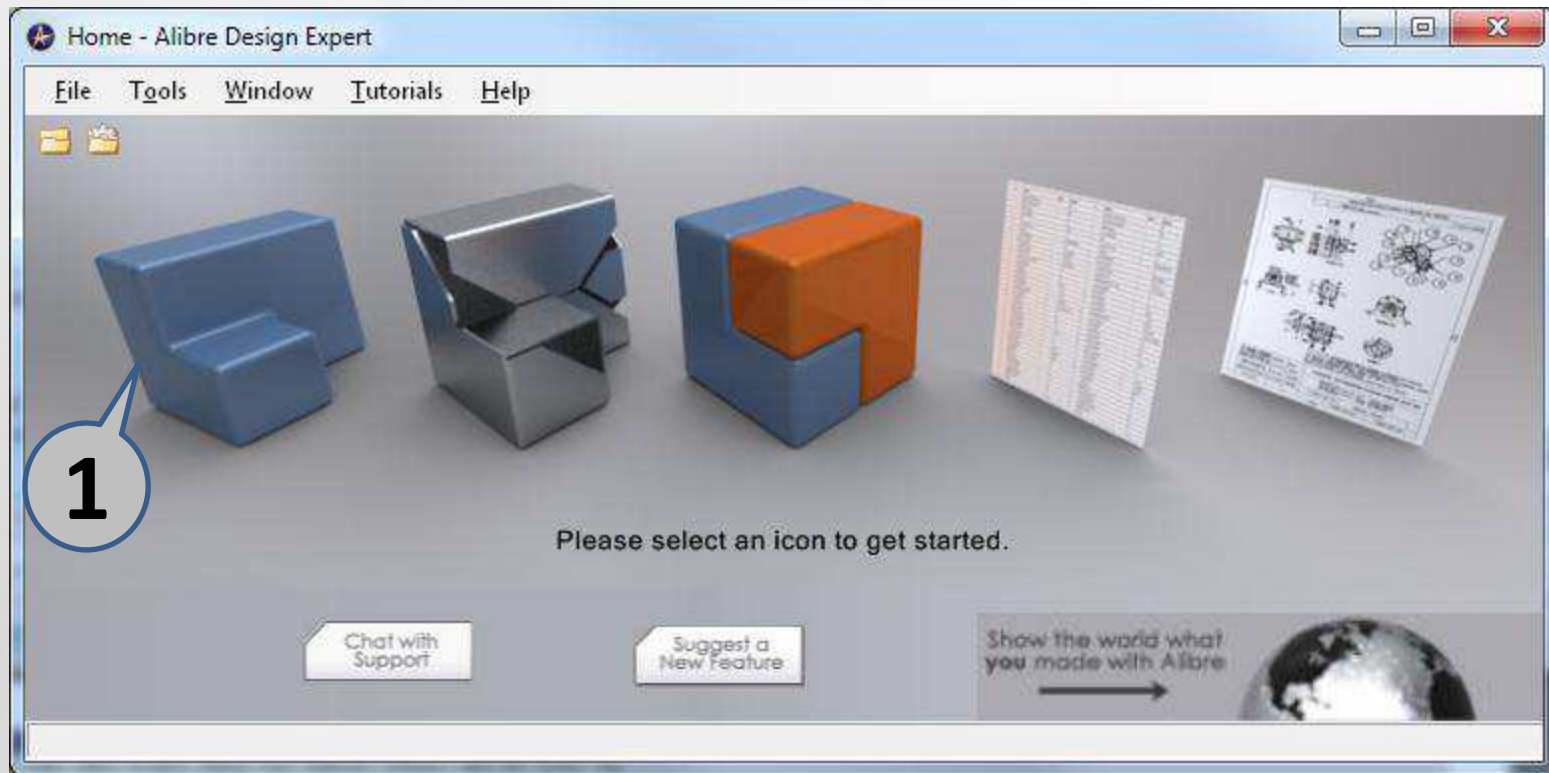
1. Select the **Part Color** icon.
2. Click on the **Color** button in the **Color Properties** dialog box.
3. Select a light blue color in the **Color** dialog box.
4. Click **OK** in the **Color** dialog box.
5. Click **OK** in the **Color Properties** dialog box.



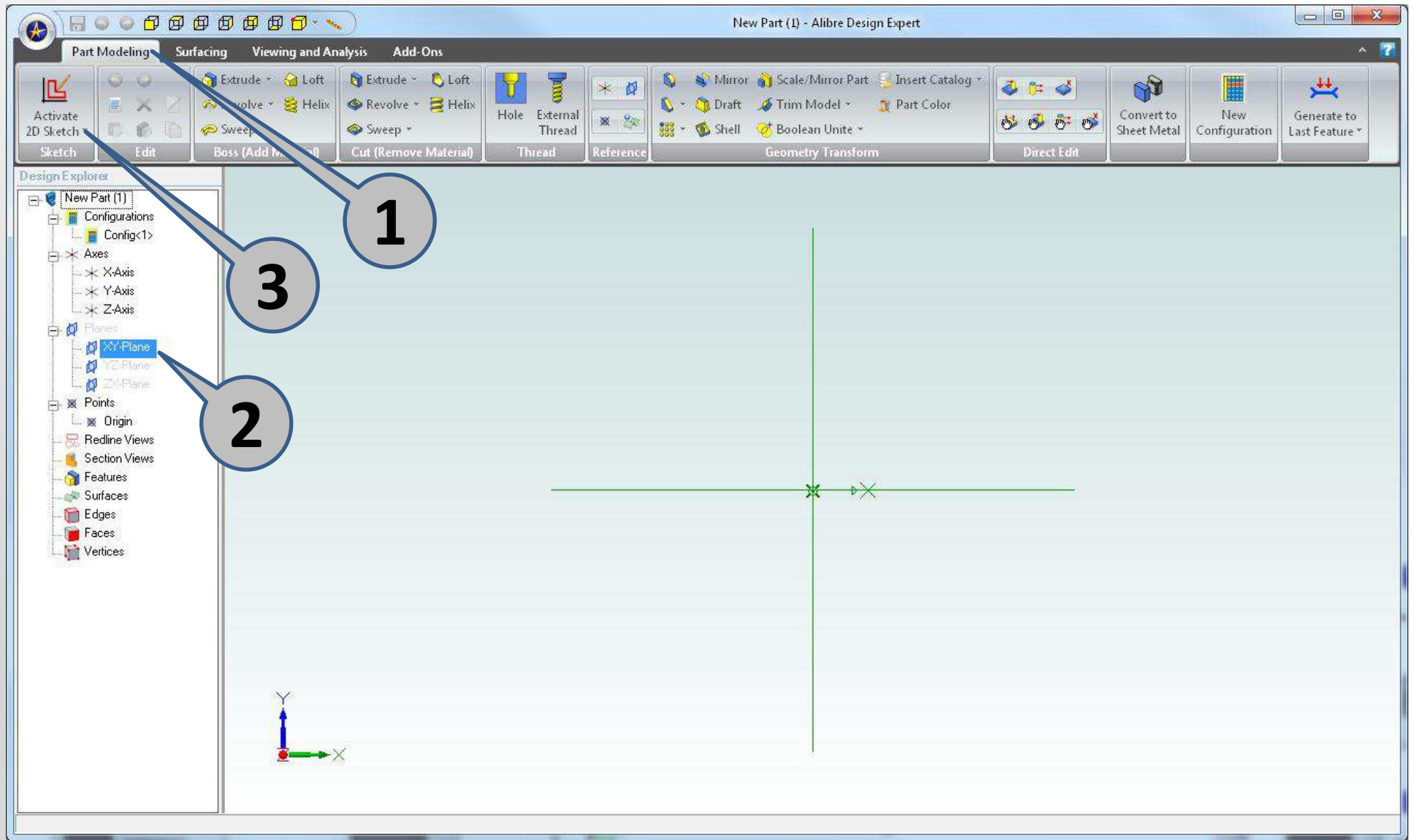
1. Save this **Part** to the same location as the **Journal** and **Crank** with the name: **Pulley** (not shown).
2. Close the **Part** workspace (not shown).



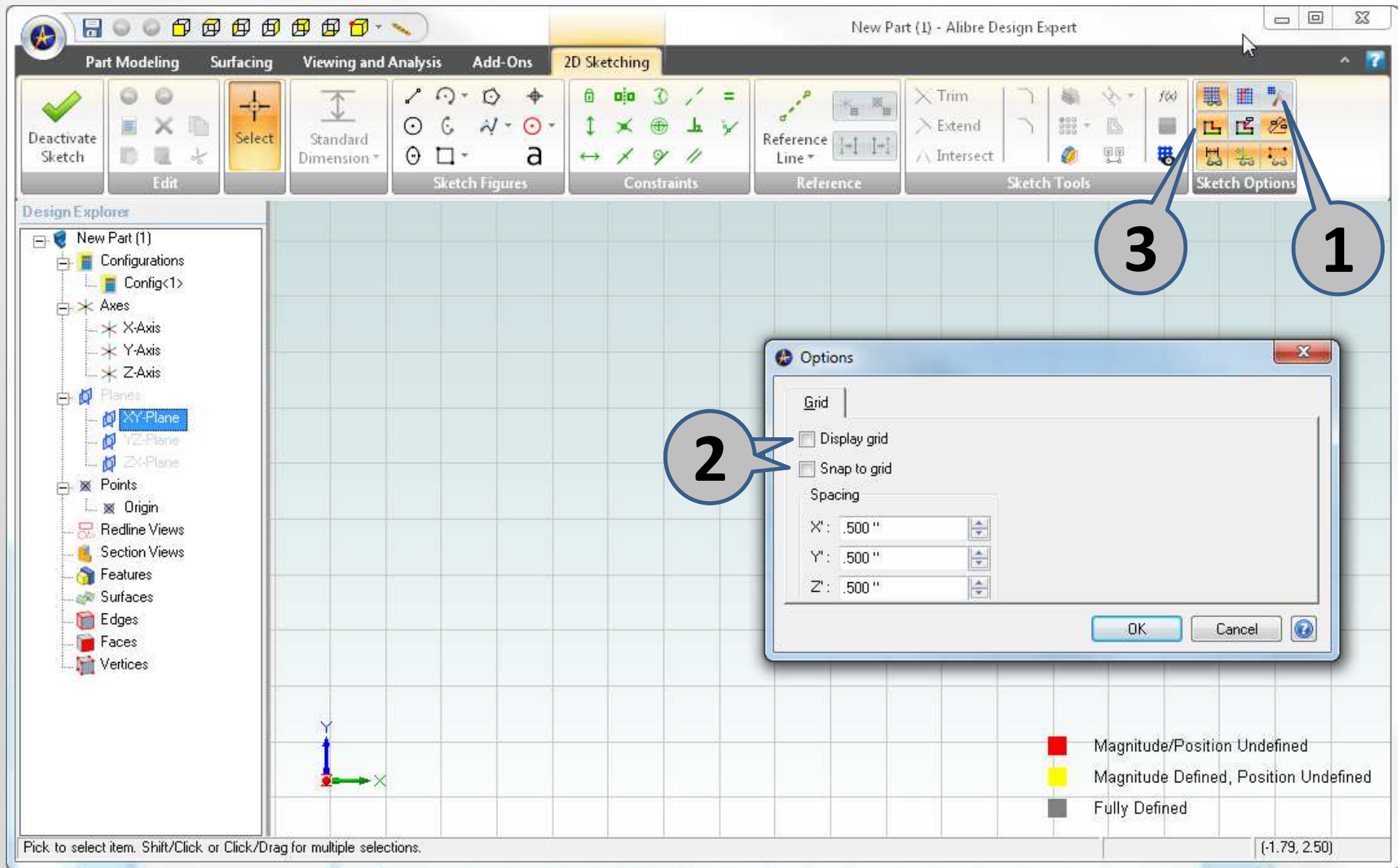
Creating the base



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

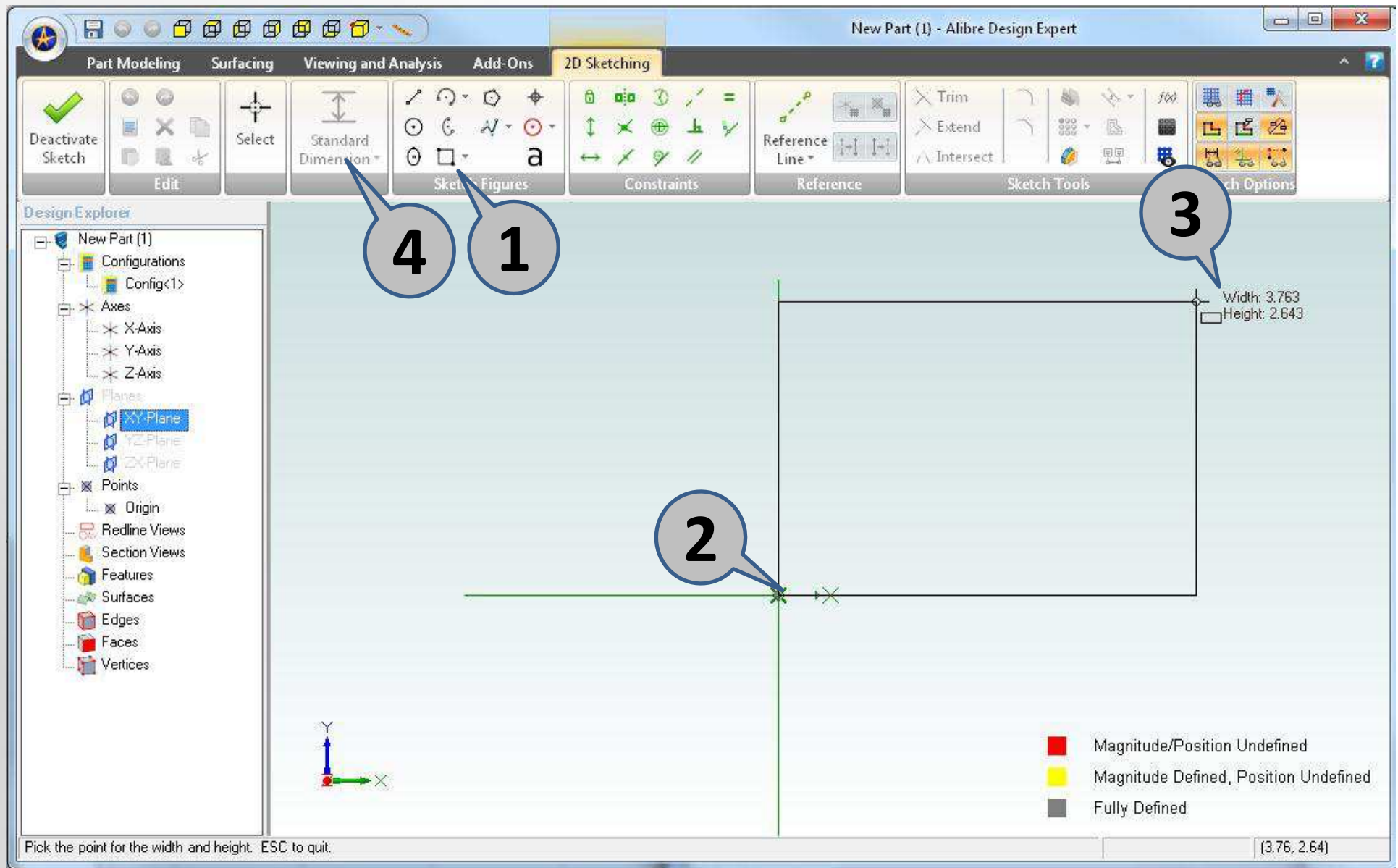


1. If necessary, click on the **Part Modeling** tab in the ribbon.
2. Click on the **XY Plane** in the **Design Explorer**.
3. Click on the **Activate 2D Sketch** icon.

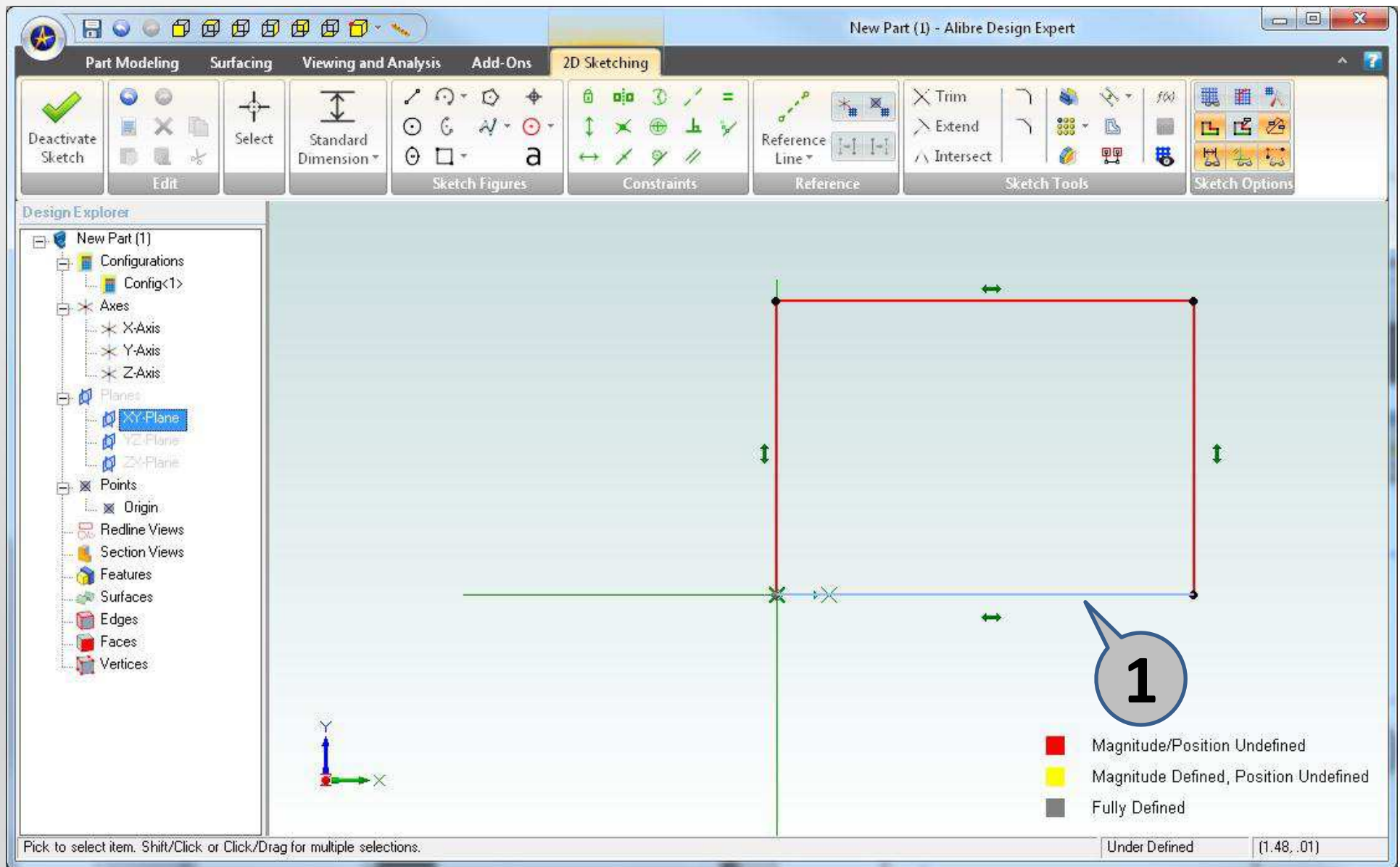


This lesson will utilize another common way to create geometry in Alibre Design (relative to the previous lessons in this tutorial). Extensive use of constraints, especially dimensions (also called parameters) characterize this method of creating and controlling geometry.

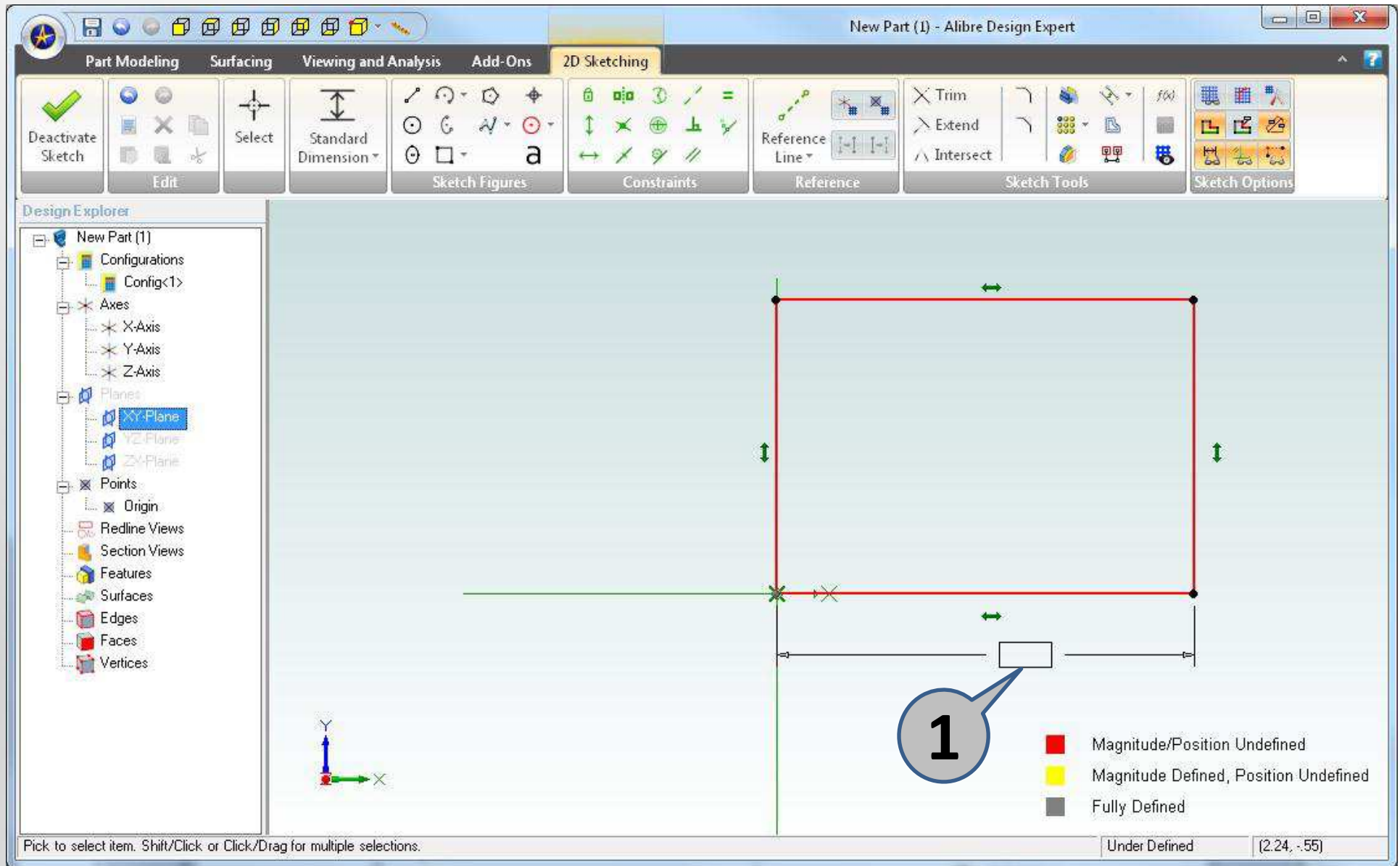
1. Click on the Grid options icon.
2. Turn off the **Grid** and **Snap** options and click the OK button.
3. Click on the DOF Colors icon.



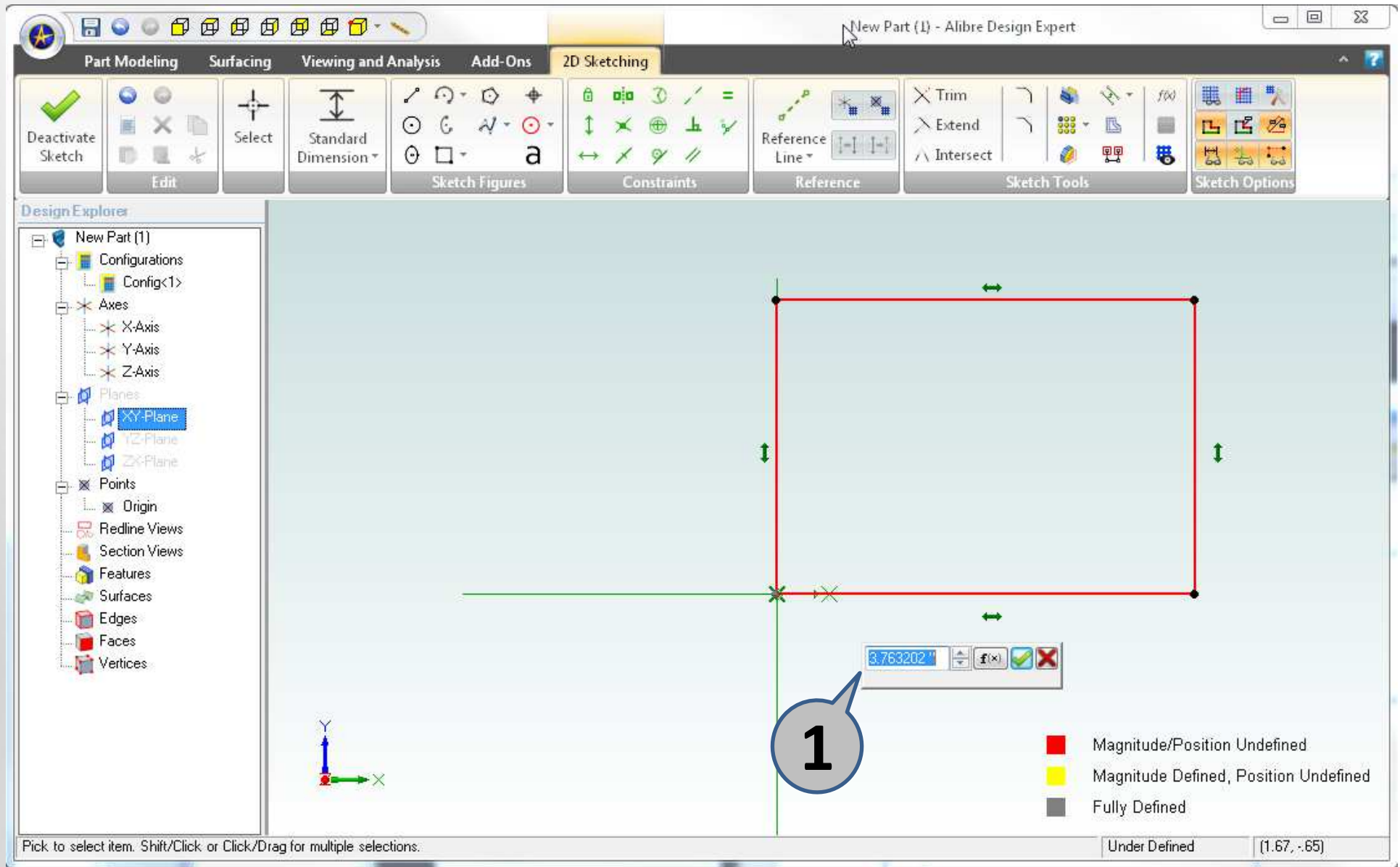
1. Click on the **Rectangle by Two Corners** tool.
2. Position the mouse cursor over the **Origin** point. Click and release to create the first corner.
3. Move the cursor upwards and to the right an arbitrary distance (approximately as shown). Click and release to complete the rectangle.
4. Click on the **Dimension** tool.



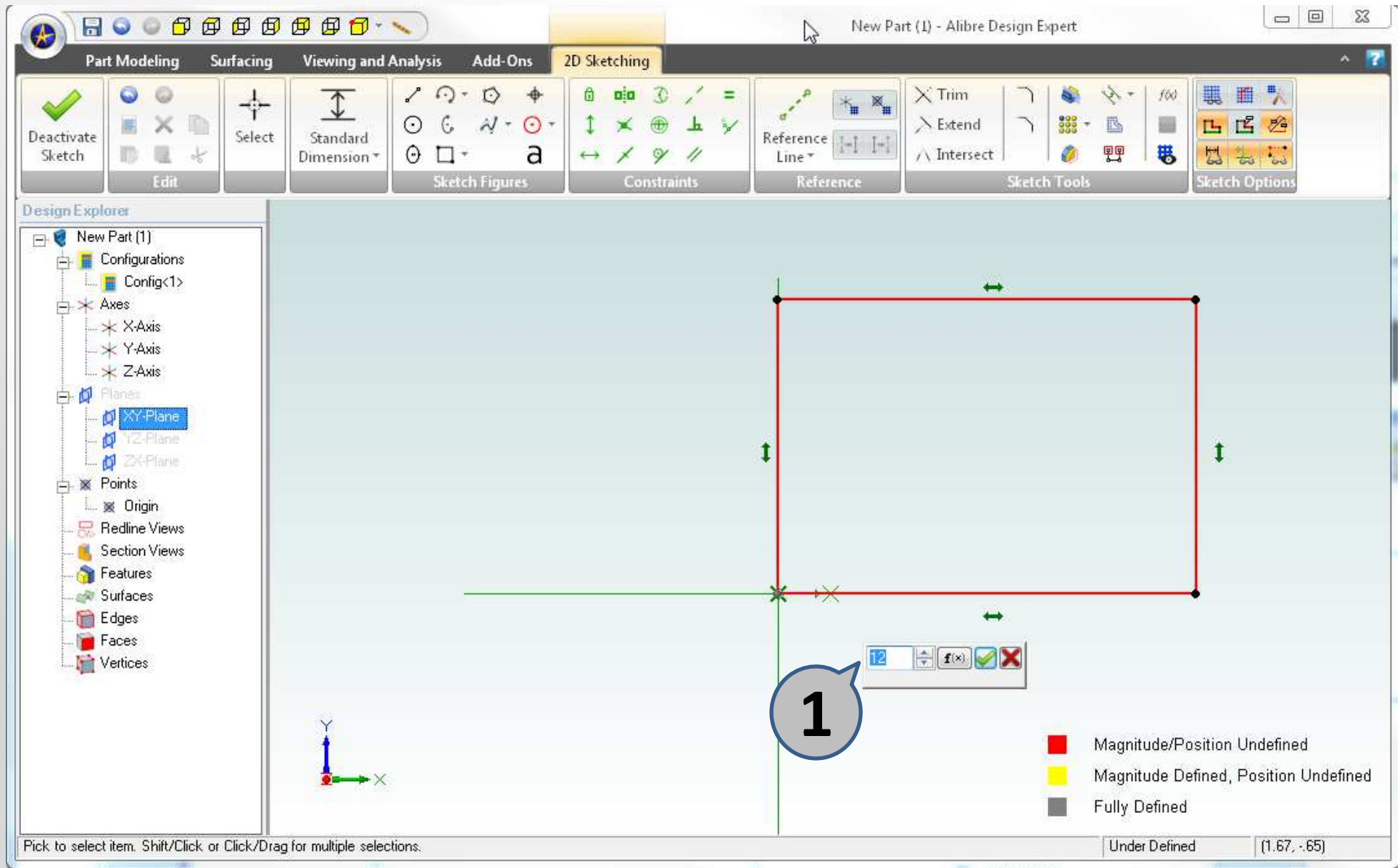
1. Click on the bottom edge of the rectangle



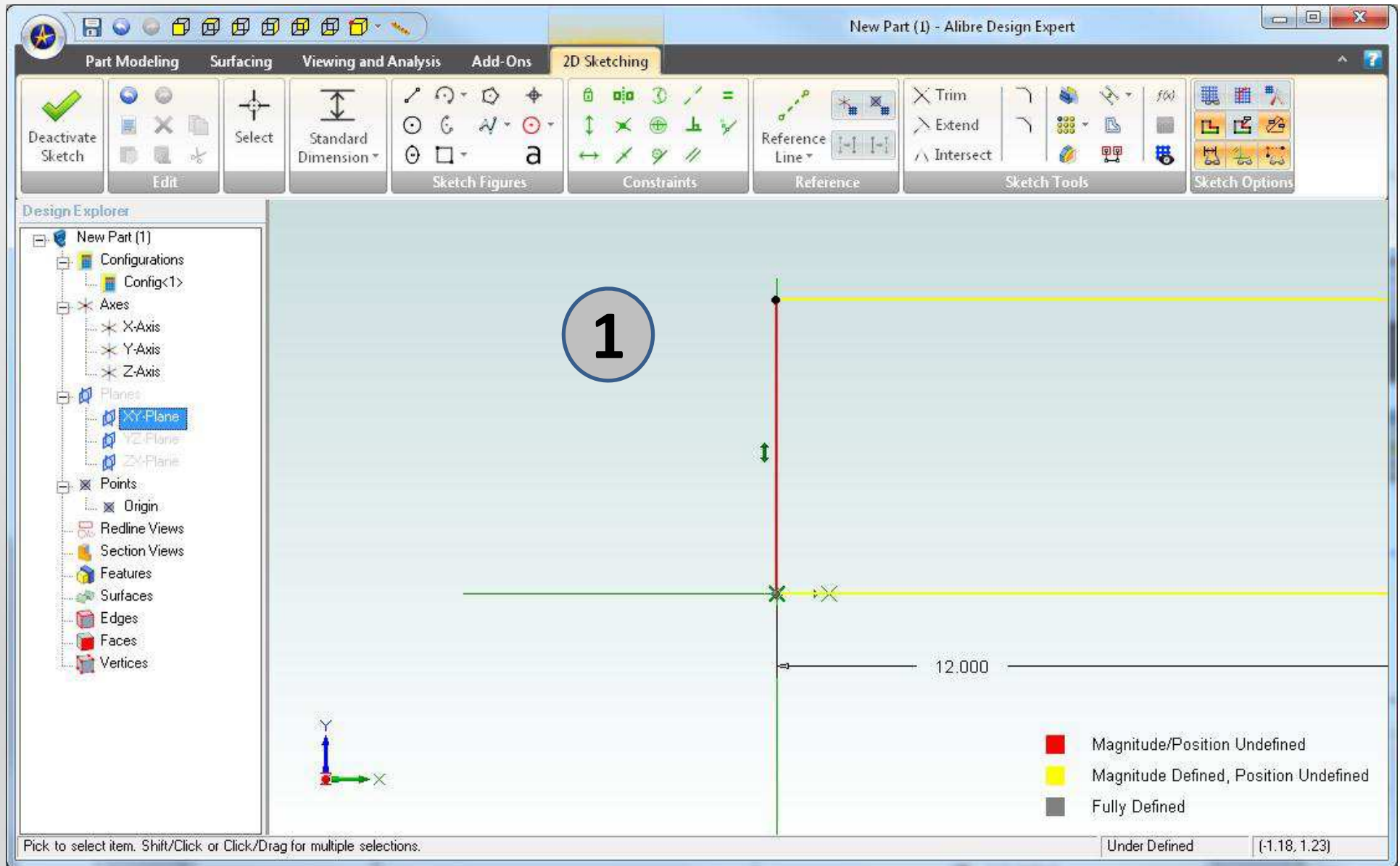
1. Establish the position of the dimension by moving the cursor downward, then click and release.



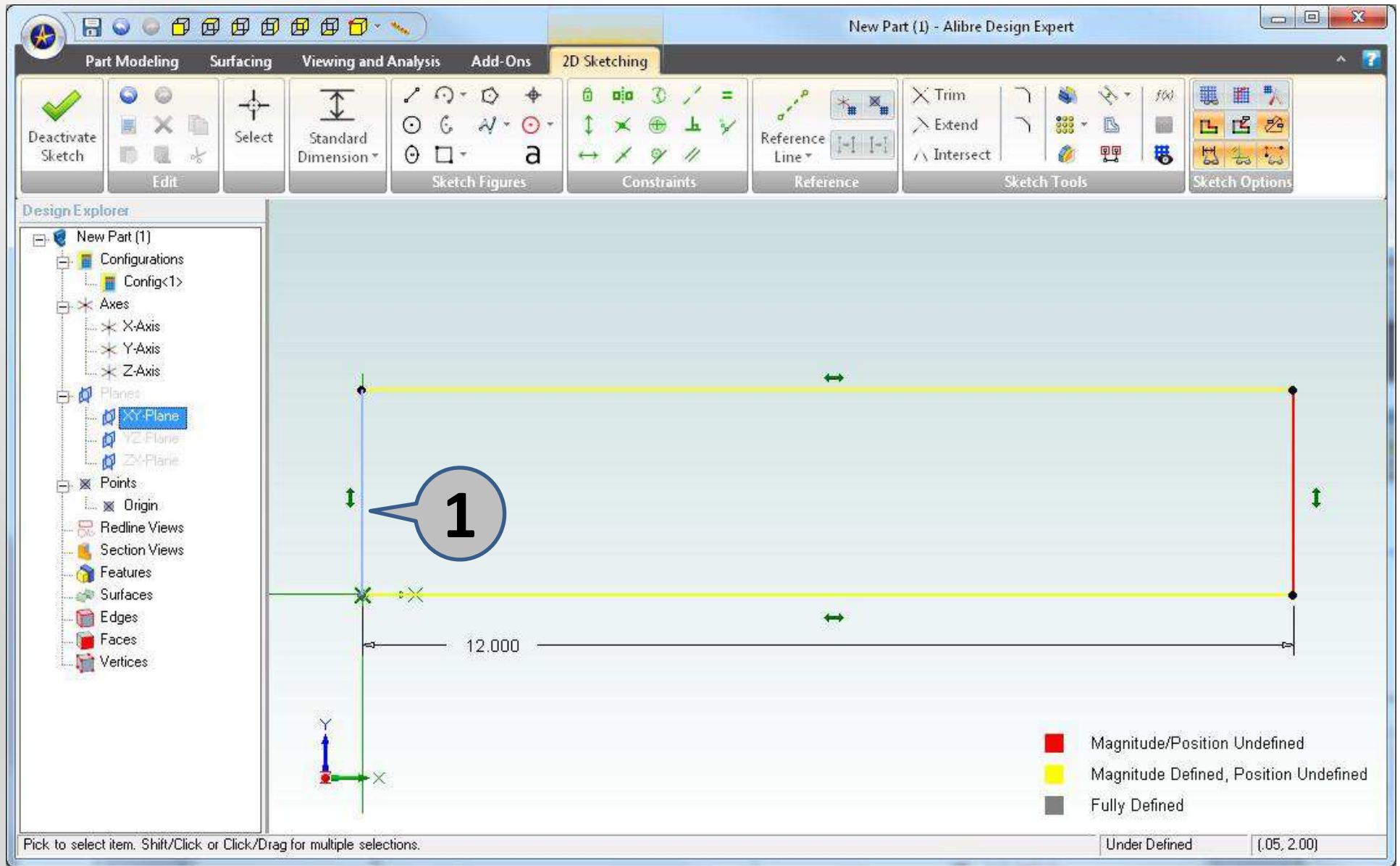
1. The **Dimension** tool displays the precise length of the line segment as it currently exists.



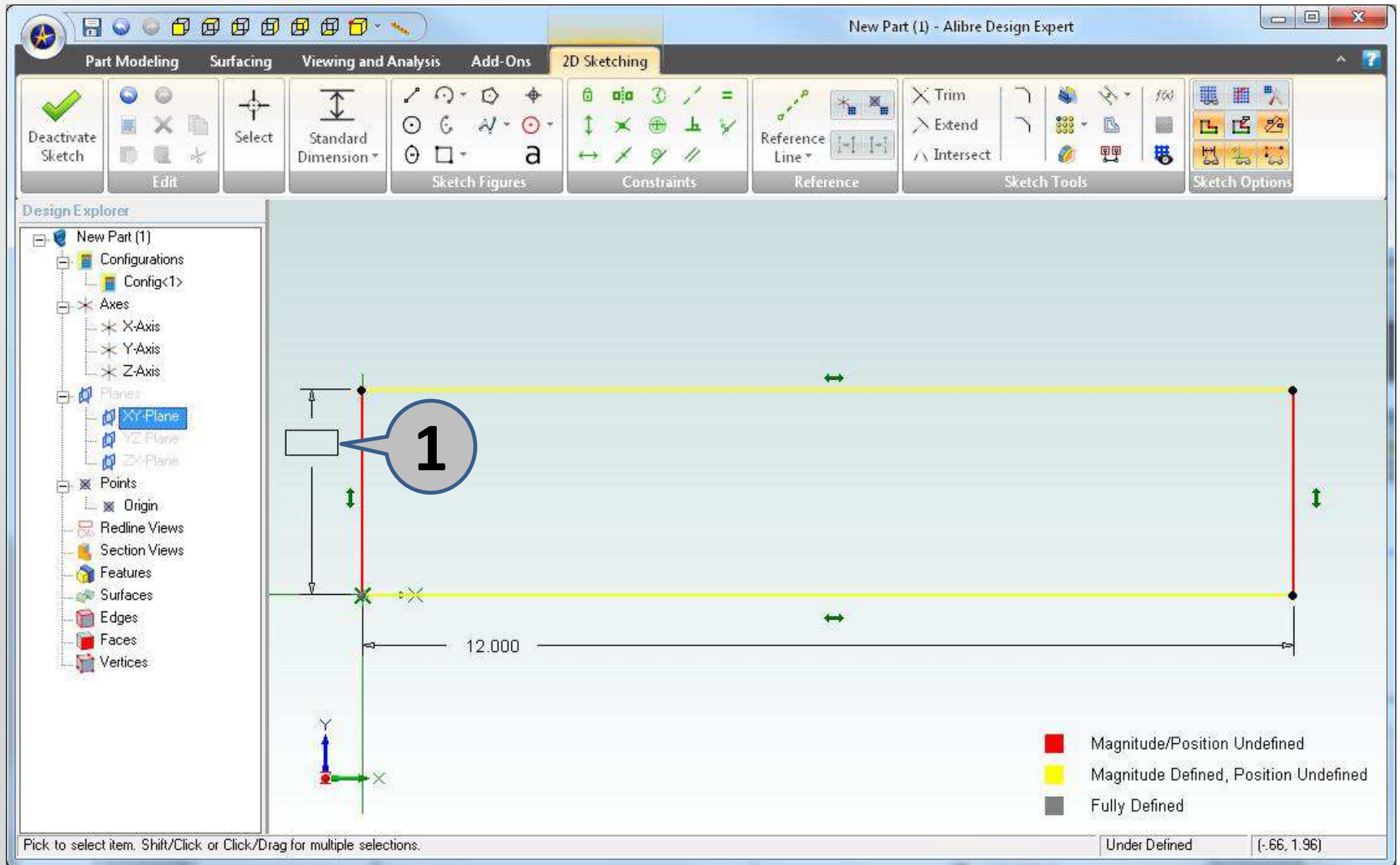
1. Enter **12** in the entry box.
2. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



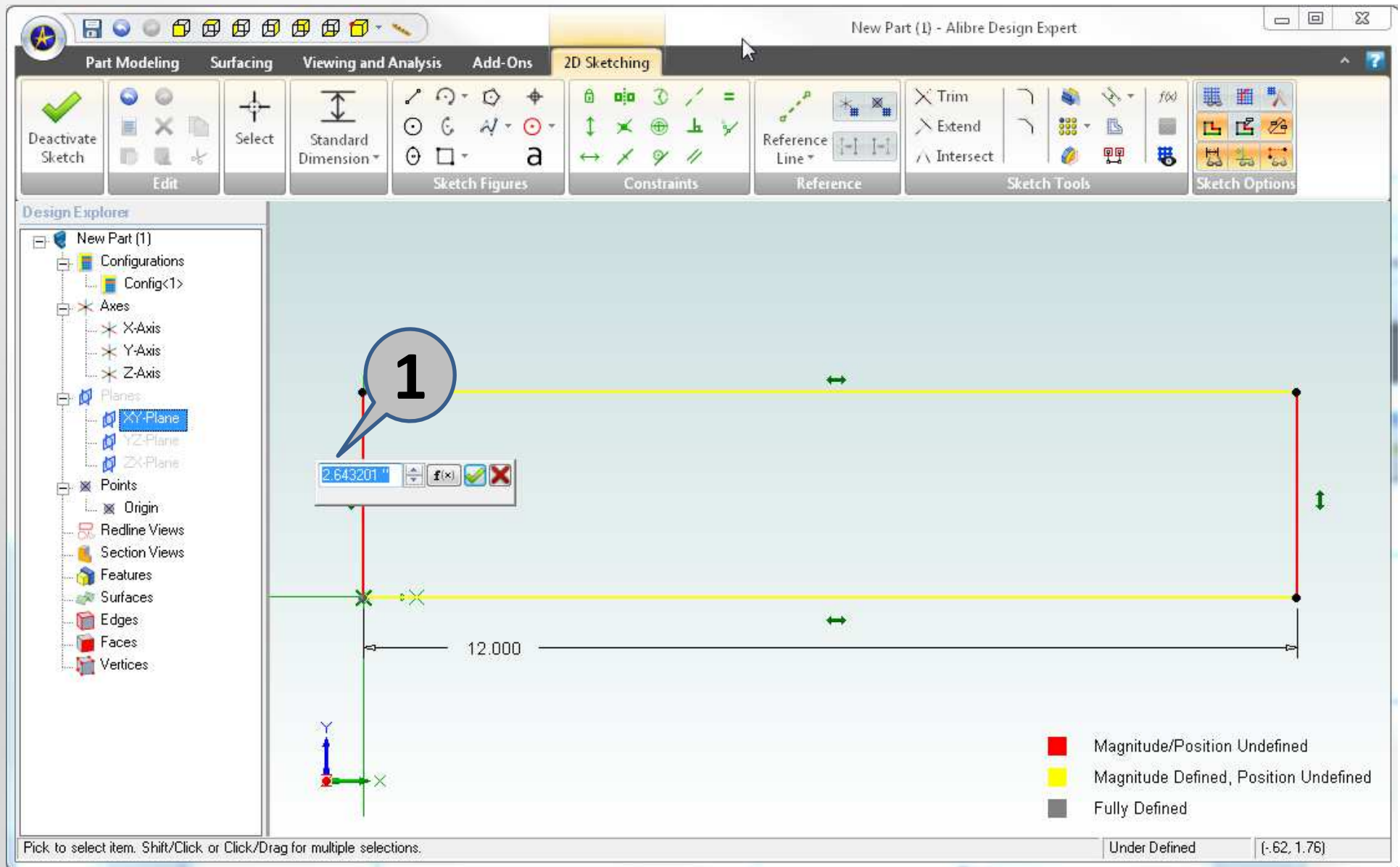
1. Confirm the result of the **Dimension**. Take note that dimensions in Alibre Design control the elements to which they're applied, rather than merely documenting them.
2. Click the **Home** key on the keyboard (not shown).



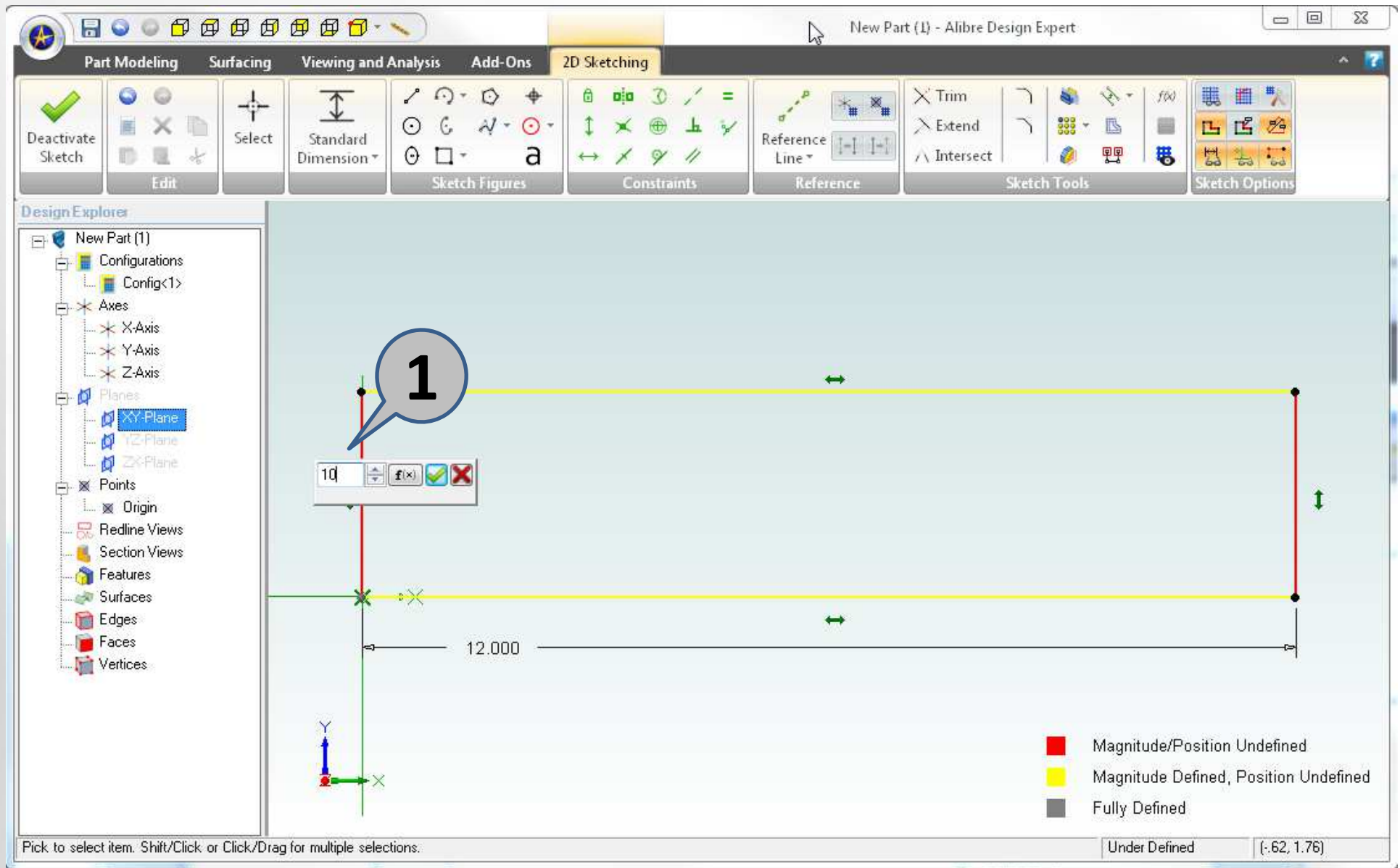
1. Click on the left segment of the rectangle.



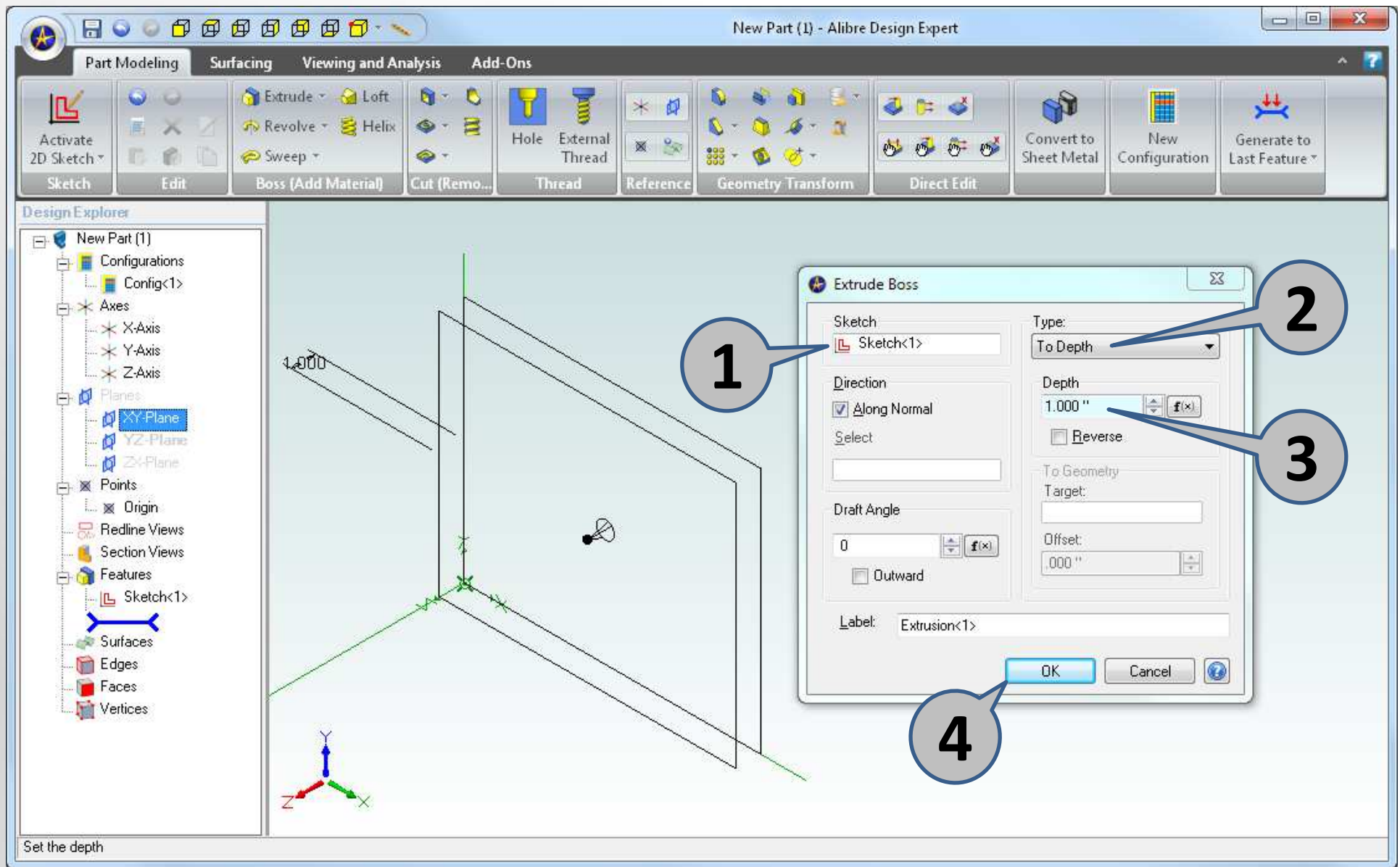
1. Establish the position of the dimension by moving the cursor to the left, then click and release.



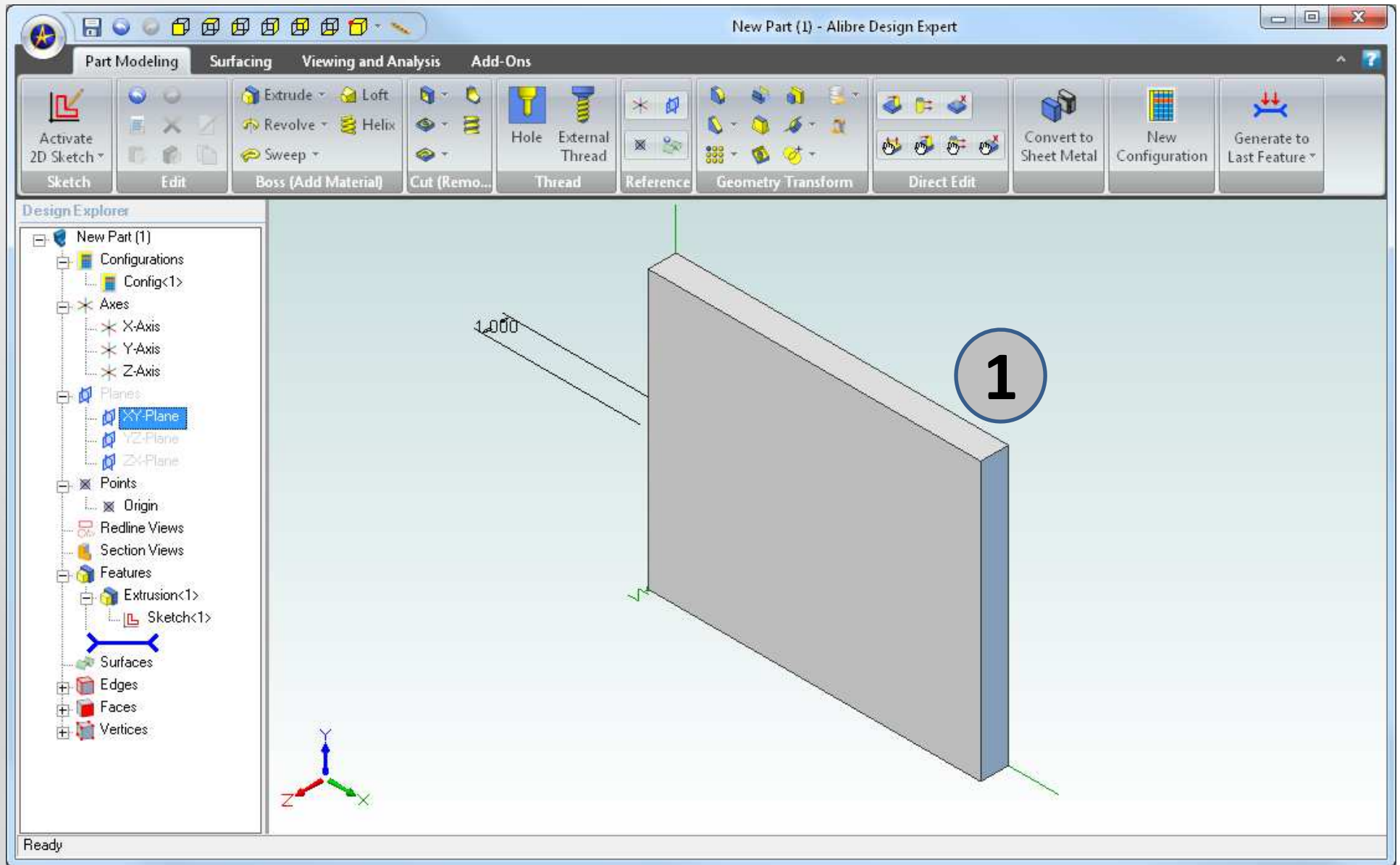
1. The **Dimension** tool displays the precise length of the line segment as it currently exists.



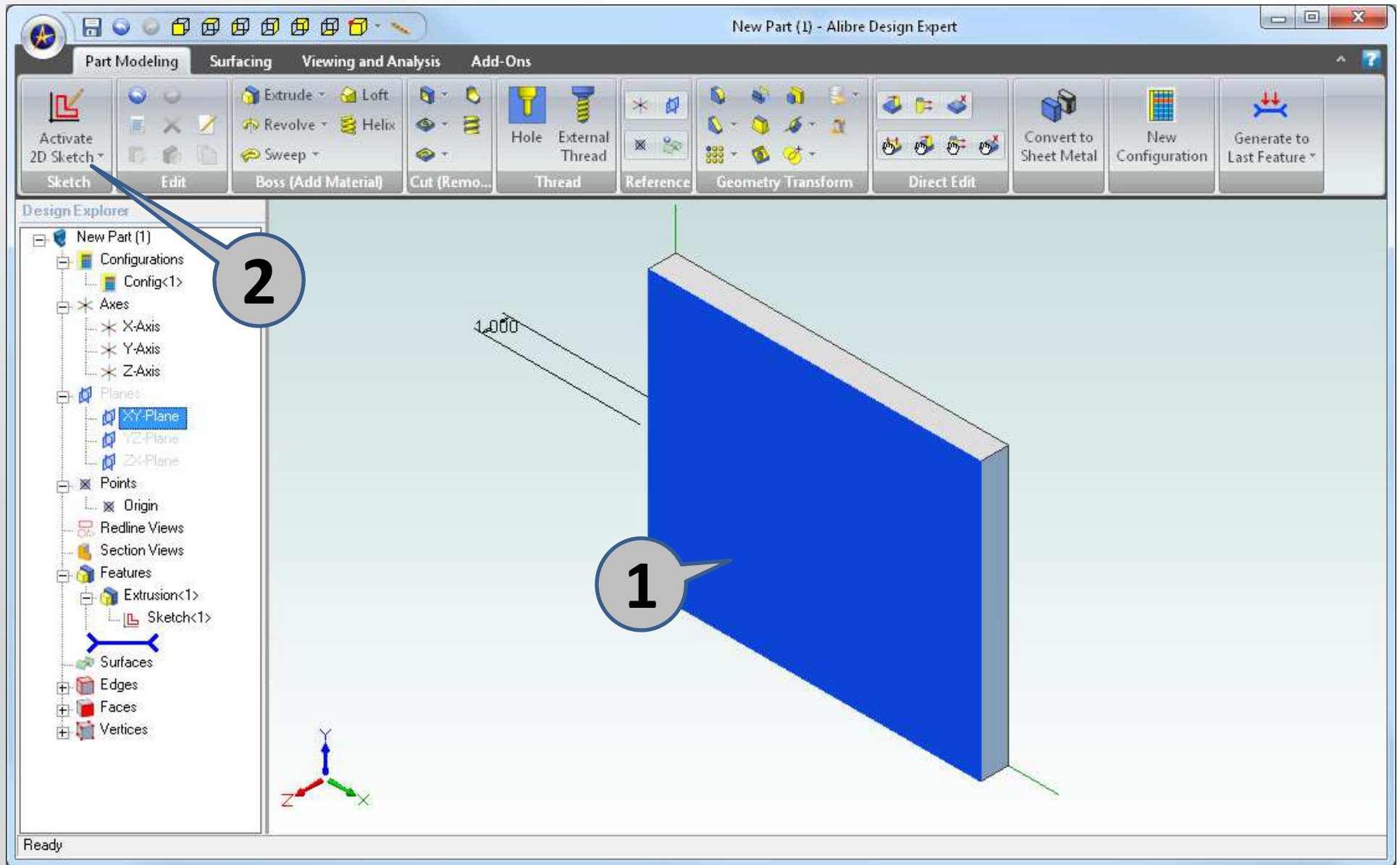
1. Enter **10** in the entry box.
2. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).
3. Click the **Home** key on the keyboard (not shown).



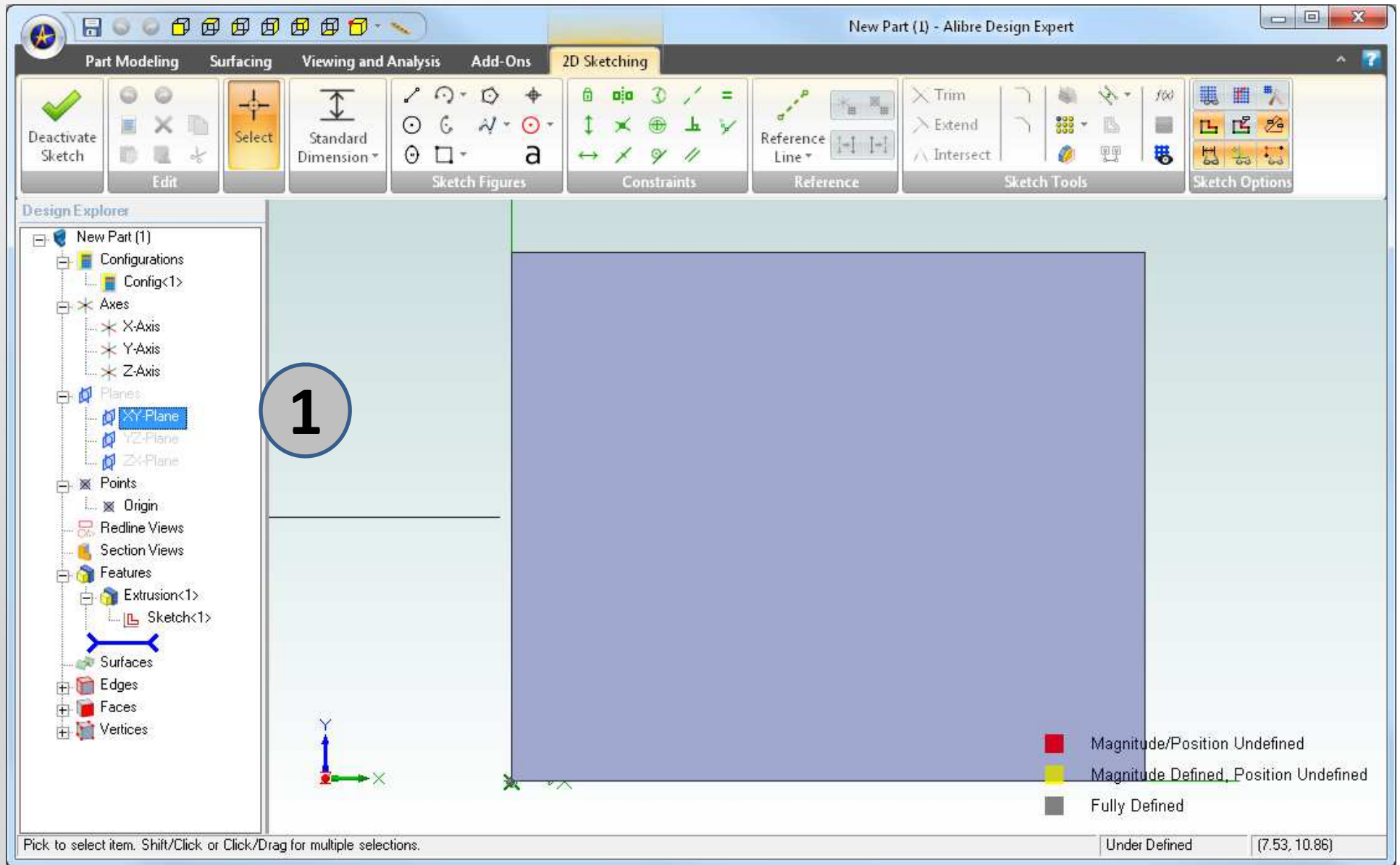
1. Confirm that **Sketch<1>** is the **Sketch** to be extruded.
2. Confirm **To Depth** is the **Type** of extrusion.
3. Enter a value of **1 inch** for the **Depth**.
4. Click **OK**.



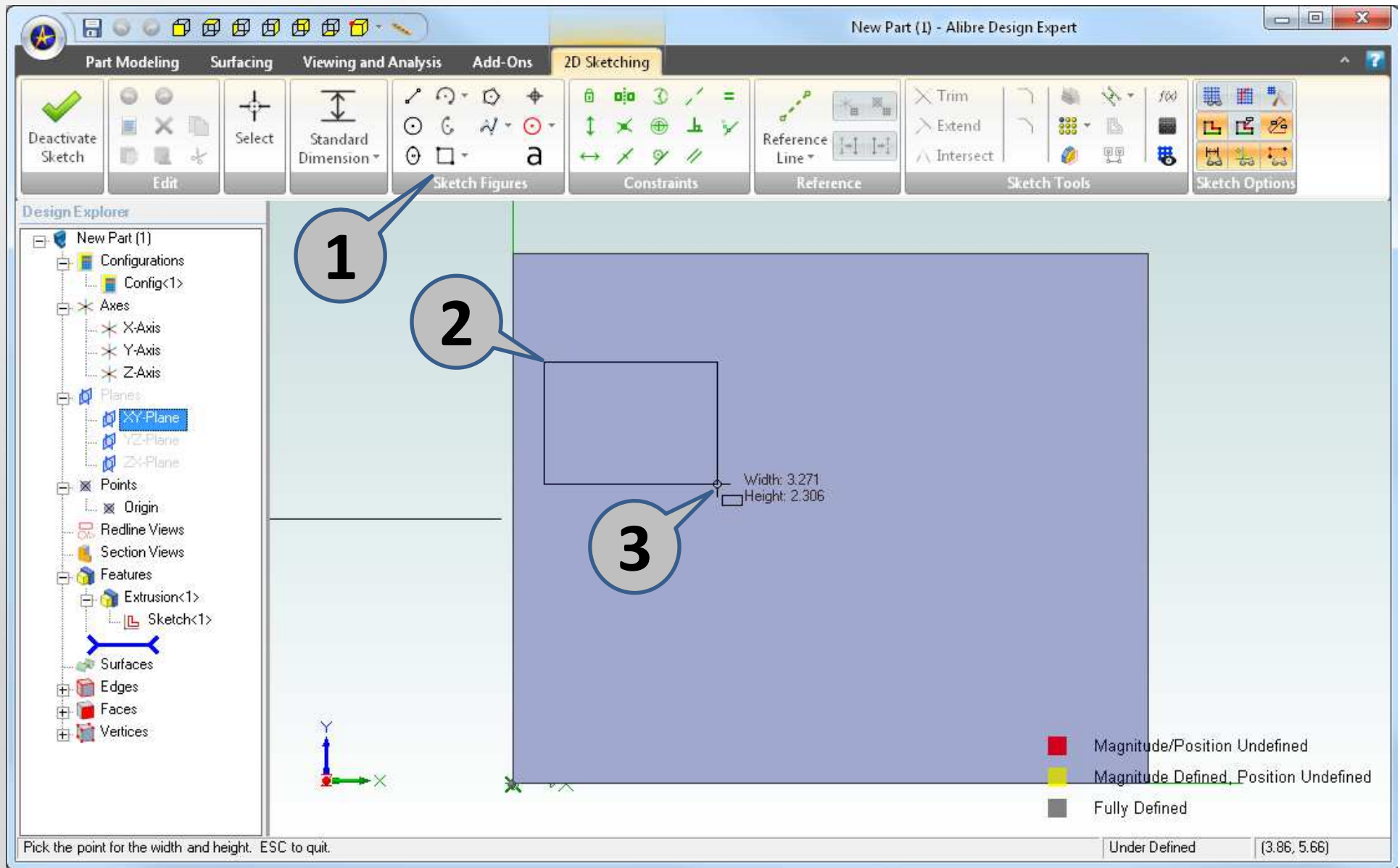
1. Confirm similar results for the **Extrude Boss** operation.
2. Press the **Home** key on the keyboard (not shown).



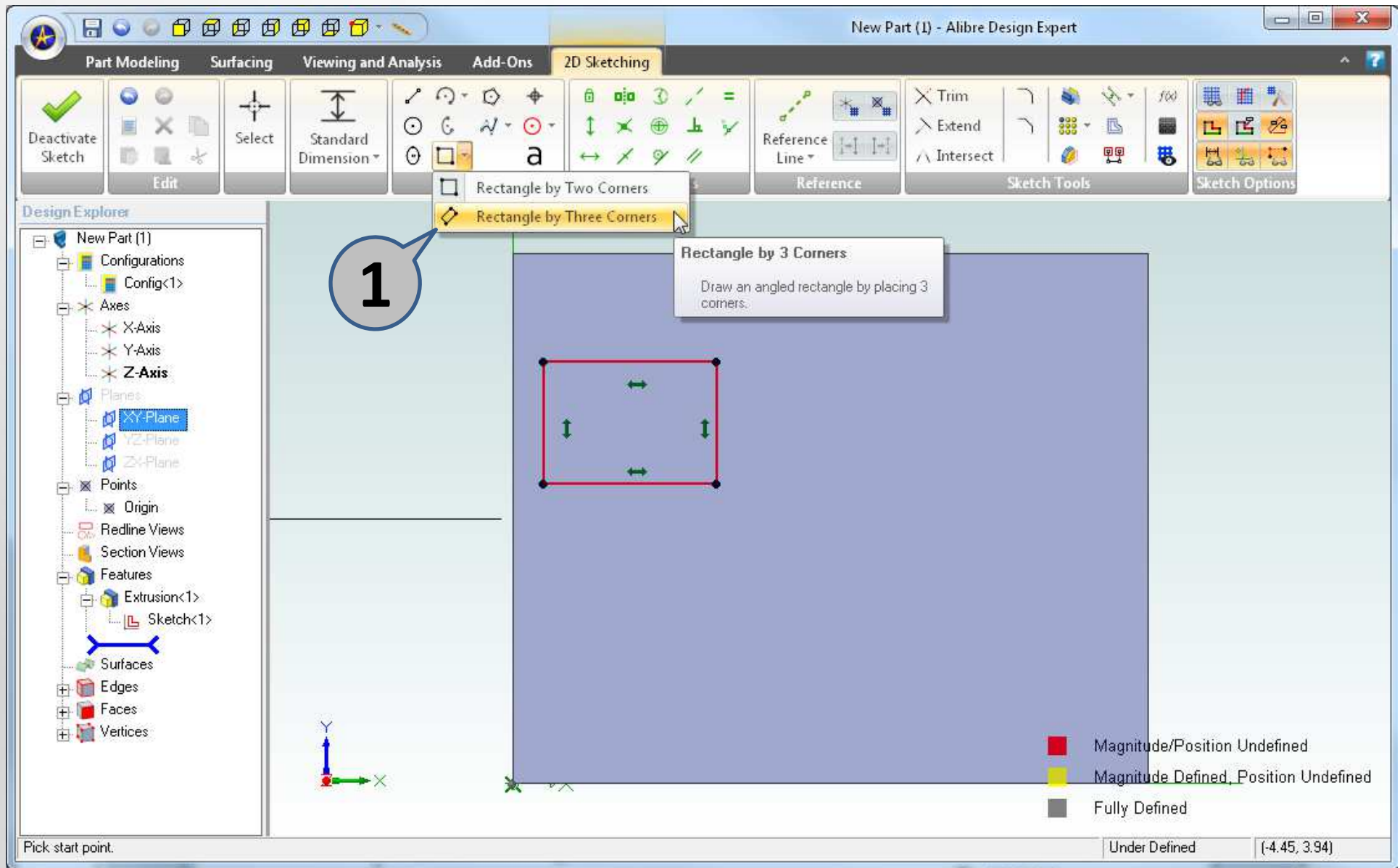
1. Click on the face indicated.
2. Click on **Activate 2D Sketch**.



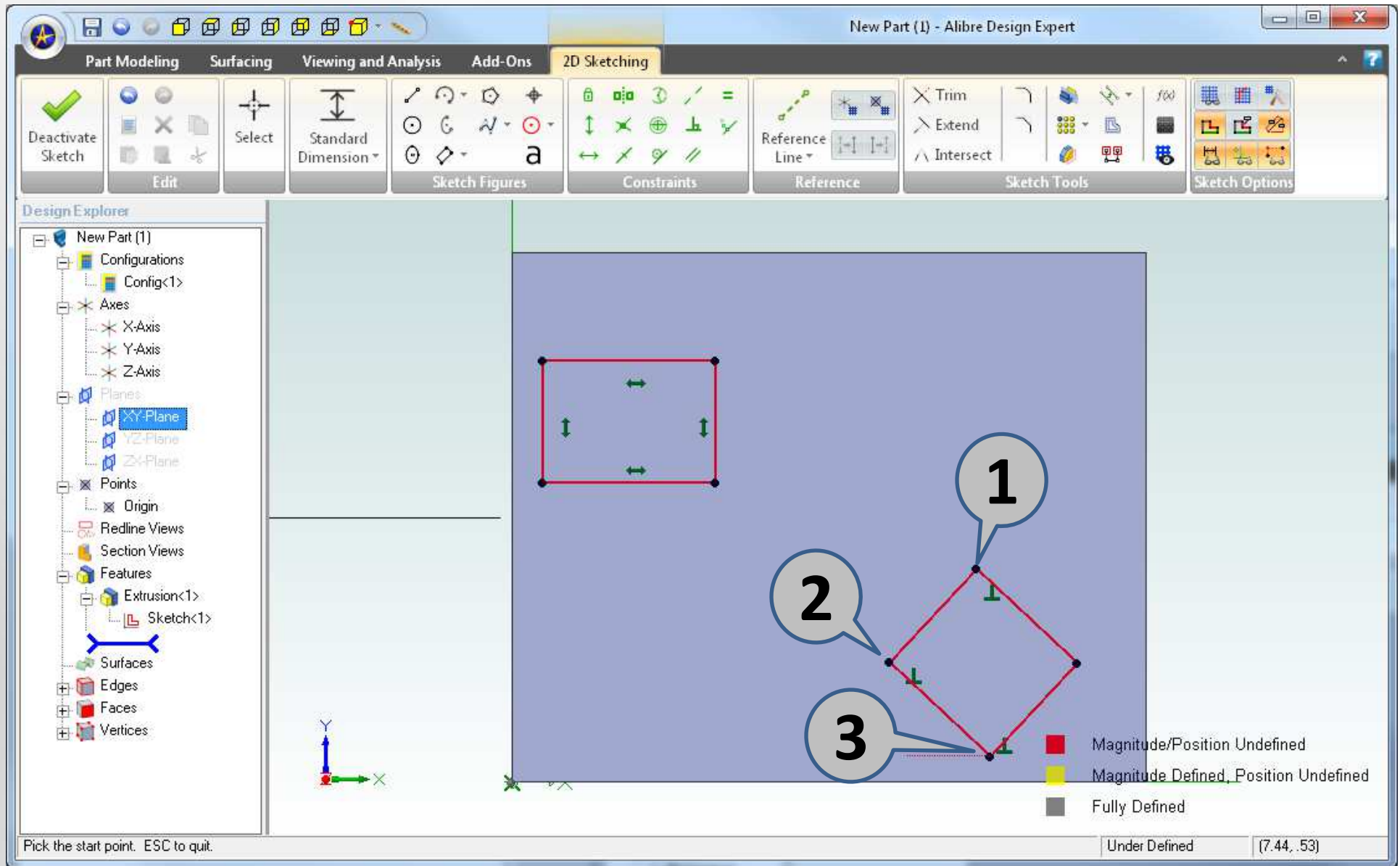
1. The view of the work area should be similar to the screen capture.



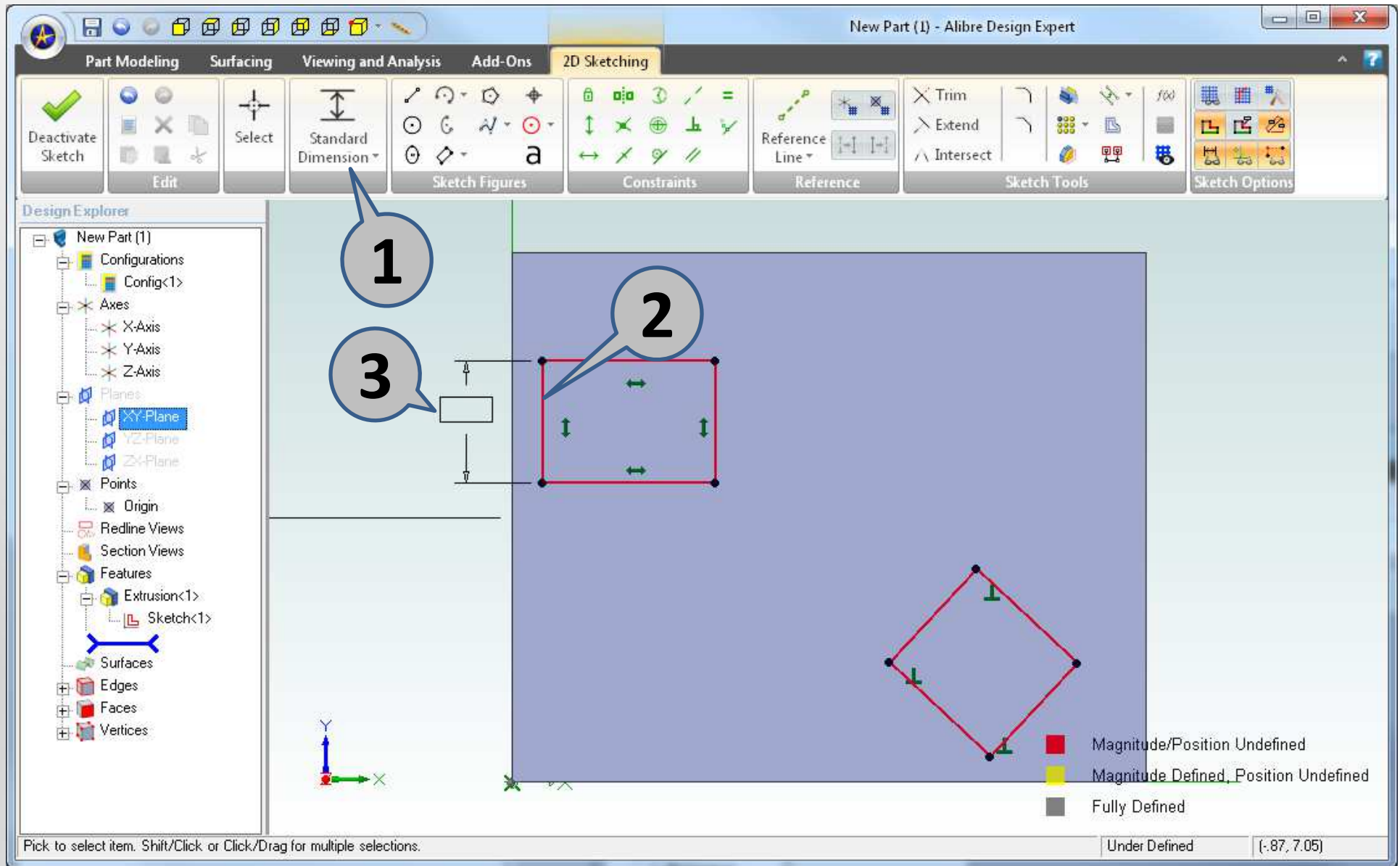
1. Click on the **Rectangle by Two Corners** tool.
2. Position the mouse cursor as shown (approximately). Click and release to create the first corner.
3. Move the mouse cursor to the position indicated (approximately). Click and release to complete the rectangle.



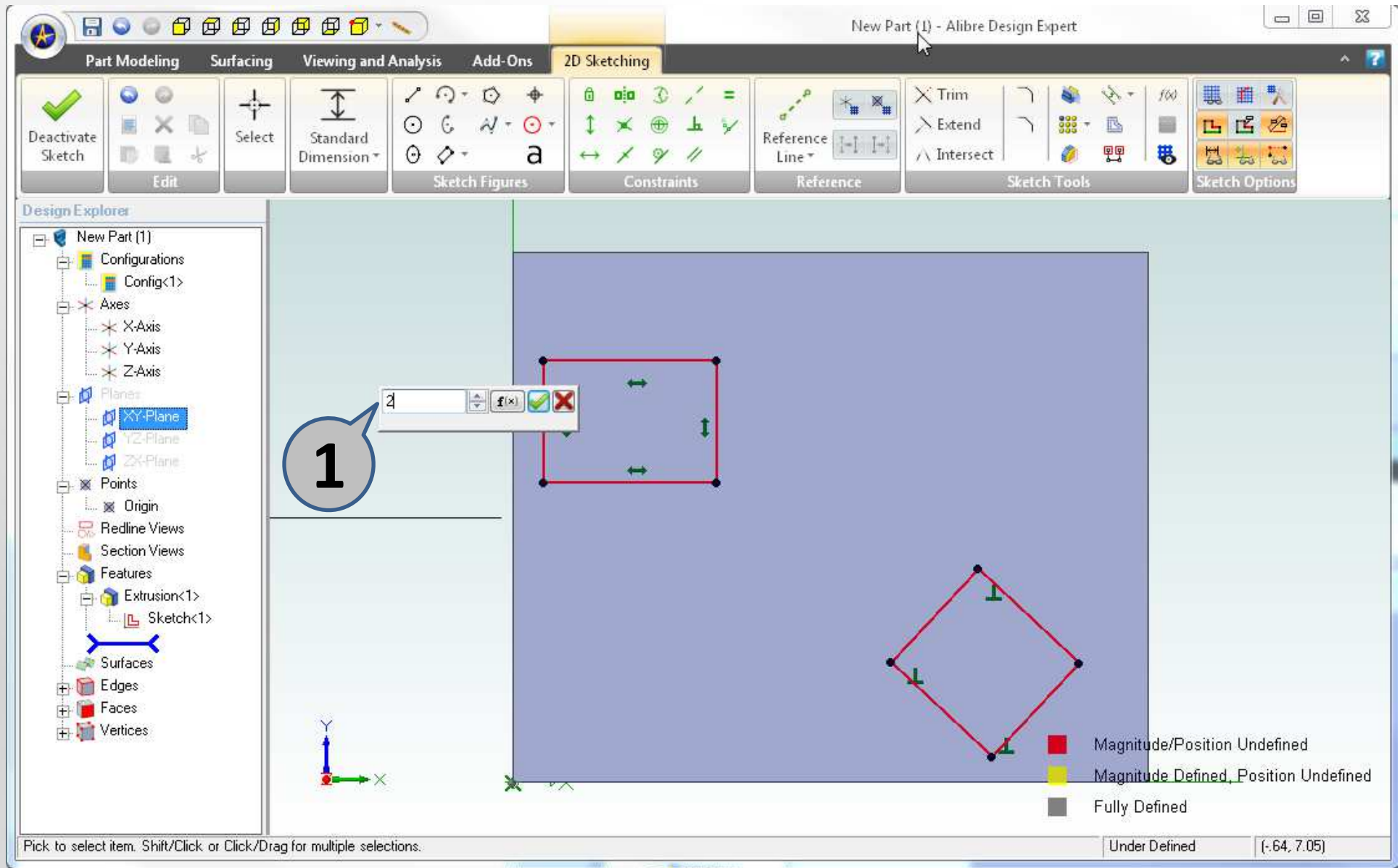
1. In the **Sketch Figures** group, click on the flyout in the **Rectangle** icon and select the **Rectangle by Three Corners** option.



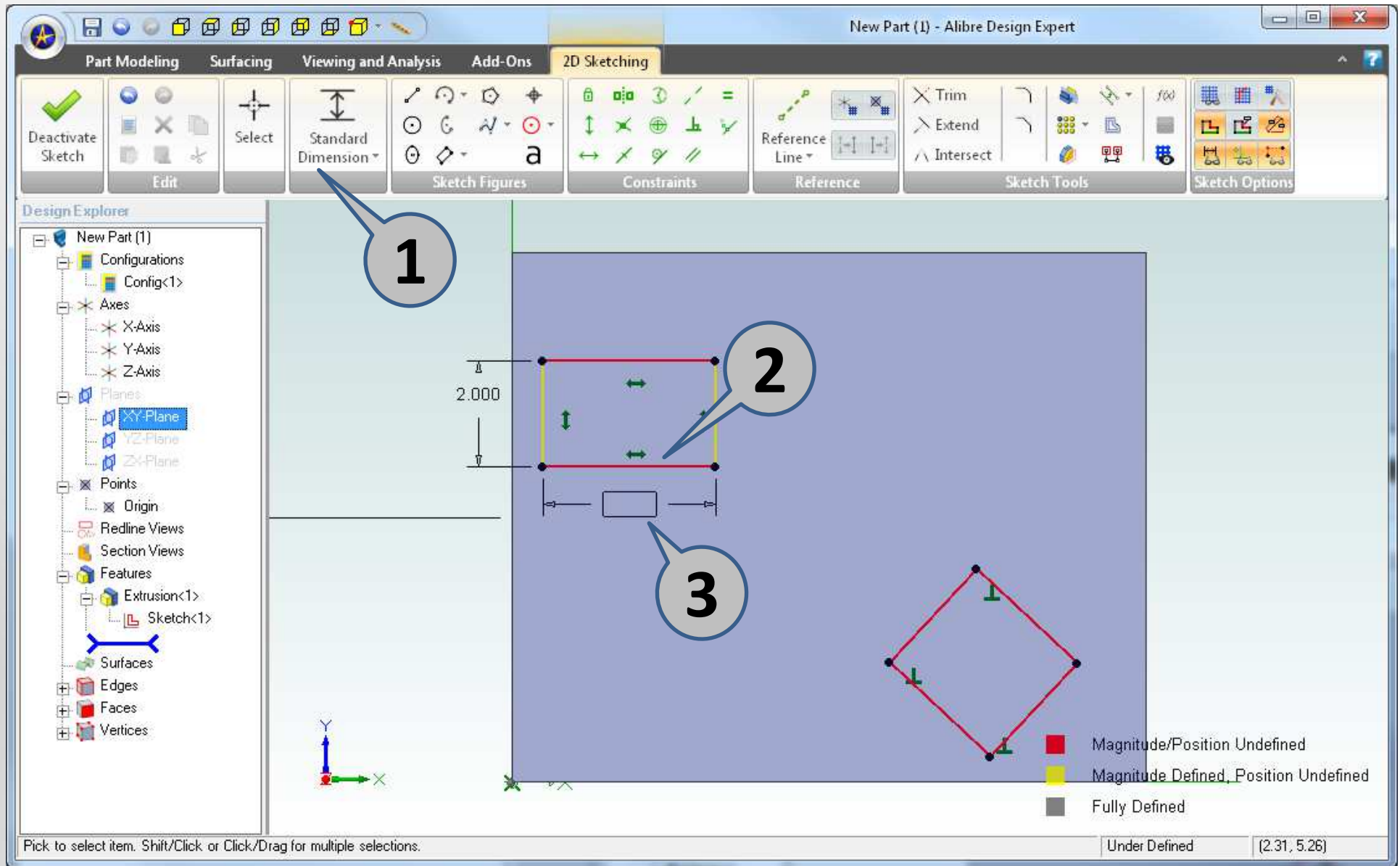
1. Position the mouse cursor as indicated (approximately). Click and release (first corner).
2. Move the mouse cursor to the position indicated (approximately). Click and release (second corner).
3. Move the mouse cursor to the position indicated (approximately). Click and release to complete the rectangle.



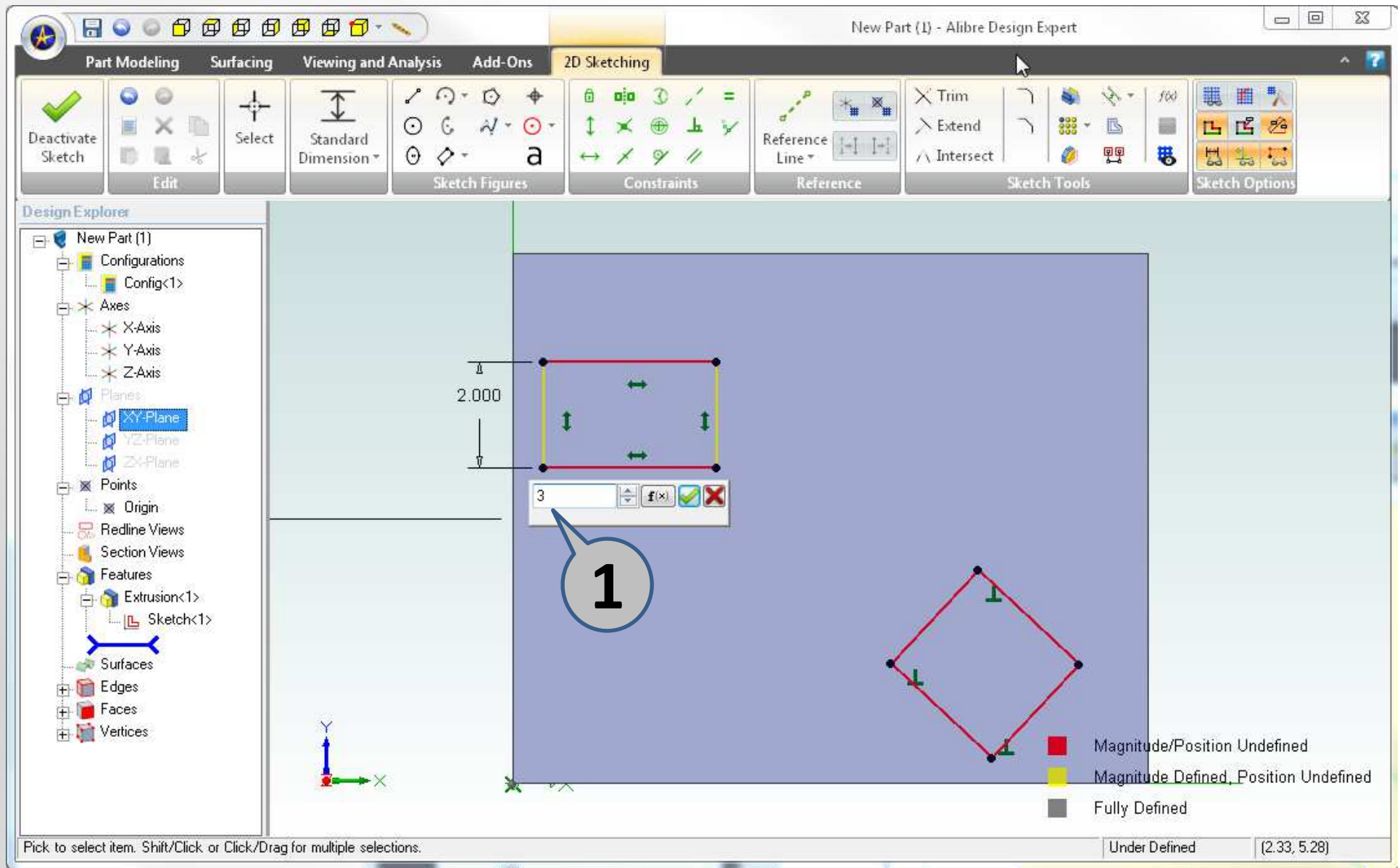
1. Click on the **Dimension** tool.
2. Click on the indicated segment to dimension.
3. Position the dimension as shown (approximately). Click and release to place the dimension.



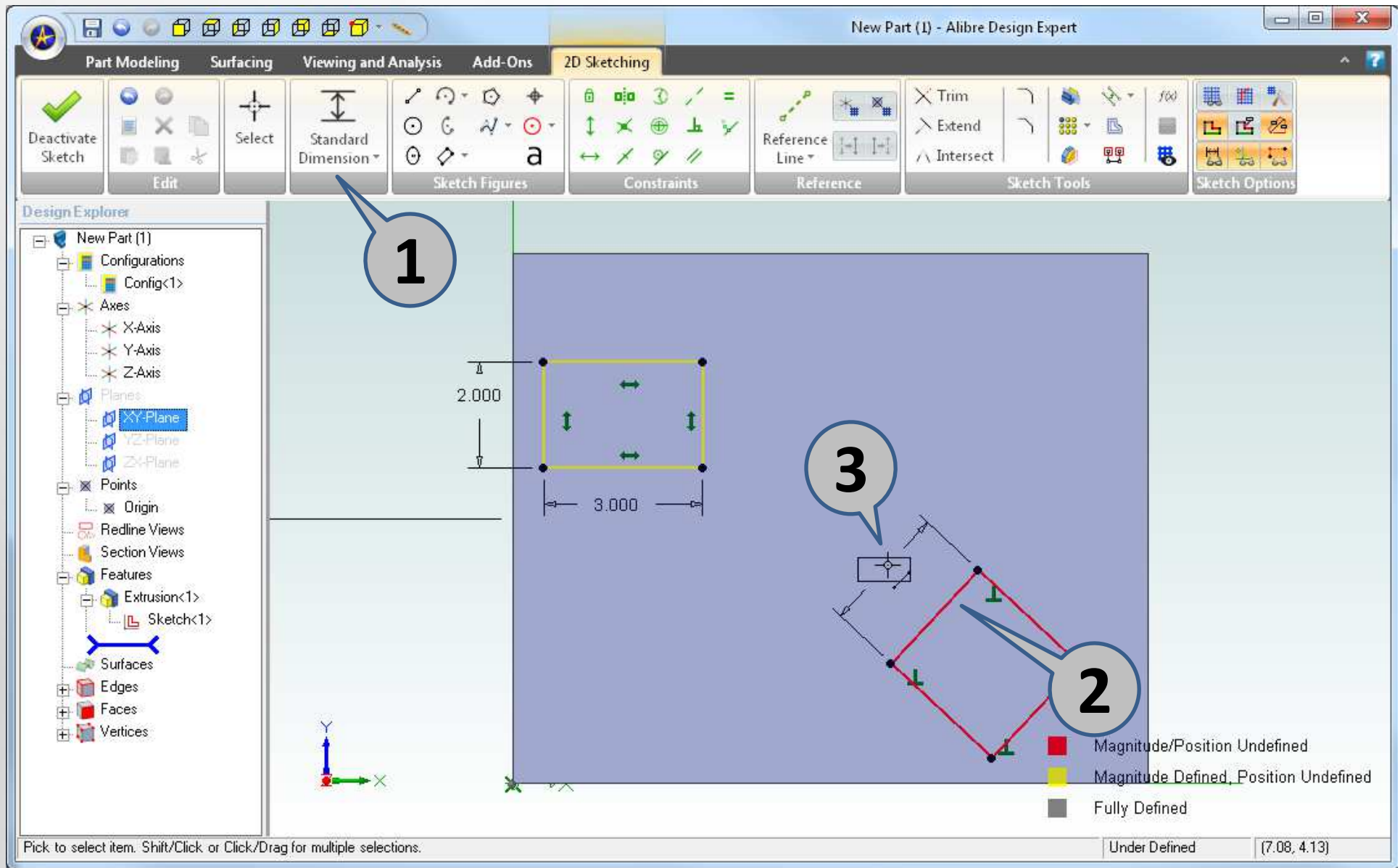
1. Enter **2** for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



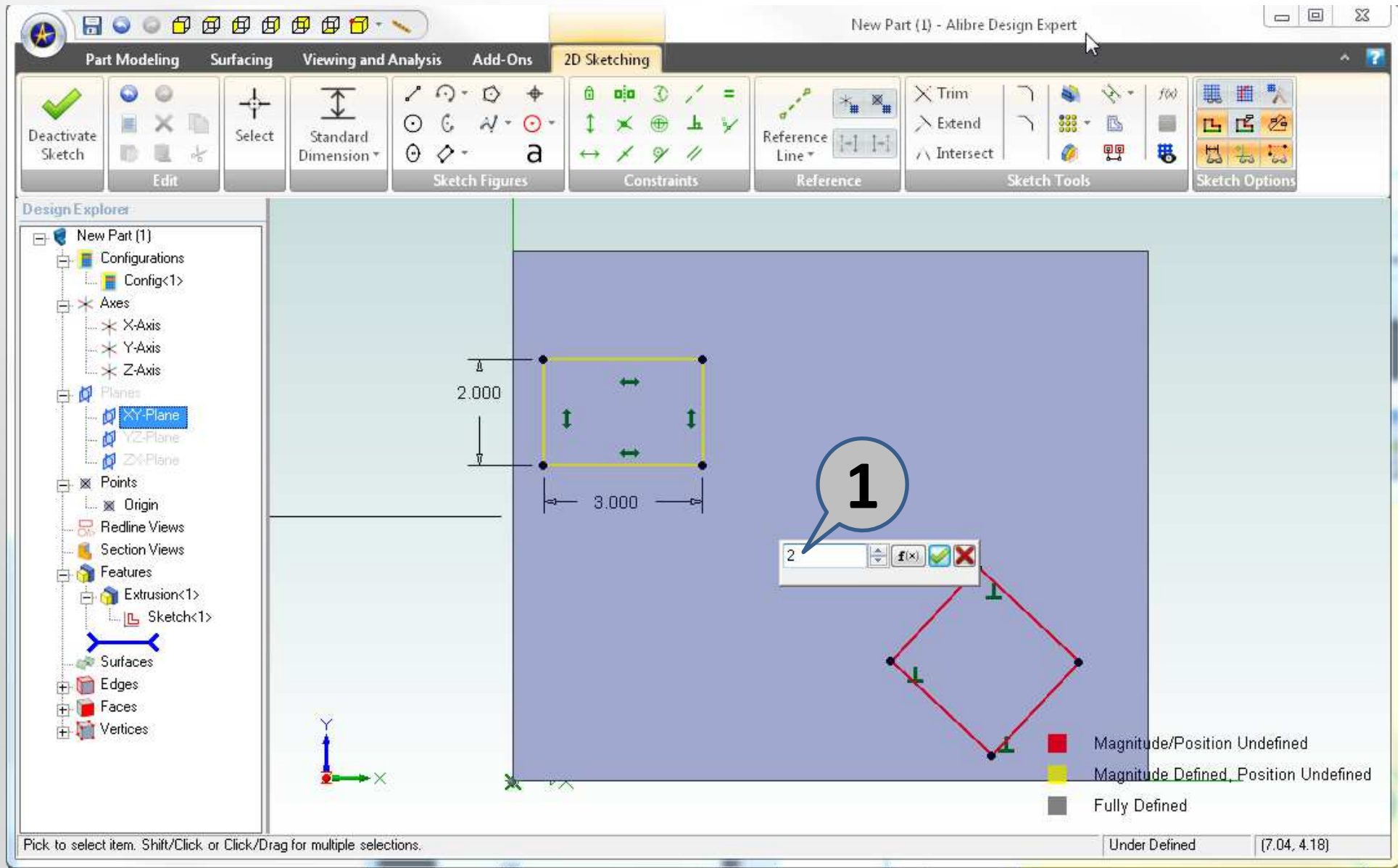
1. The **Dimension** tool should still be enabled from the previous step. If it is not still enabled then click on the **Dimension** tool.
2. Click on the indicated segment to dimension.
3. Position the dimension as shown (approximately). Click and release to place the dimension.



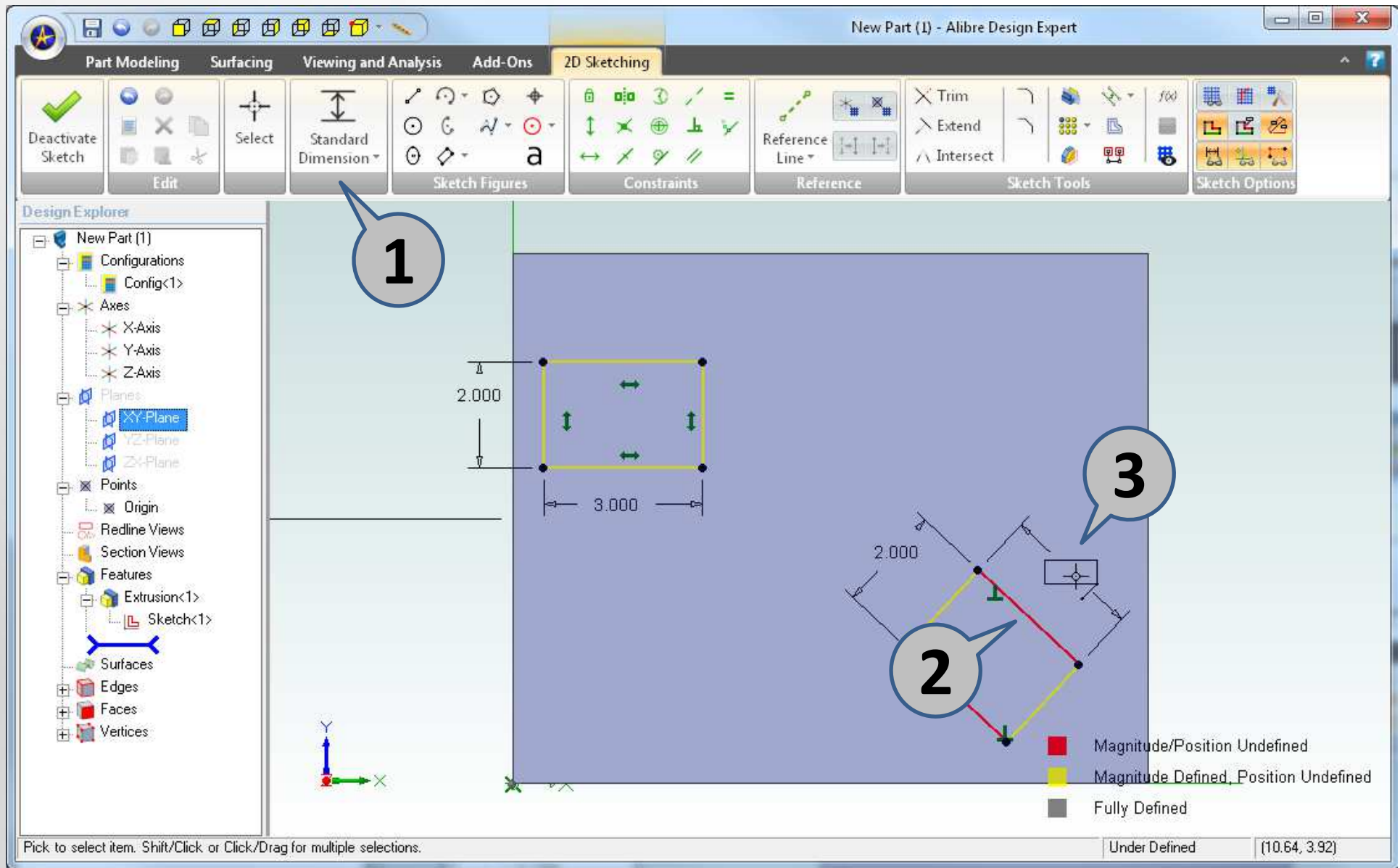
1. Enter **3** for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



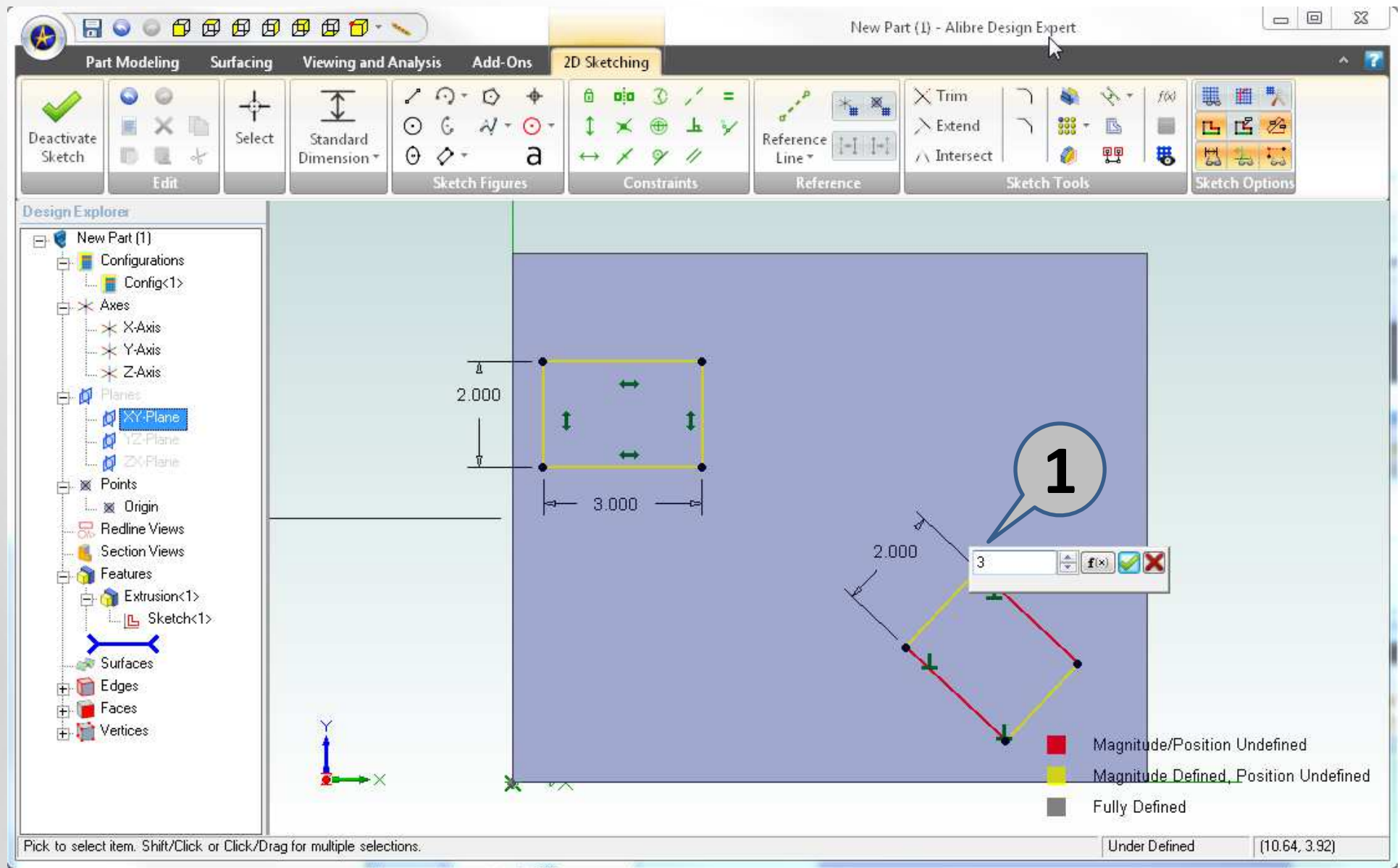
1. The **Dimension** tool should still be enabled from the previous steps. If it is not still enabled then click on the **Dimension** tool.
2. Click on the indicated segment to dimension.
3. Position the dimension as shown (approximately). Click and release to place the dimension.



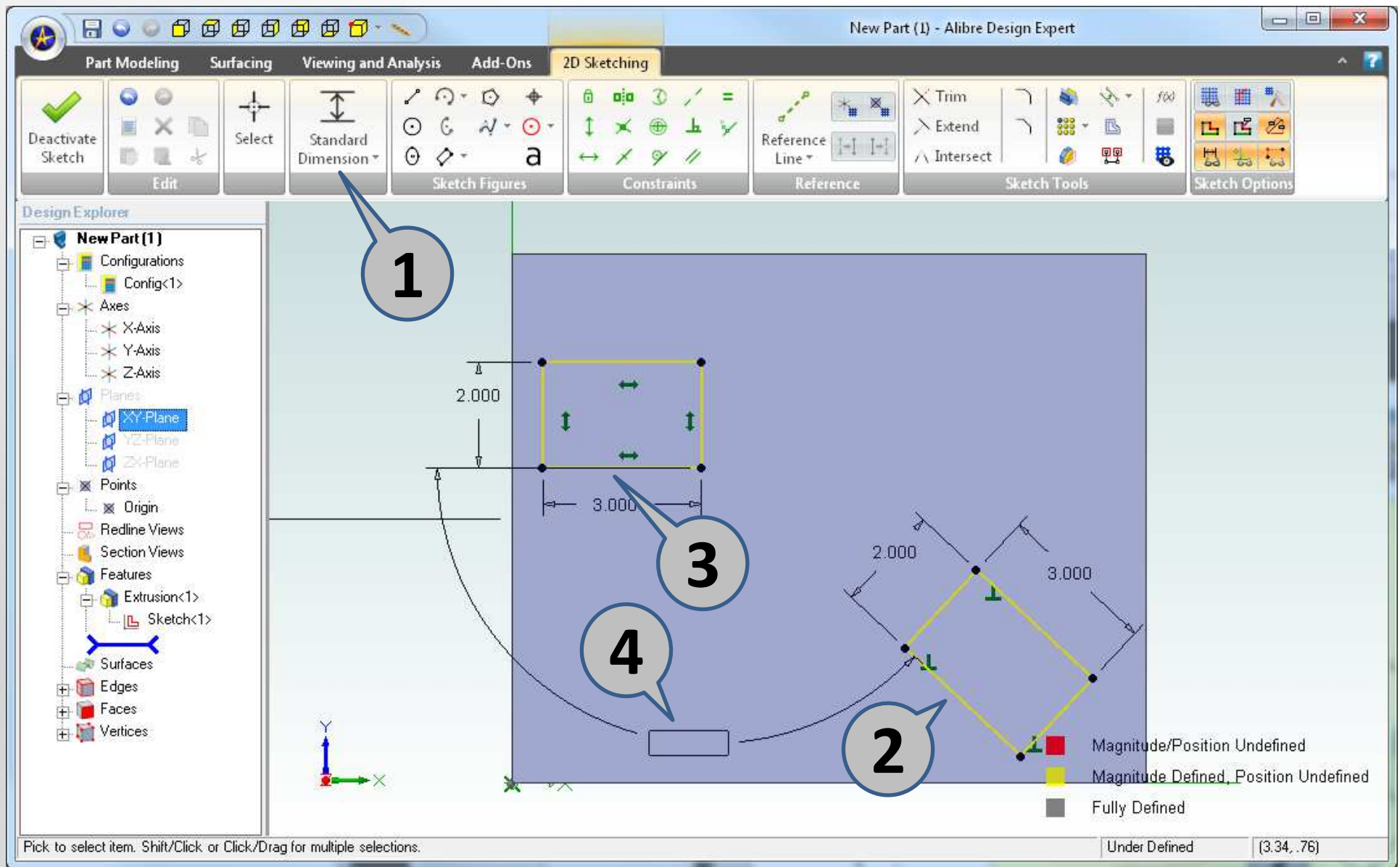
1. Enter **2** for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



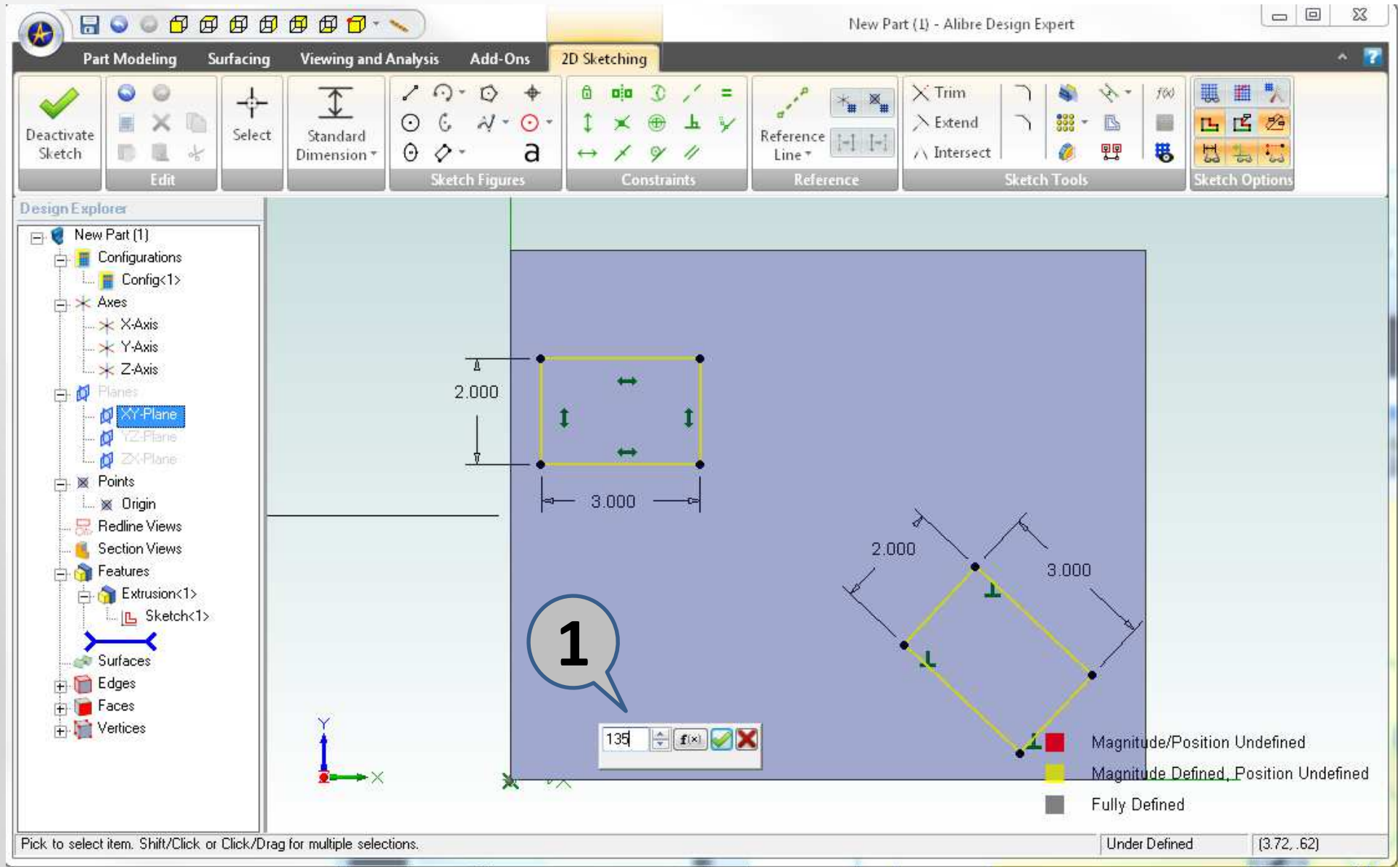
1. The **Dimension** tool should still be enabled from the previous steps. If it is not still enabled then click on the **Dimension** tool.
2. Click on the indicated segment to dimension.
3. Position the dimension as shown (approximately). Click and release to place the dimension.



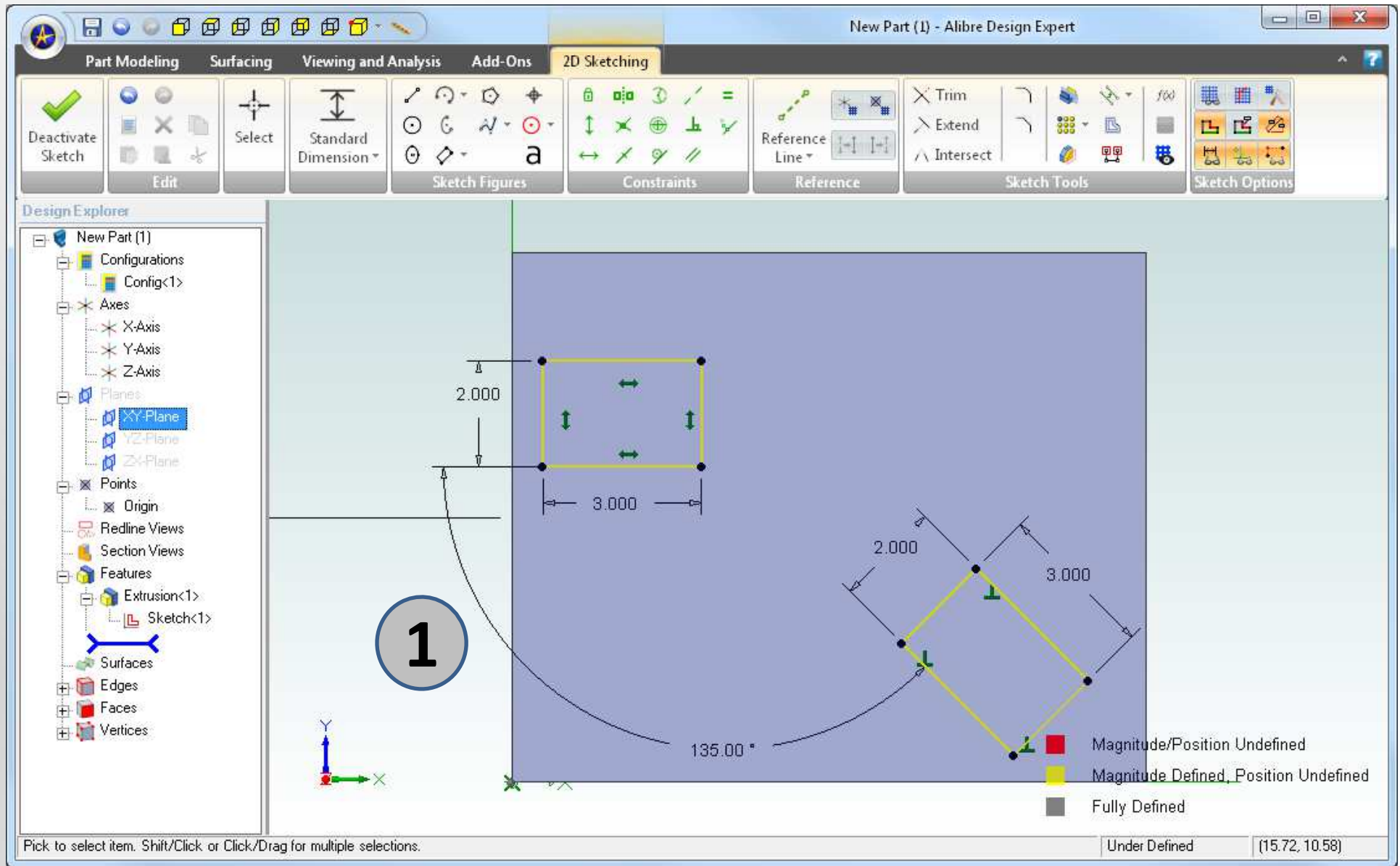
1. Enter **3** for the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



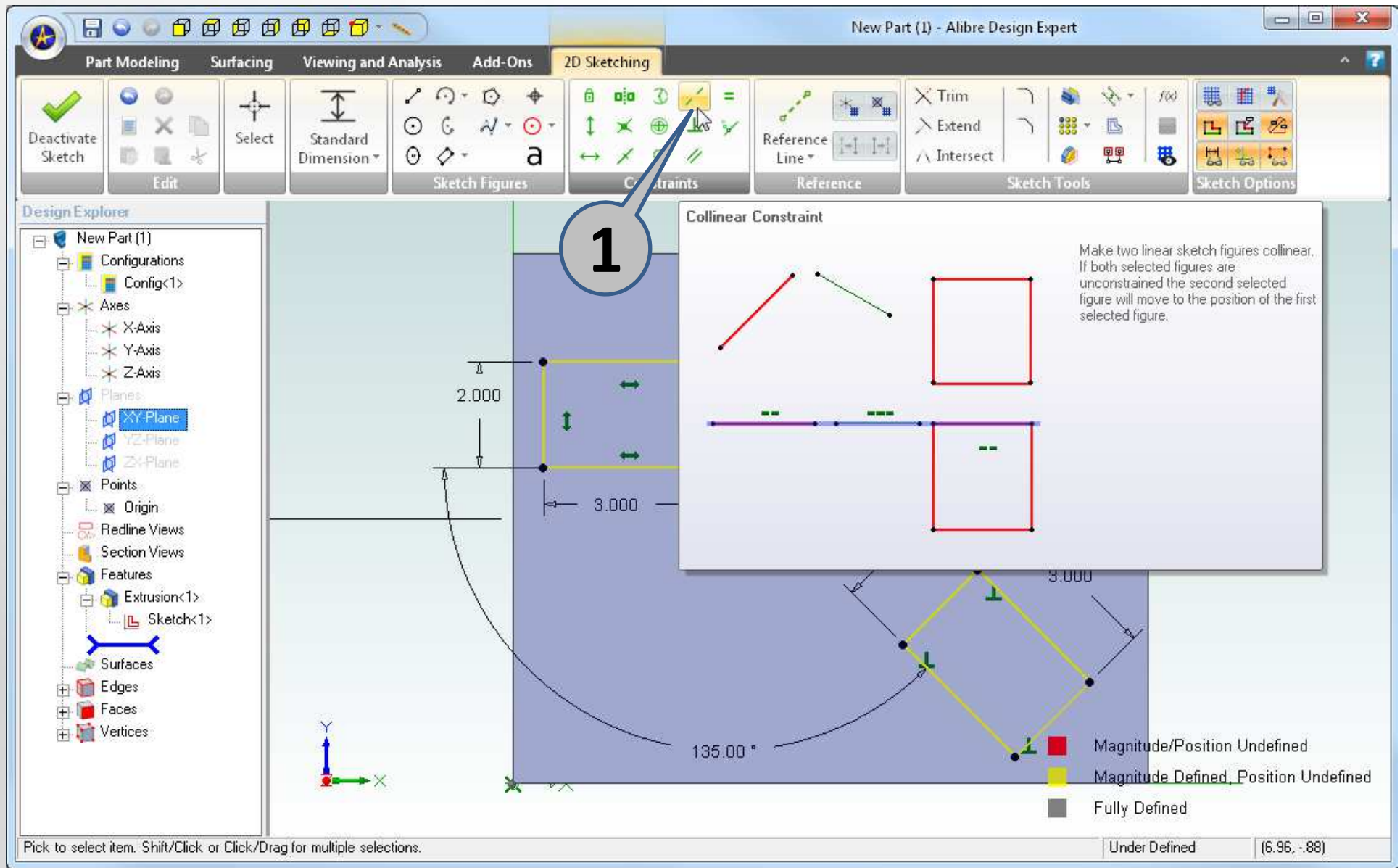
1. Click on the **Dimension** tool.
2. Click on the indicated segment.
3. Click on the indicated segment. The dimension now applies to the angle between the two selected segments.
4. Position the dimension as shown (approximately). Click and release to place the dimension.



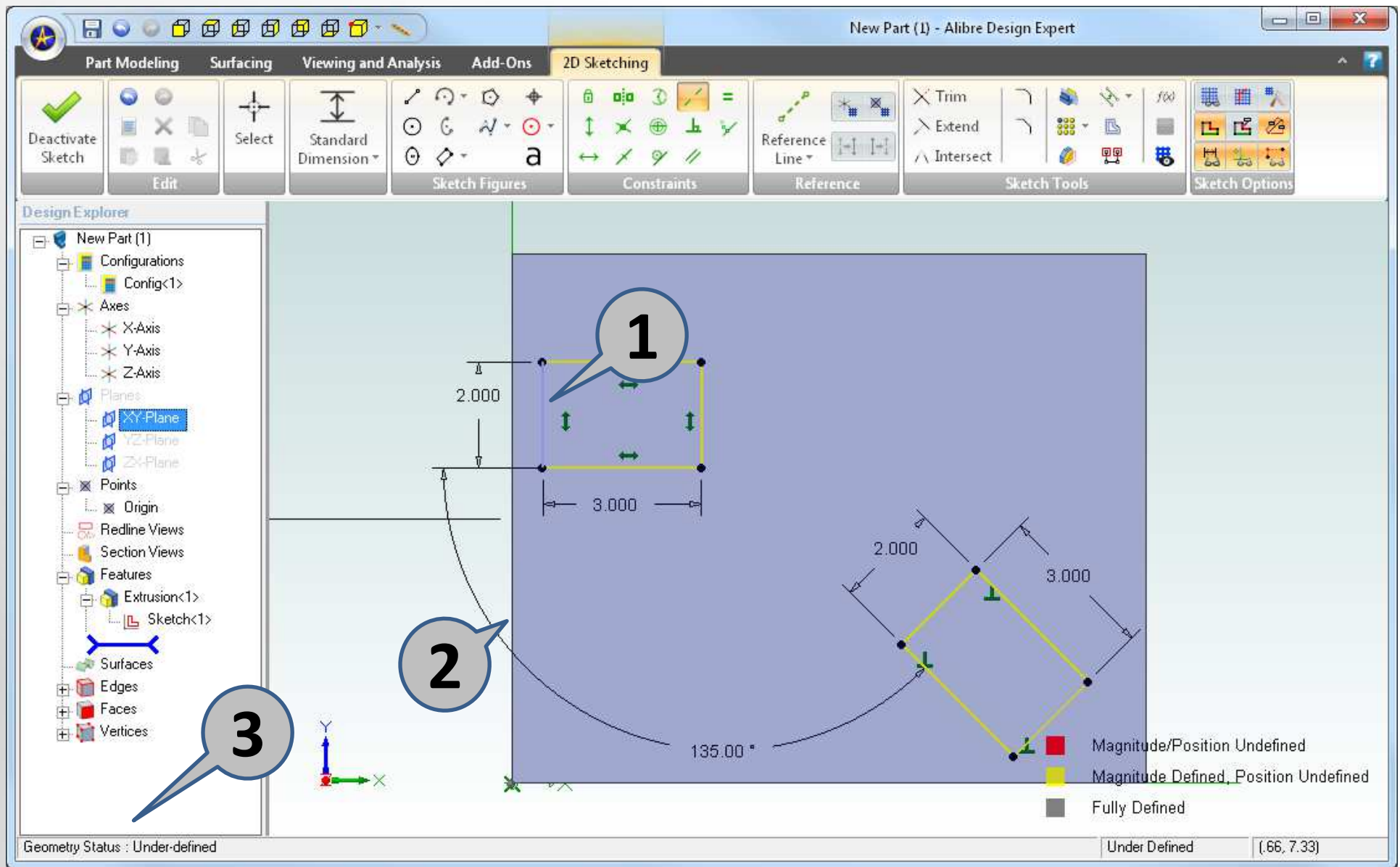
1. Enter **135** for the angular dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



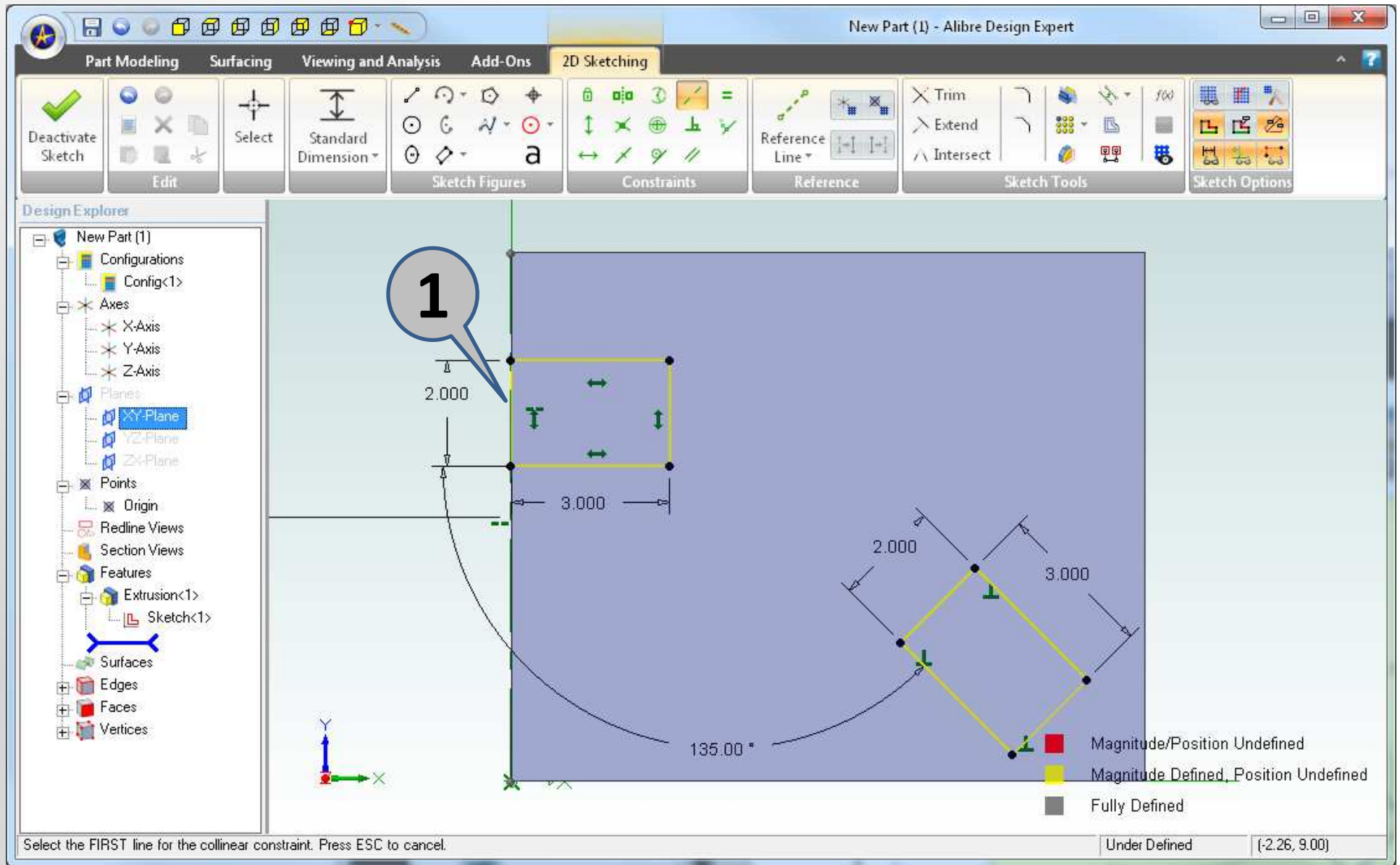
1. Confirm the results of the angular dimension. The angled line will rotate relative to the horizontal line.



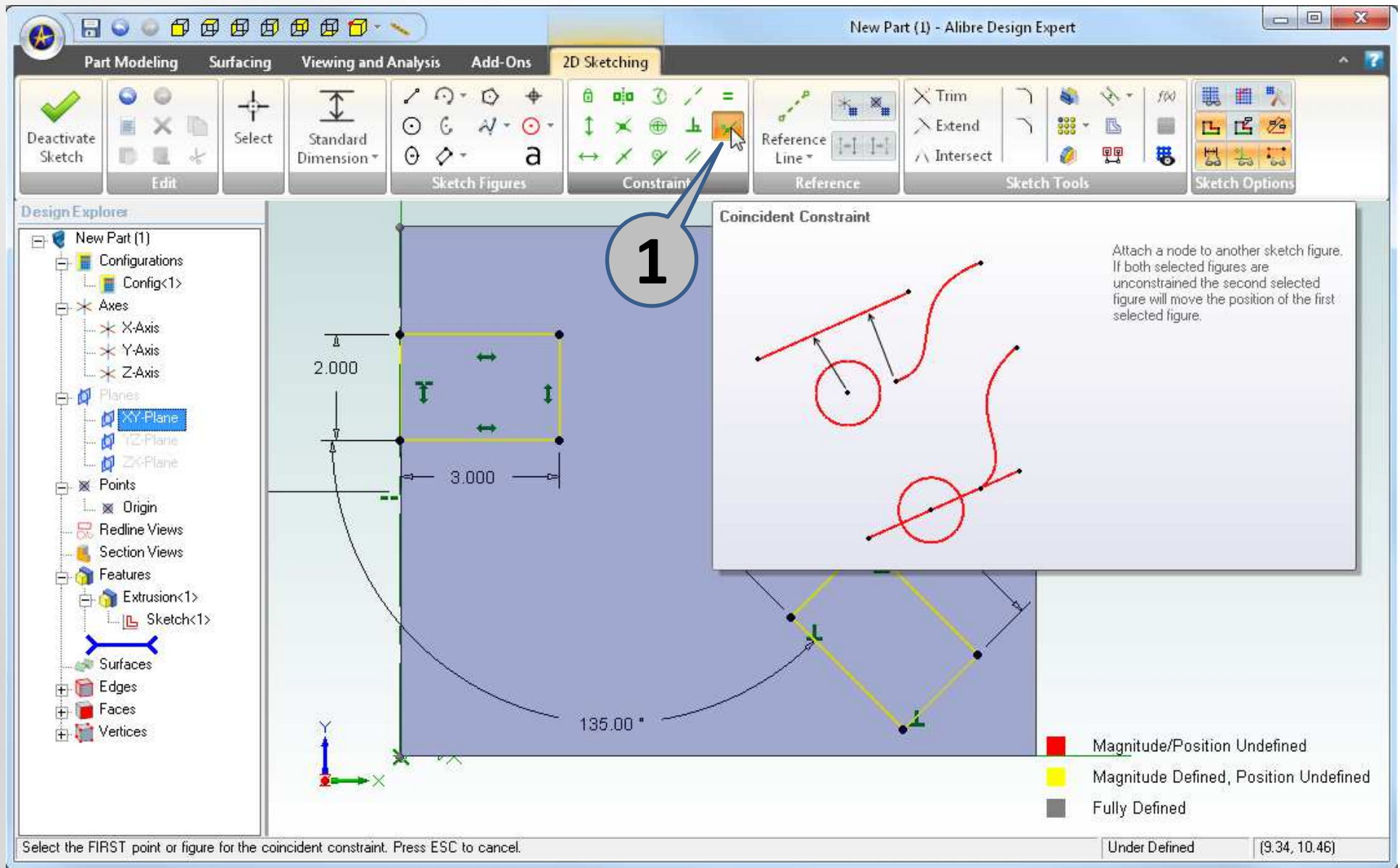
1. From the **Constraints** group, select the **Collinear** option.



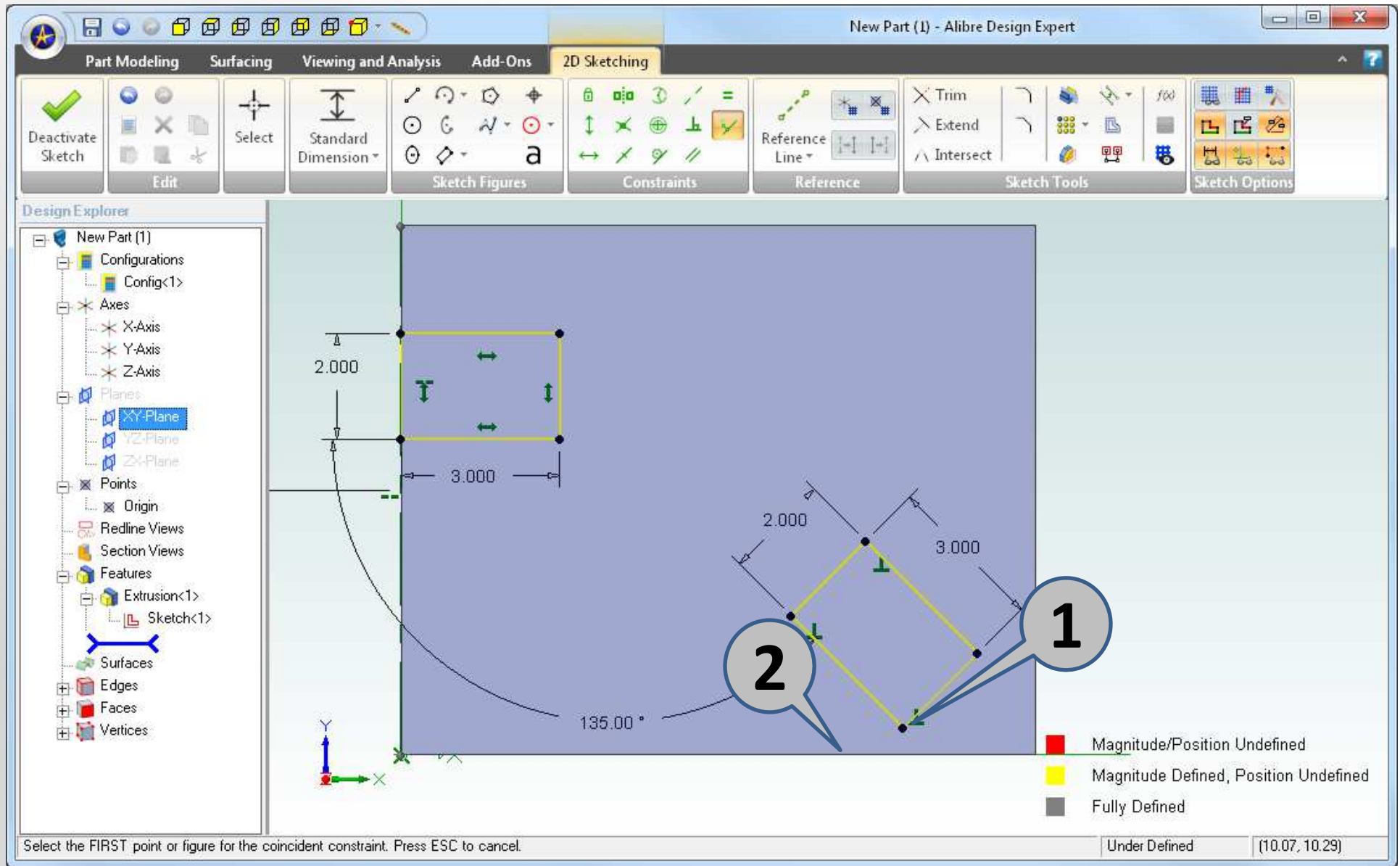
1. Click on the indicated line segment.
2. Click on the indicated **Edge** to complete the **Collinear** constraint.
3. When placing constraints Alibre Design will tell you what it expects.

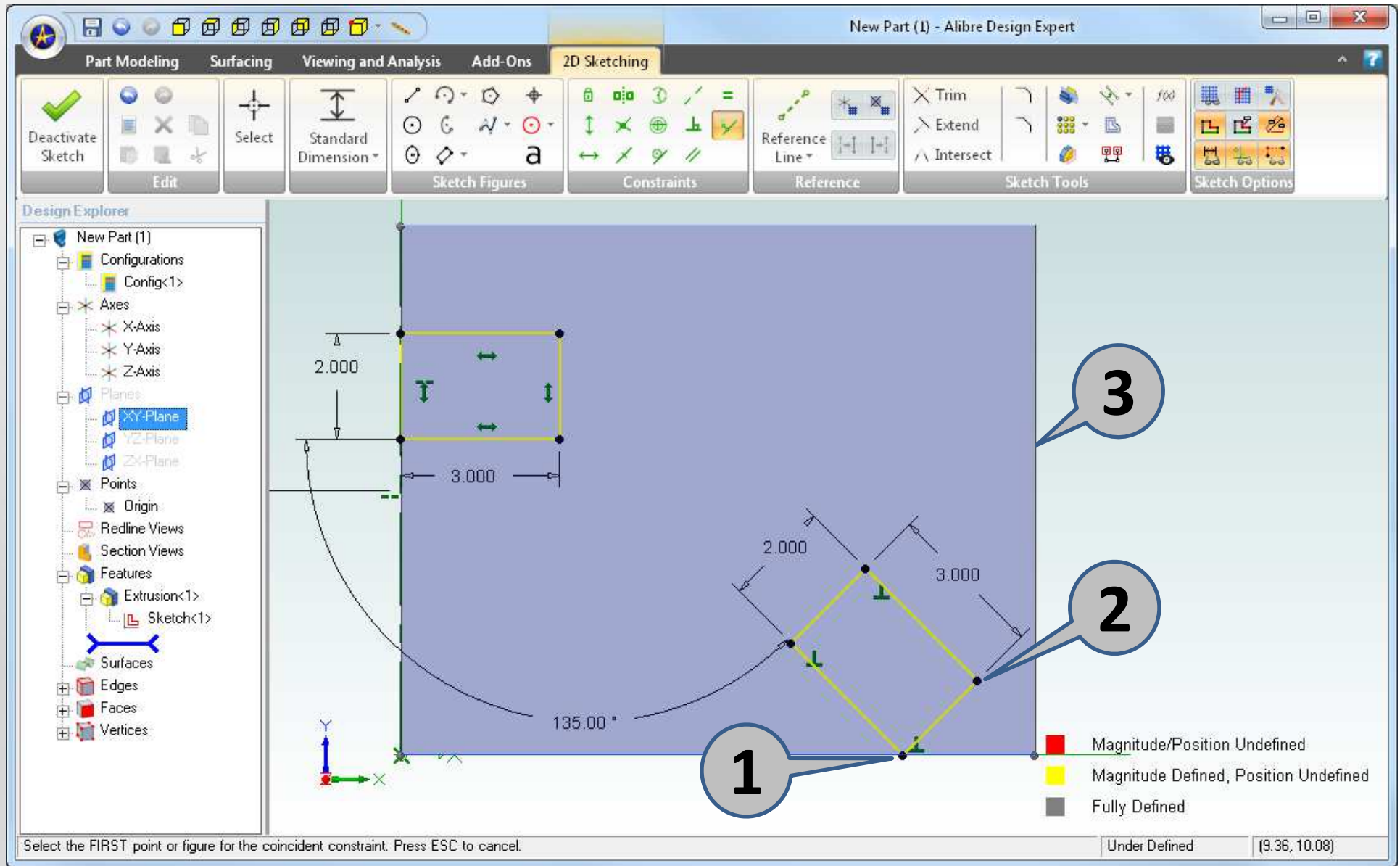


1. Confirm the results of the **Collinear** constraint.

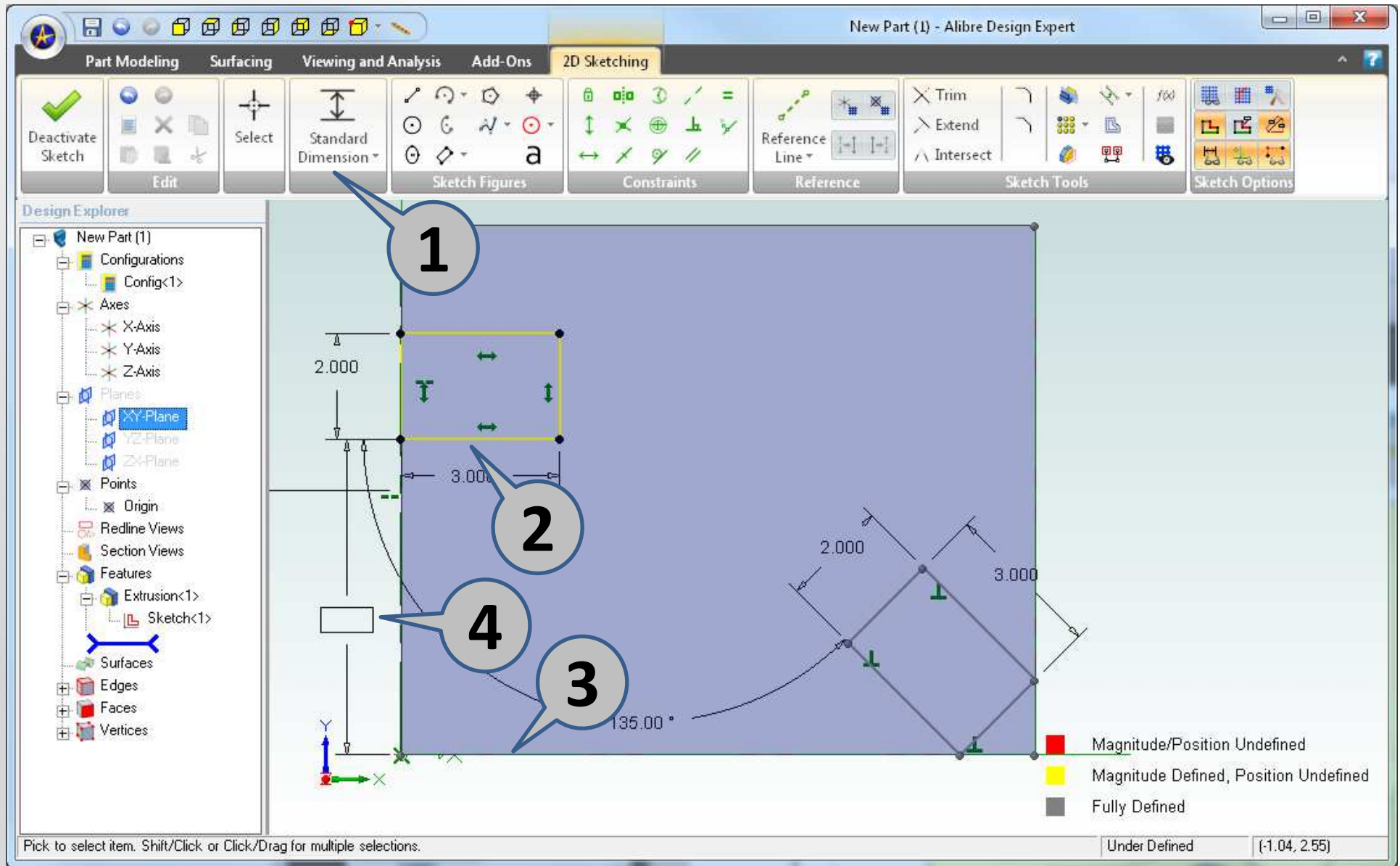


1. From the **Constraints** group, select the **Coincident** option.

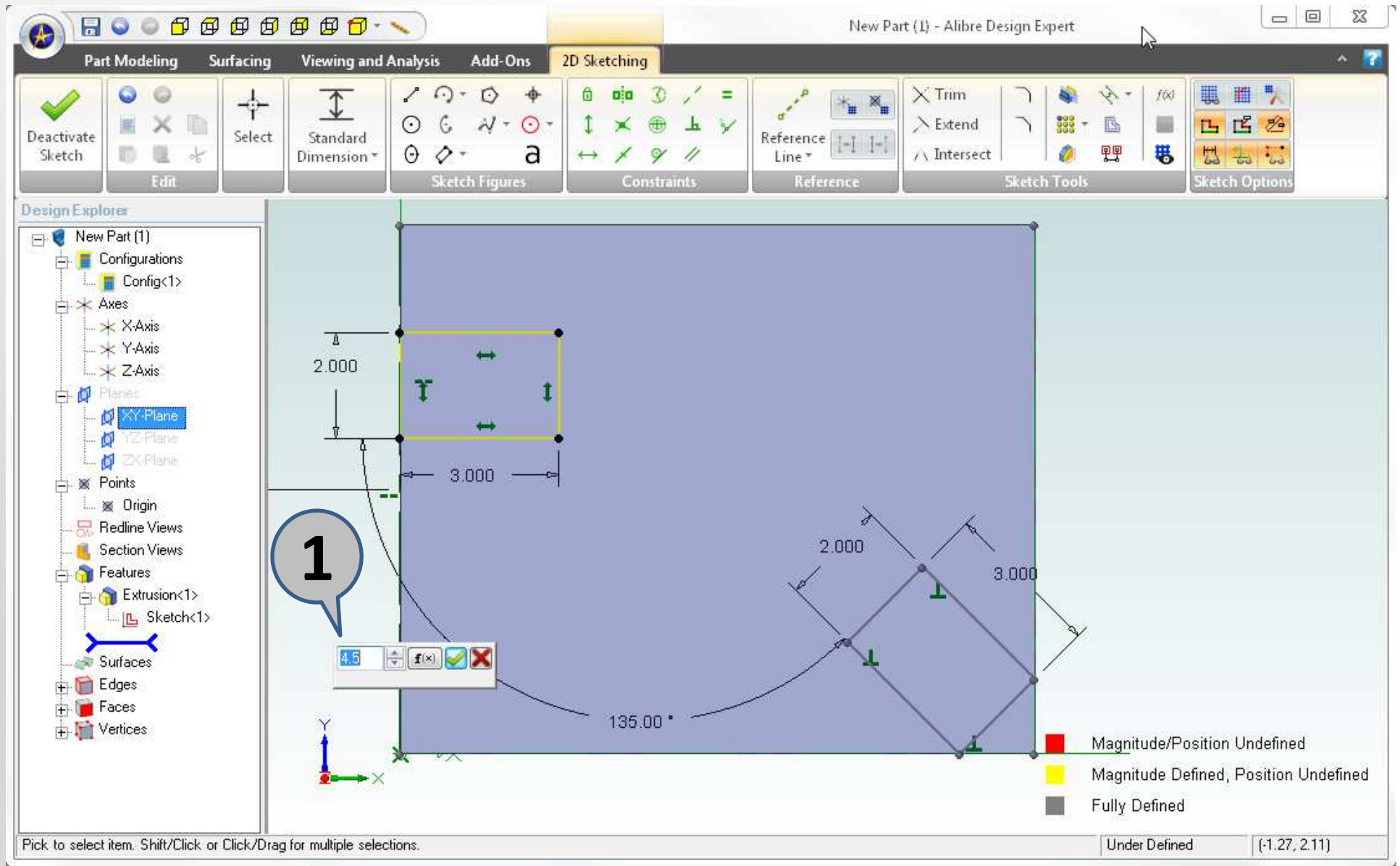




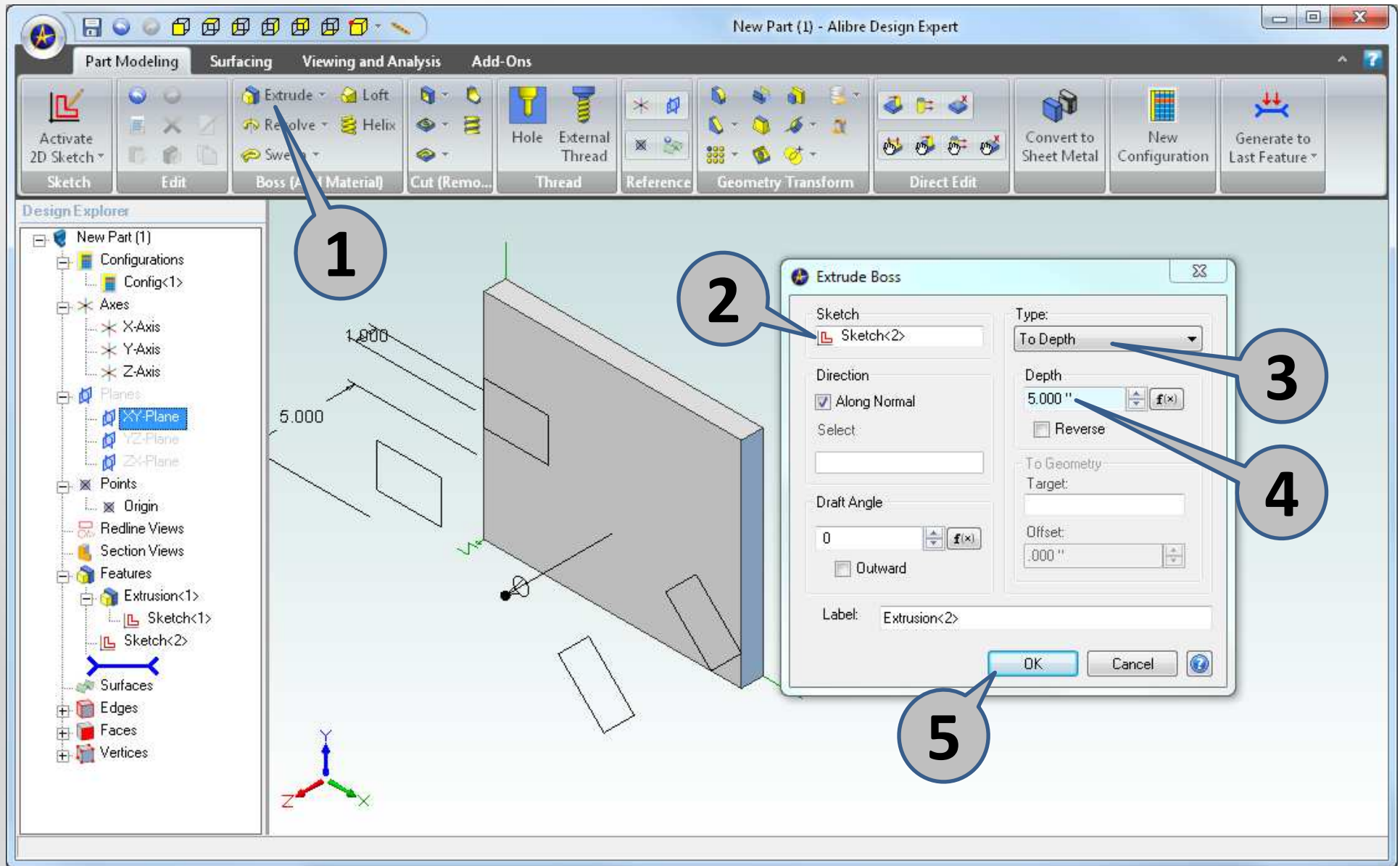
1. Confirm the result of the previous **Coincident** constraint.
2. Click on the indicated **Node**.
3. Click on the indicated **Edge** to complete the **Coincident** constraint.



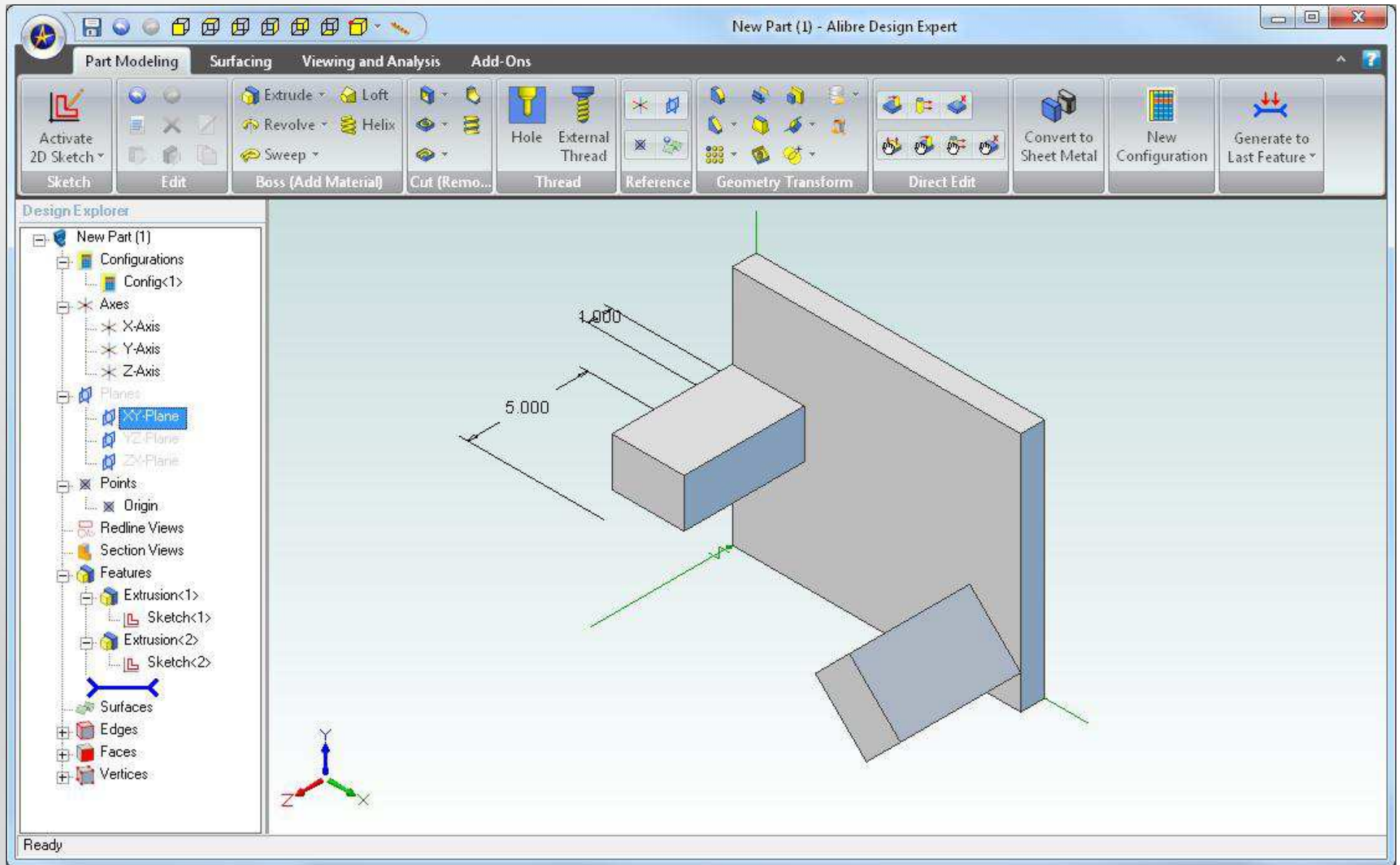
1. Click on the **Dimension** tool.
2. Click on the segment indicated.
3. Click on the **Edge** indicated.
4. Position the dimension as shown. Click and release to place it.



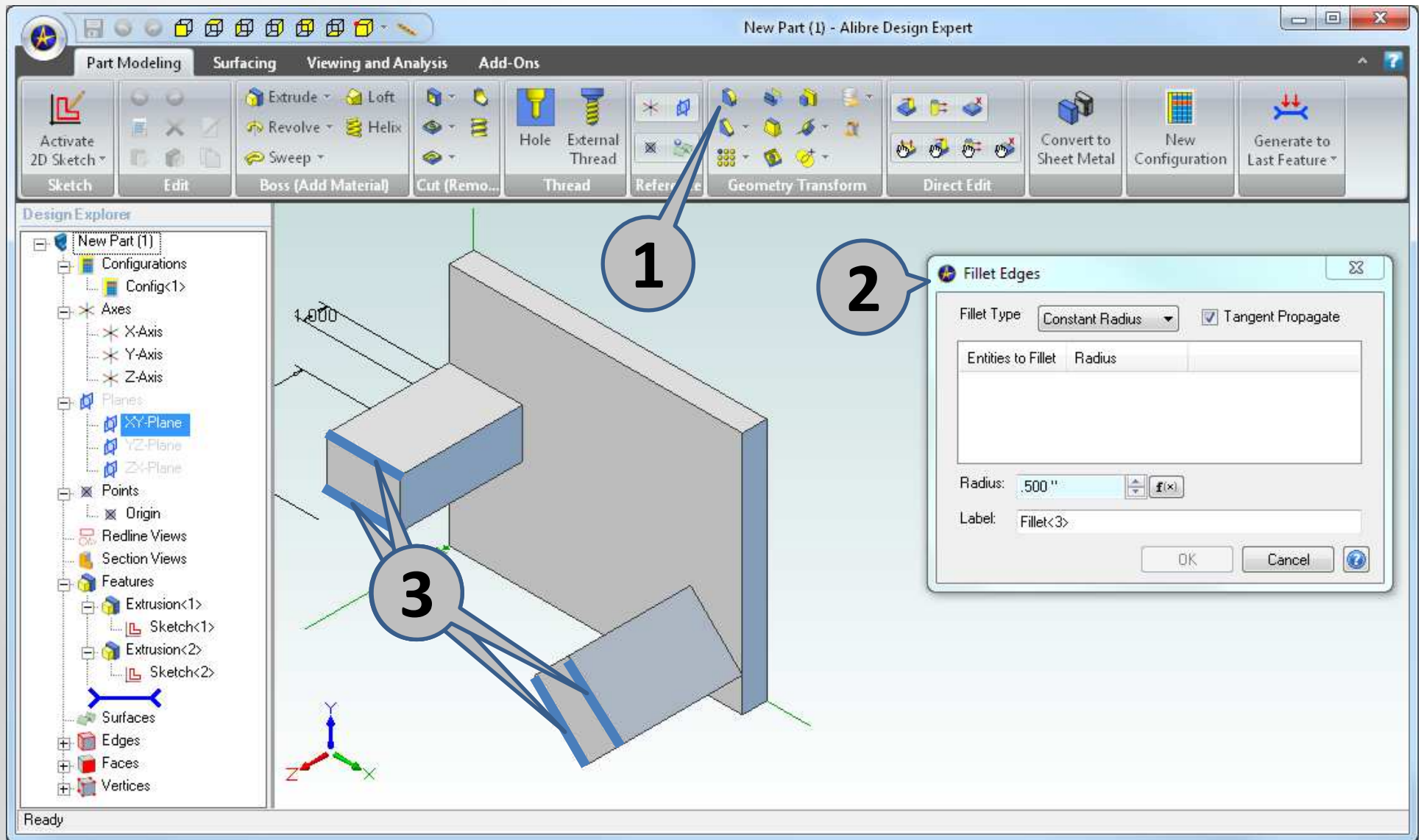
1. Enter **4.5** as the dimension value. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



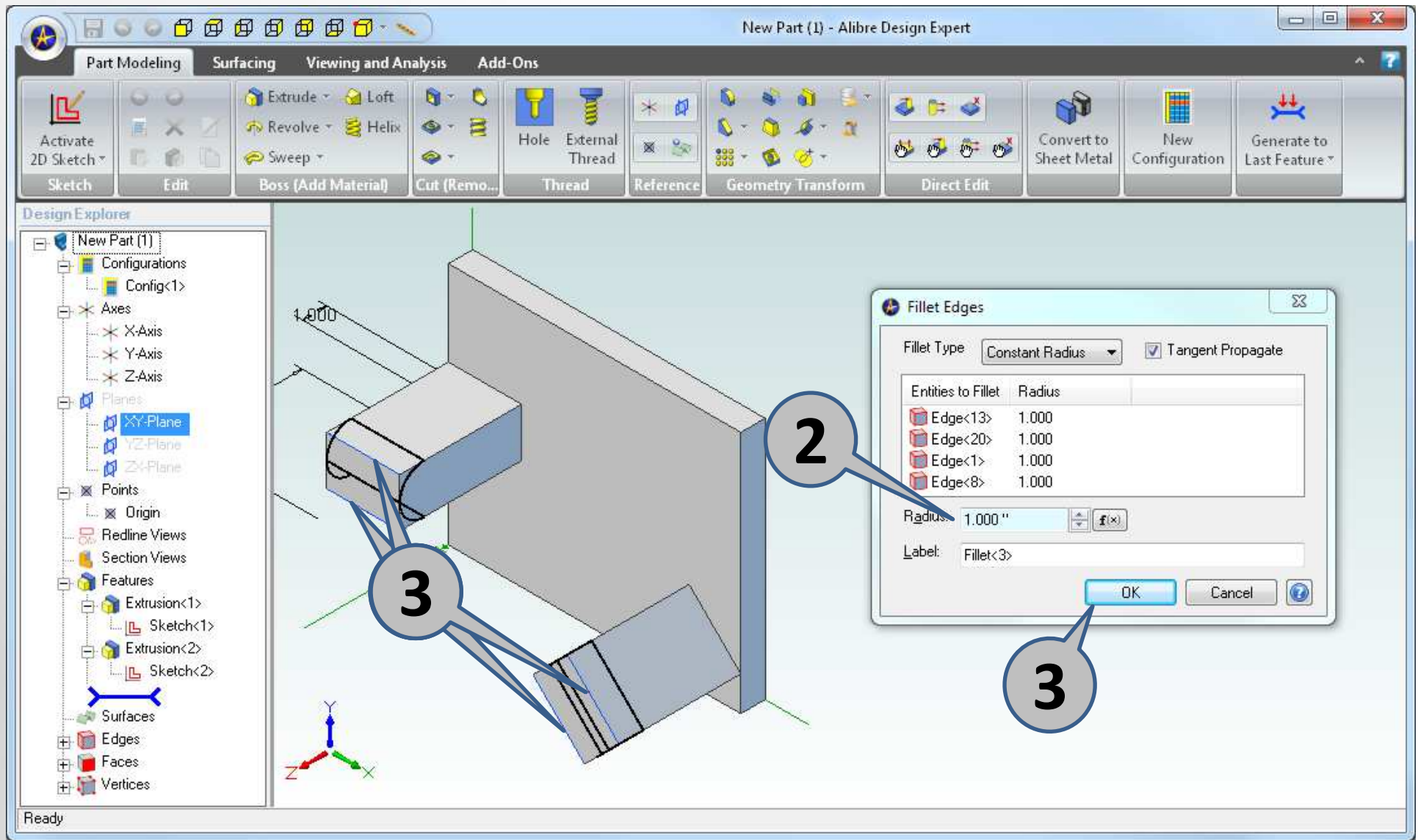
1. Click on the **Part Modeling** tab in the ribbon and click on the **Extrude Boss** icon.
2. Confirm that **Sketch<2>** is the **Sketch** to be extruded.
3. Confirm **To Depth** is **Type** of extrusion.
4. Enter a value of **5 inches** in the **Depth** entry box.
5. Click **OK**.

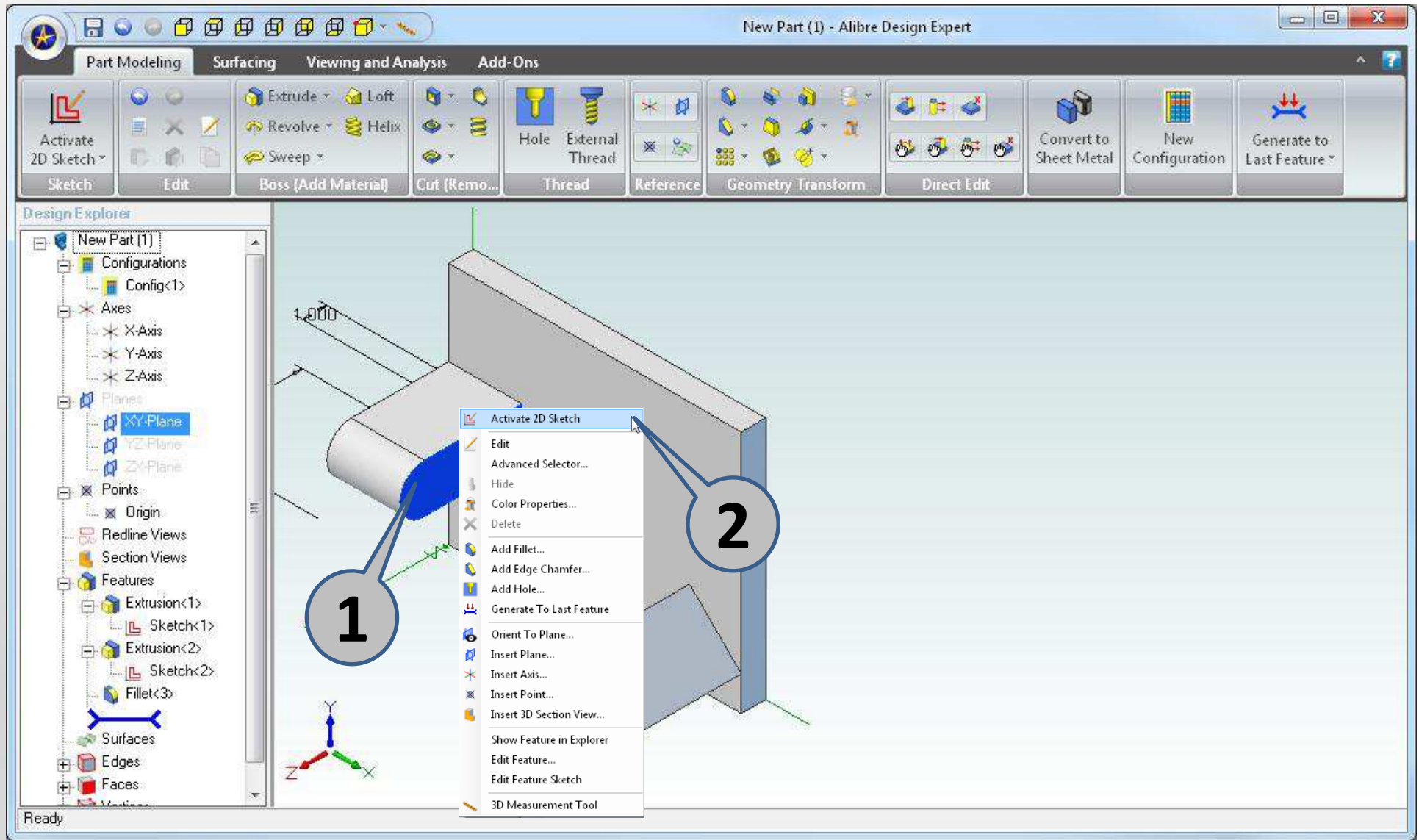


1. Confirm similar results for the **Extrude Boss** operation.

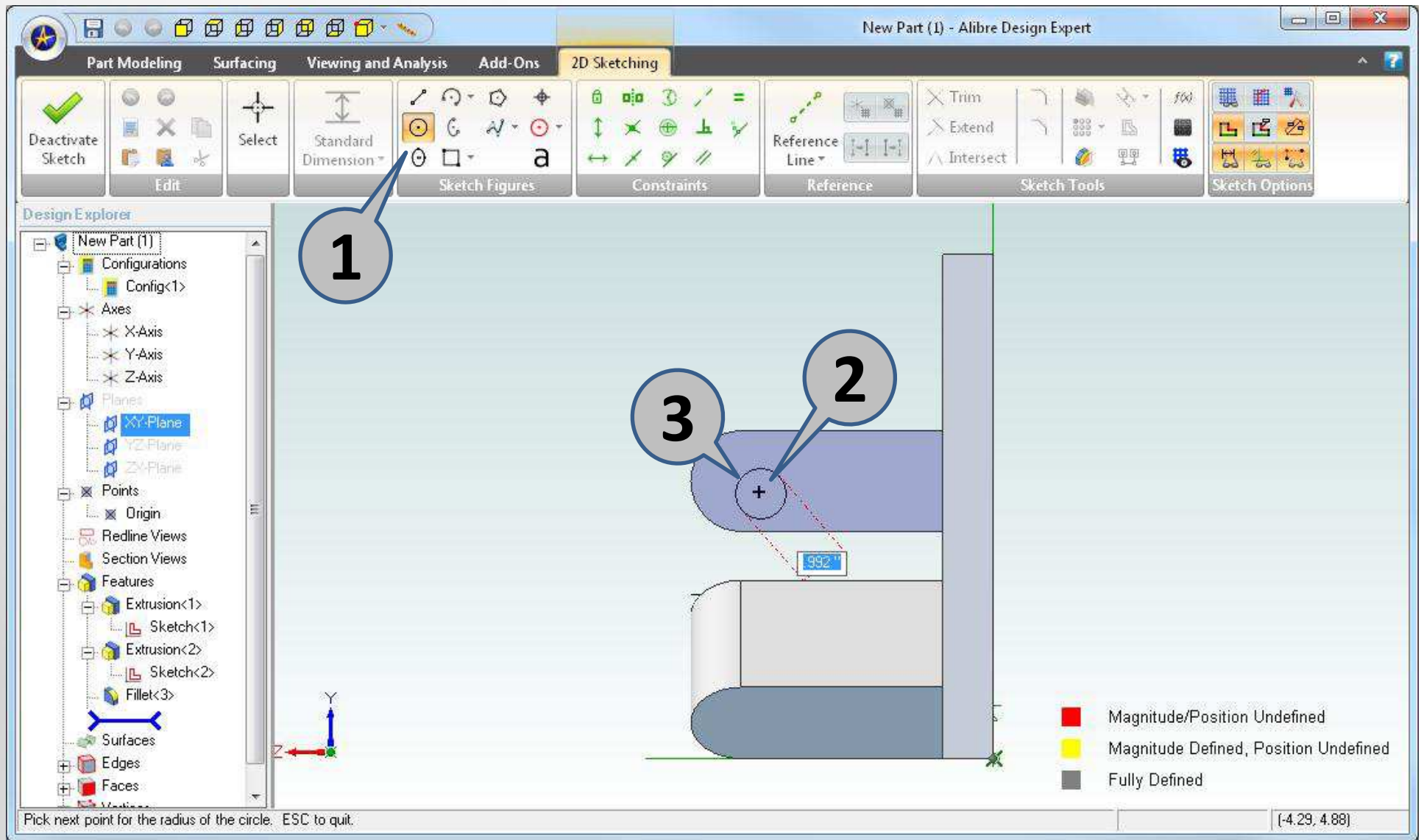


1. Click on the **Fillet** icon in the **Geometry Transform** group.
2. Position the **Fillet Edges** dialog box such that the entire part is visible.
3. Click on the edges indicated.

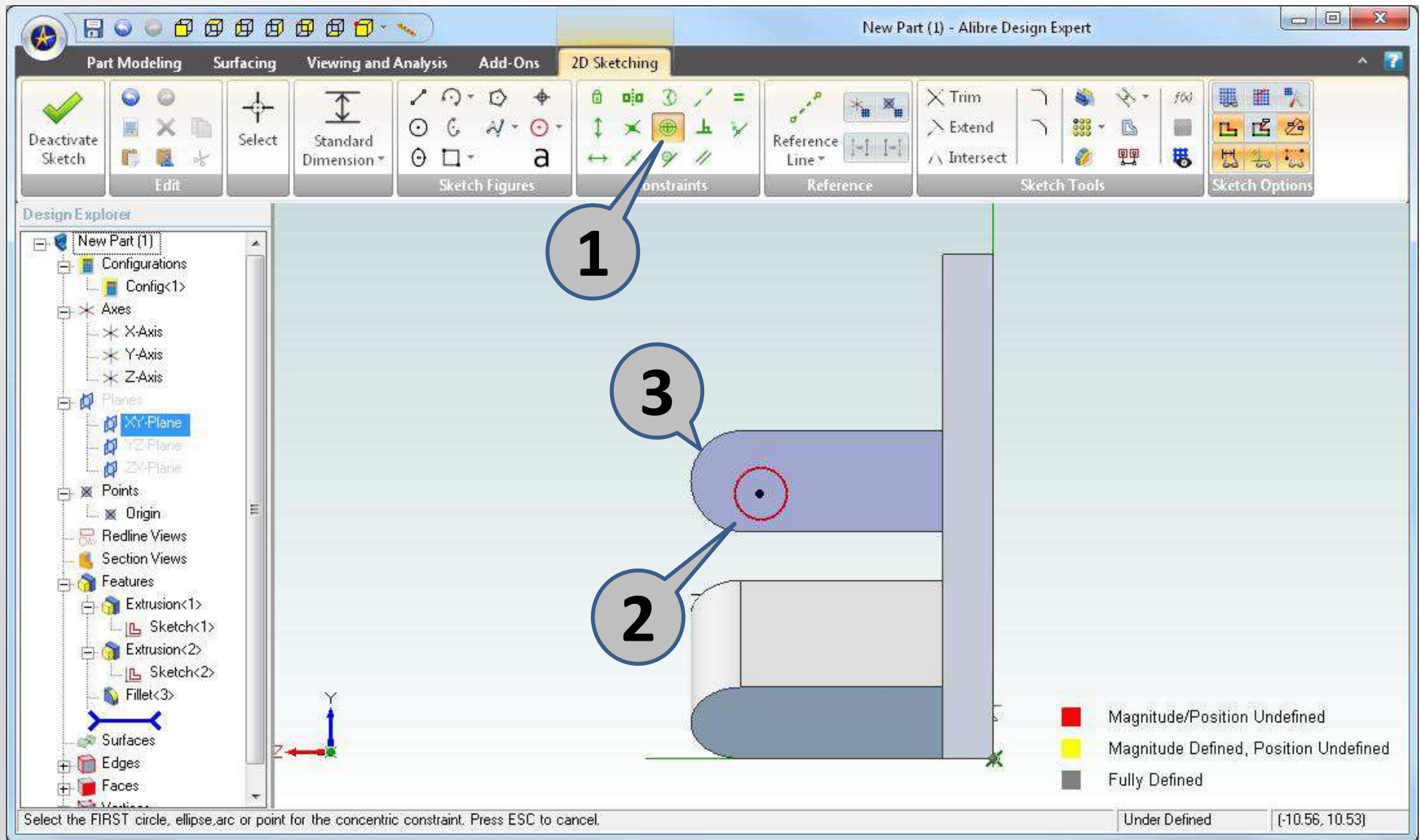




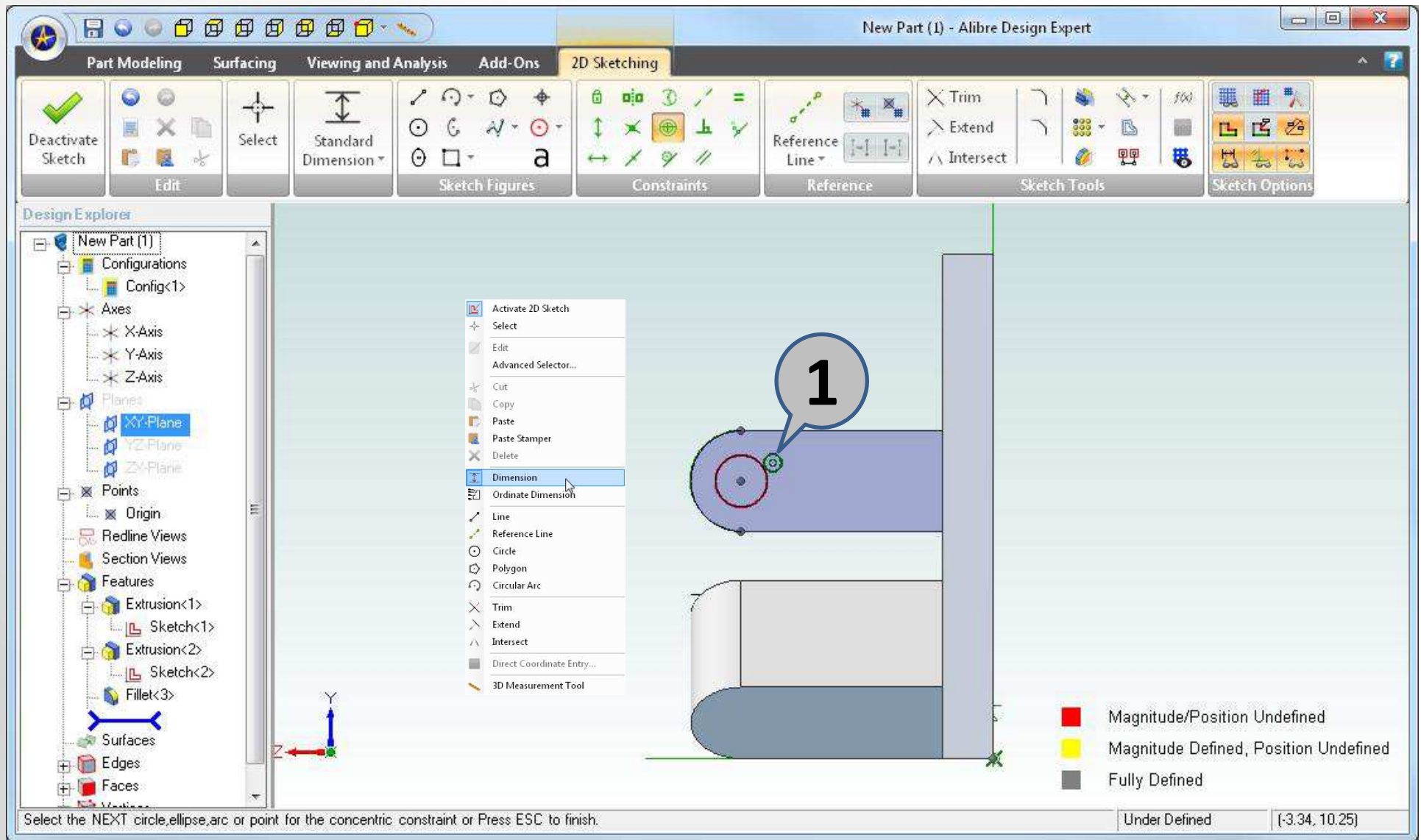
1. Right click on the face indicated.
2. From the popup menu select **Activate 2D Sketch**.



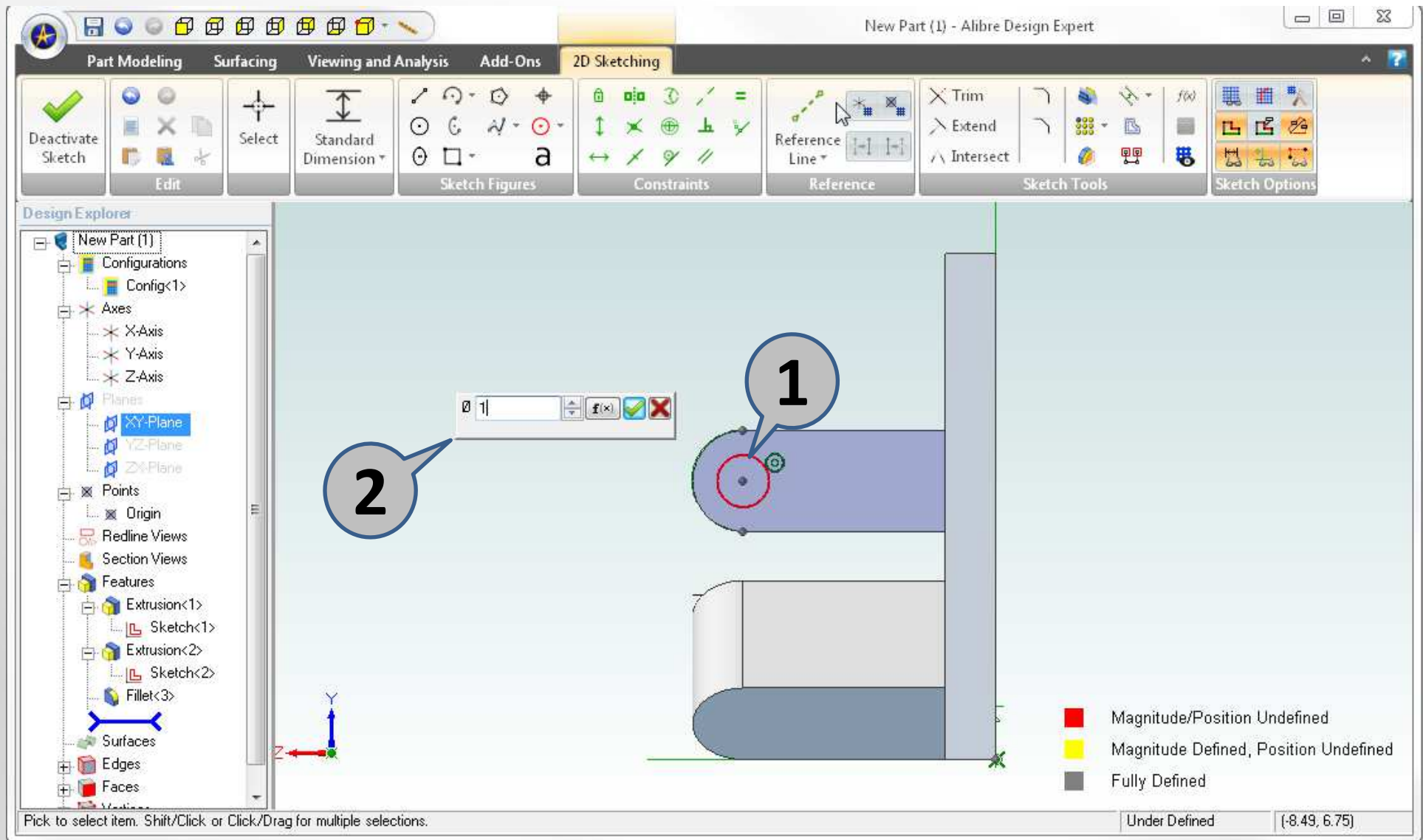
1. Click on the **Circle** tool.
2. Position the mouse cursor as shown (approximately). Click and release to create the center.
3. Move the mouse to the position indicated (approximately). Click and release to complete the circle.



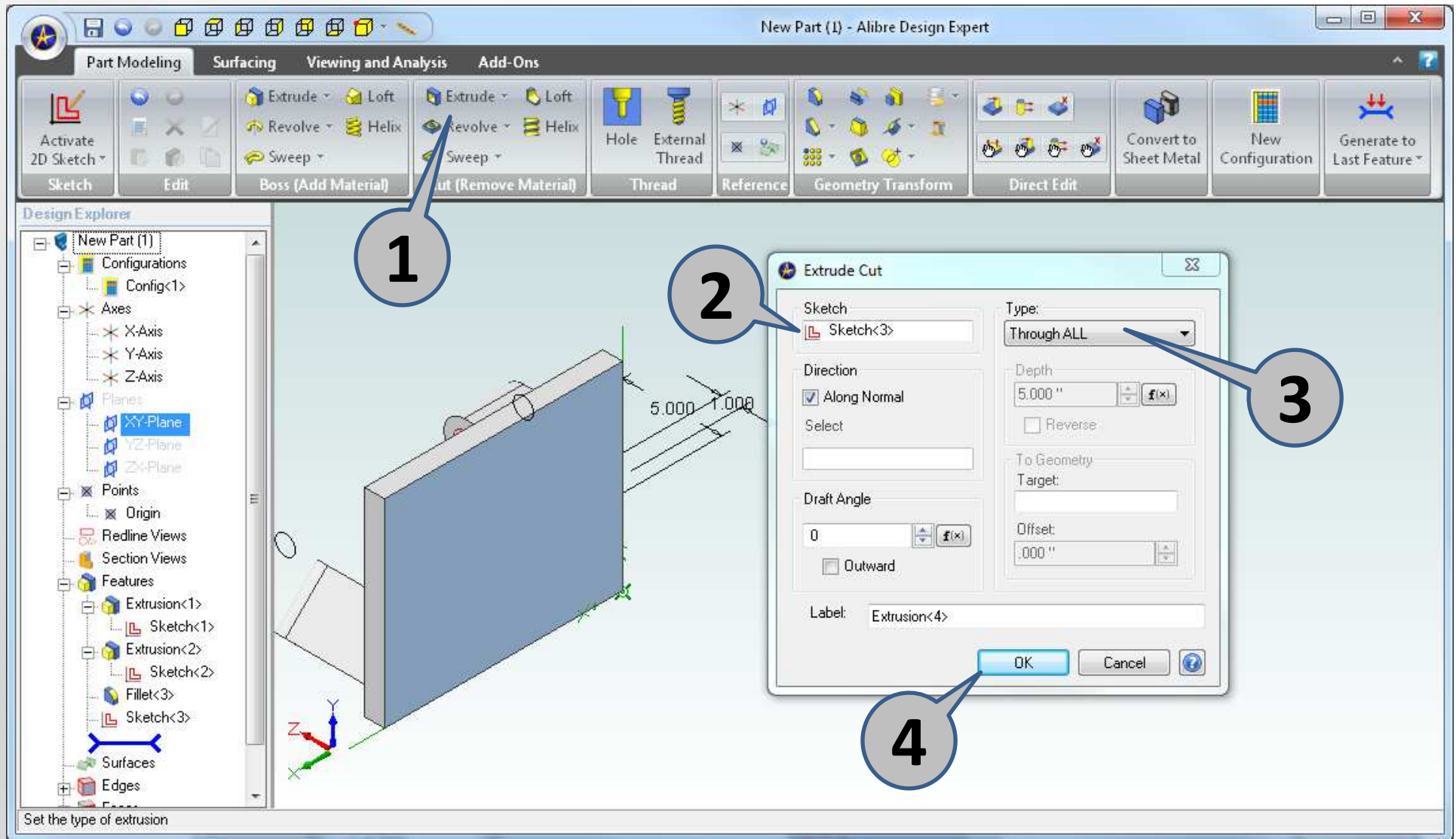
1. Select the **Concentric** sketch constraint.
2. Click on the perimeter of the circle.
3. Click on the circular edge shown in the screen capture.



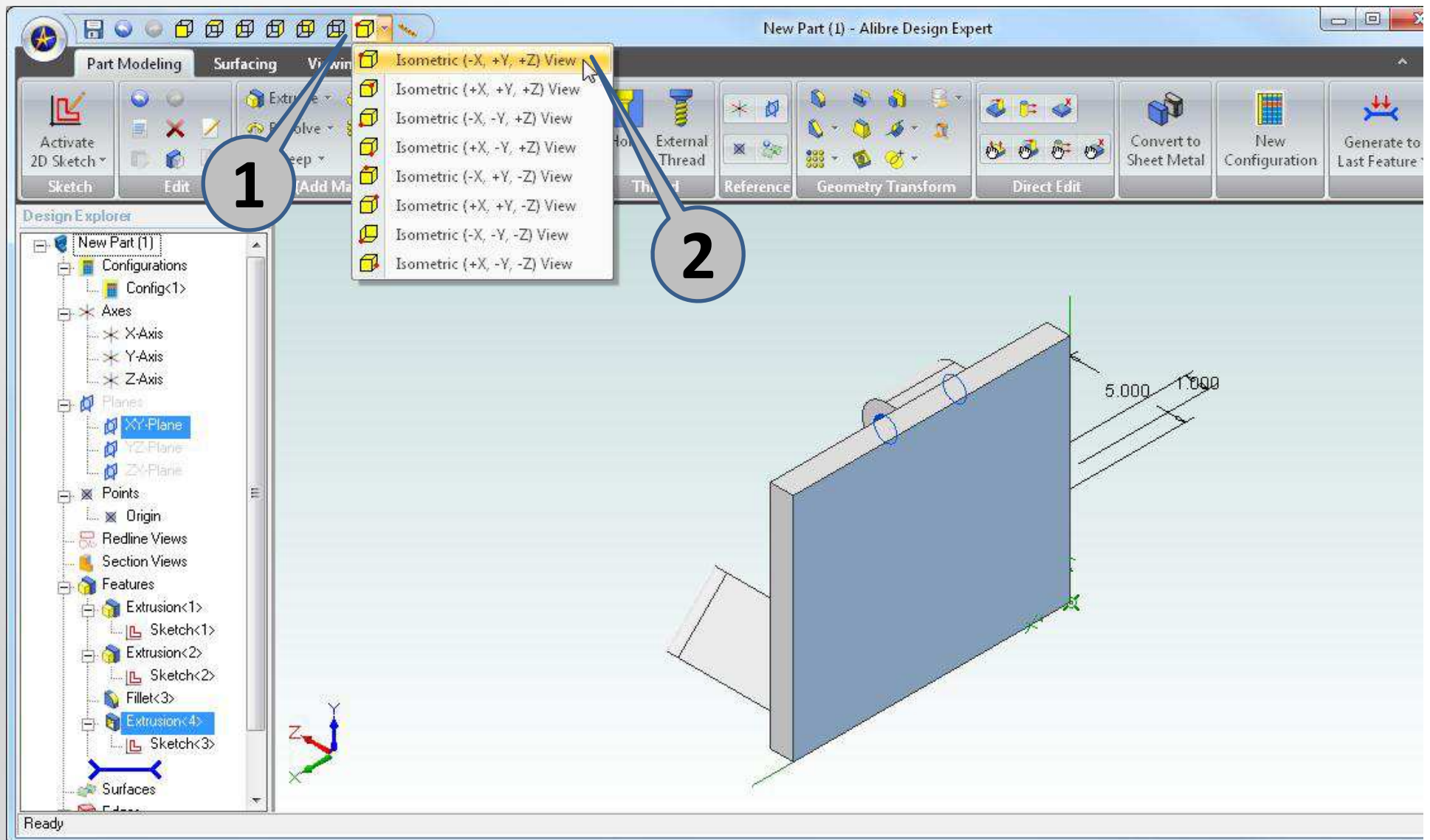
1. Confirm that the result of the **Concentric** constraint is similar to the screen capture. Take note of the **Concentric** constraint symbol.
2. Right click and select **Dimension** from the popup menu.



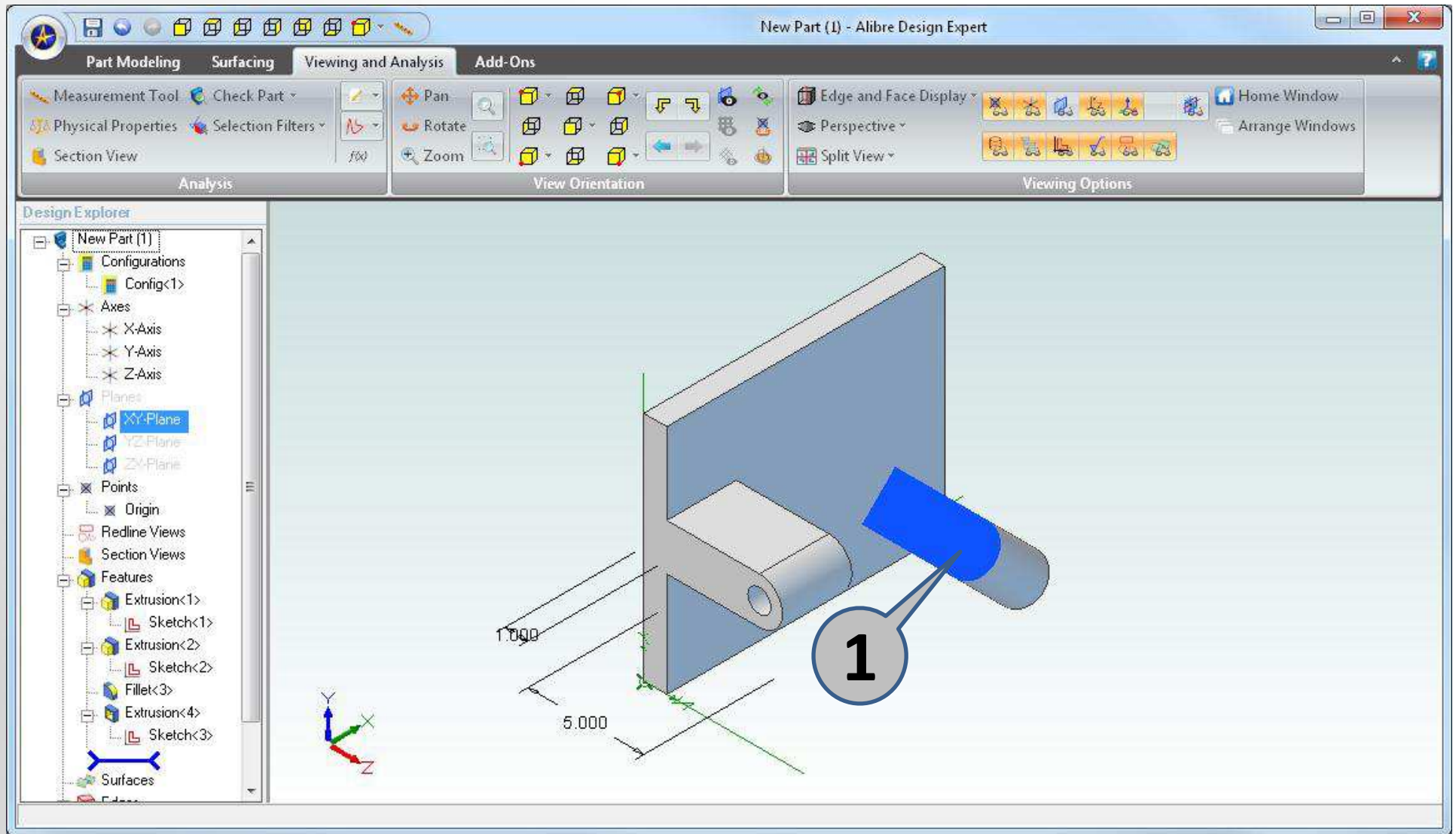
1. Click on the perimeter of the circle.
2. Position the dimension approximately as shown. Click and release to place the dimension. Enter a value of **1** for the dimension.
3. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



1. Click on the **Part Modeling** tab in the ribbon and click on the **Extrude Cut** icon.
2. Confirm that the **Sketch<3>** is the **Sketch** to be extruded.
3. Select **Through All** as the **Type** of extrusion.
4. Click **OK**.

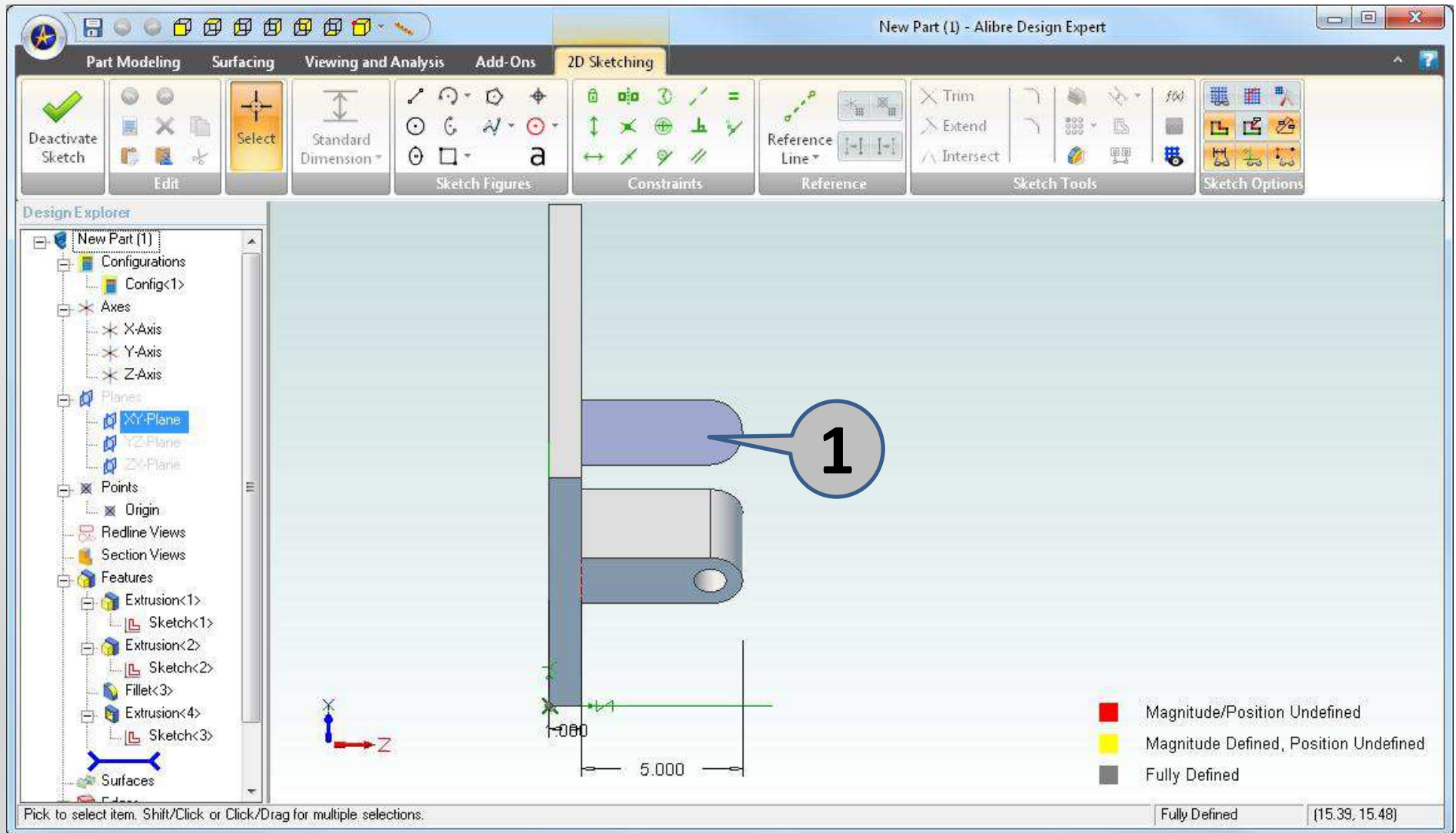


1. From the Quick Access toolbar
2. Select the first **Isometric** option.

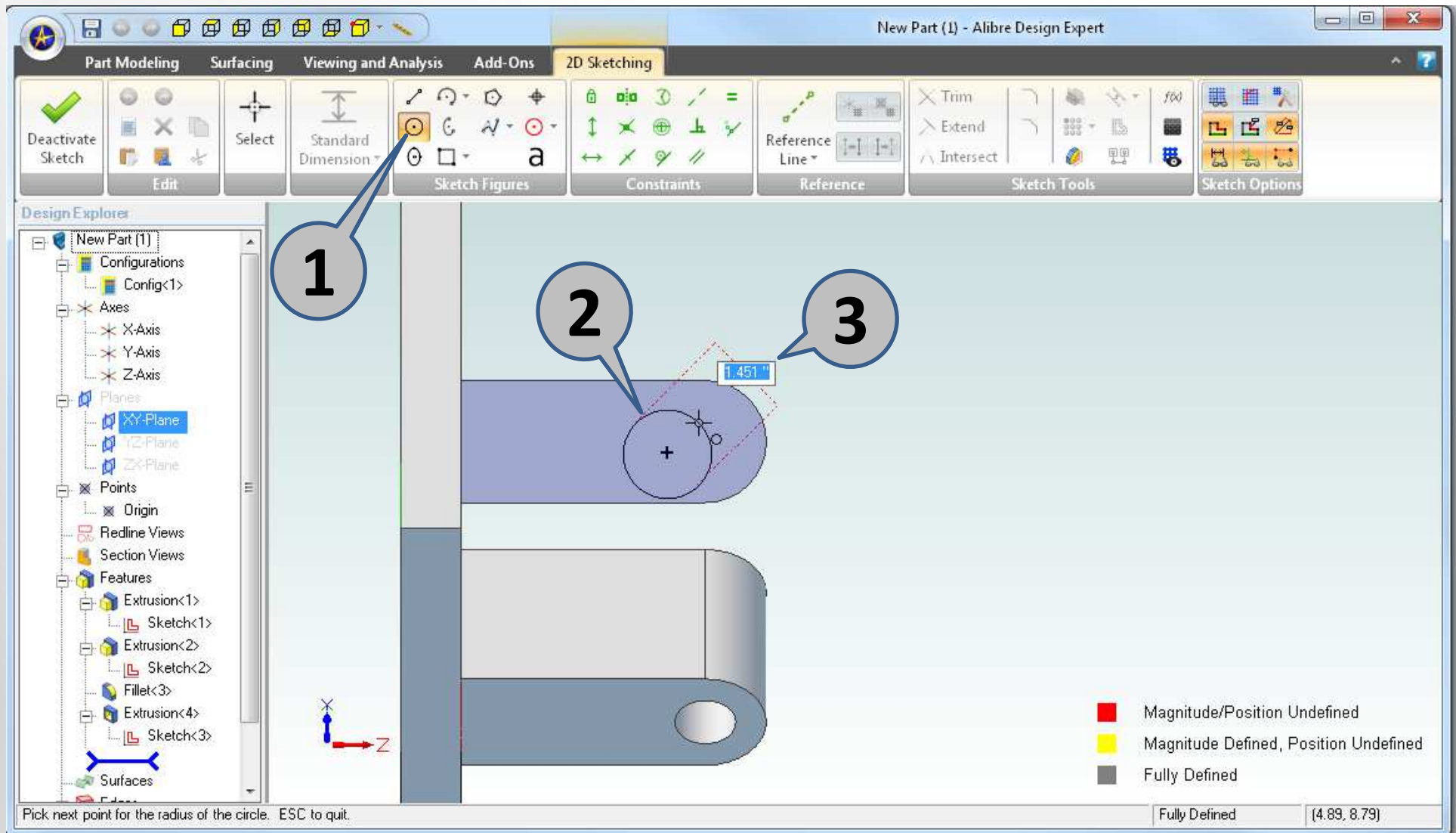


1. Click on the face indicated and enter into sketch mode.

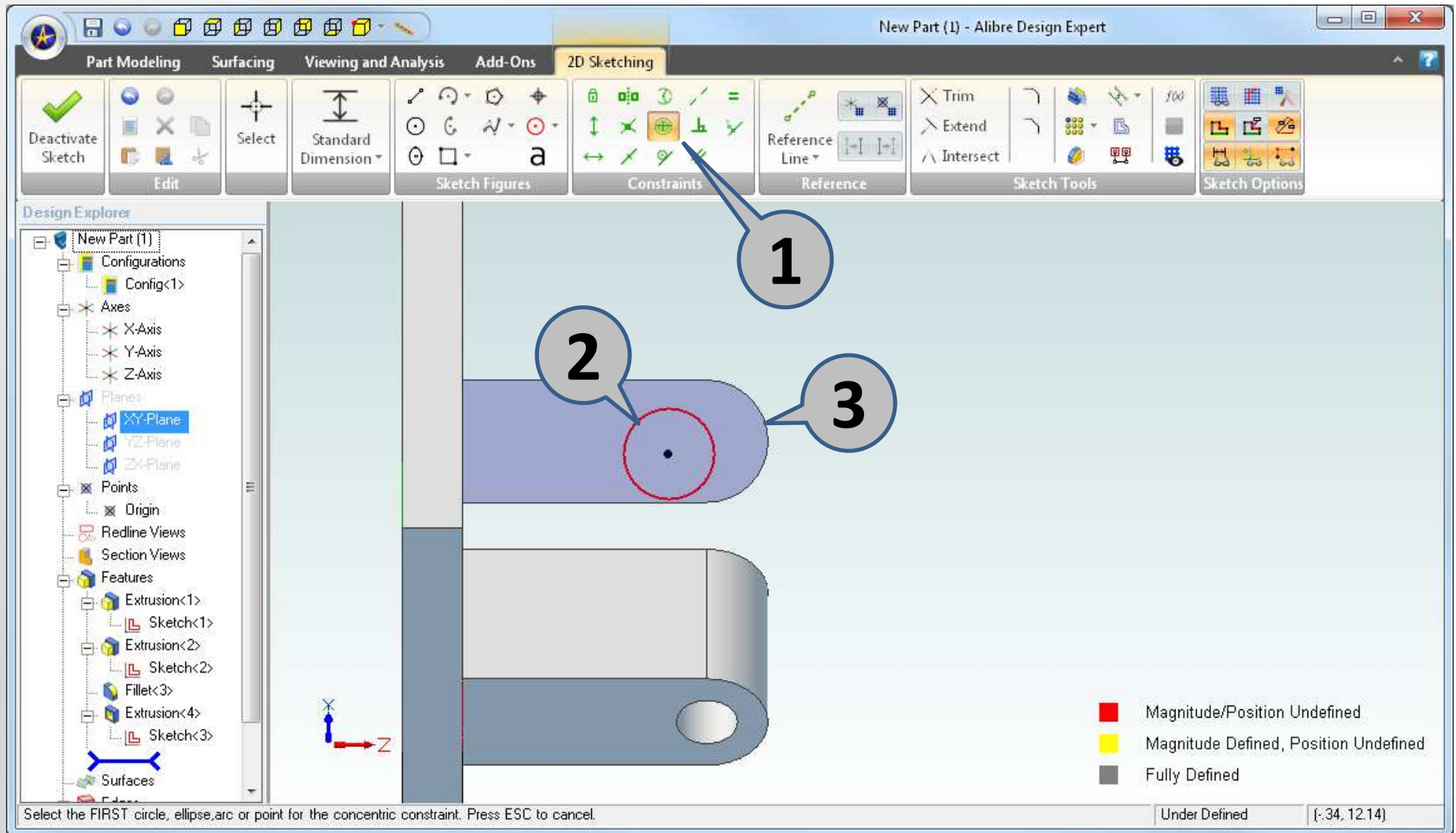
You can do this either by clicking on the **Part Modeling** tab in the ribbon and then click on the **Activate 2D Sketch** icon, or you can right click in the workspace and select **Activate 2D Sketch** from the popup menu.



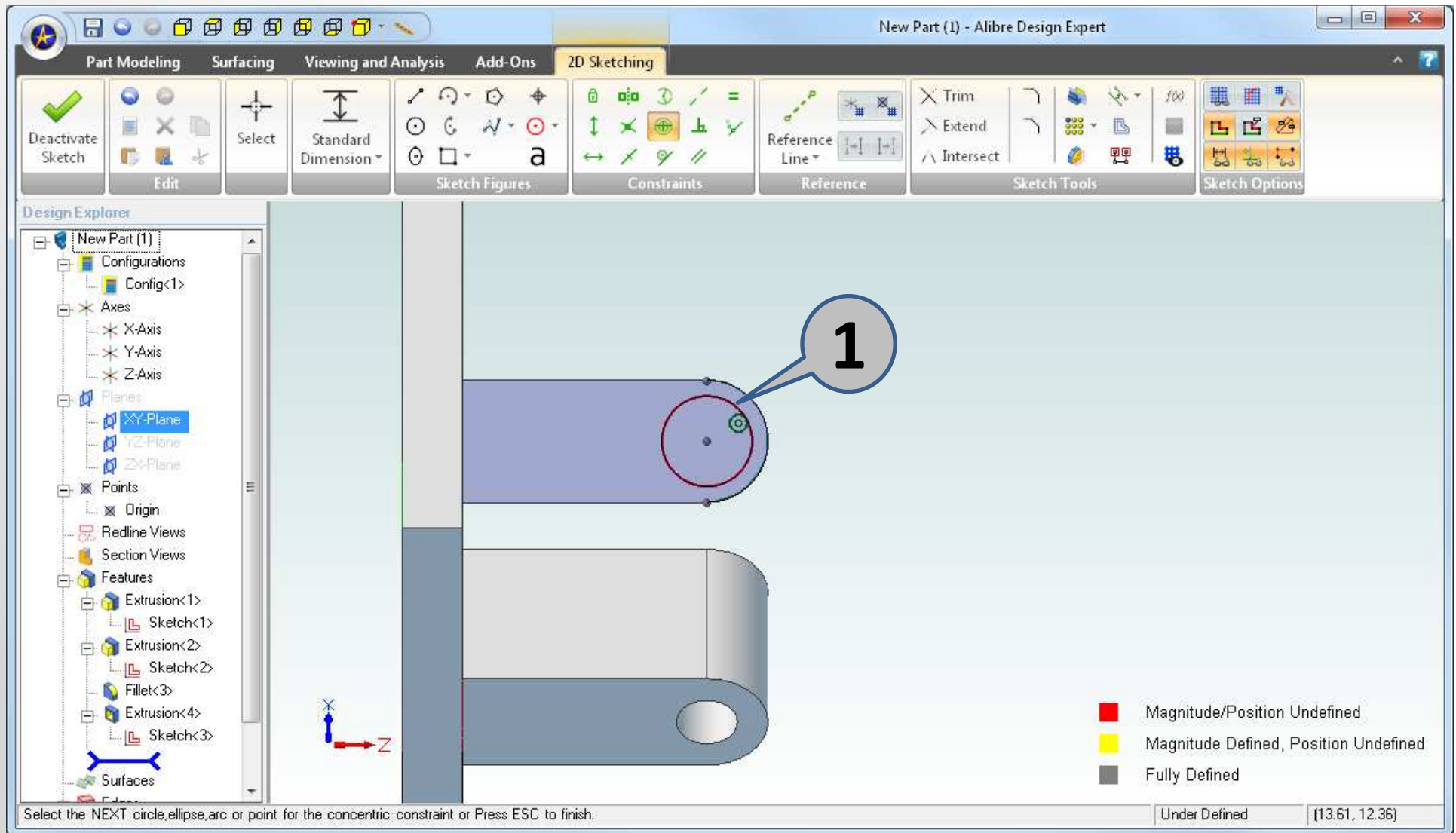
1. Position the mouse cursor over the face indicated; and roll the mouse wheel forward until the visible geometry is roughly twice as large.



1. Click on the **Circle** tool.
2. Position the mouse cursor as shown (approximately). Click and release to create the center.
3. Move the mouse cursor to the position indicated (approximately). Click and release to complete the circle.

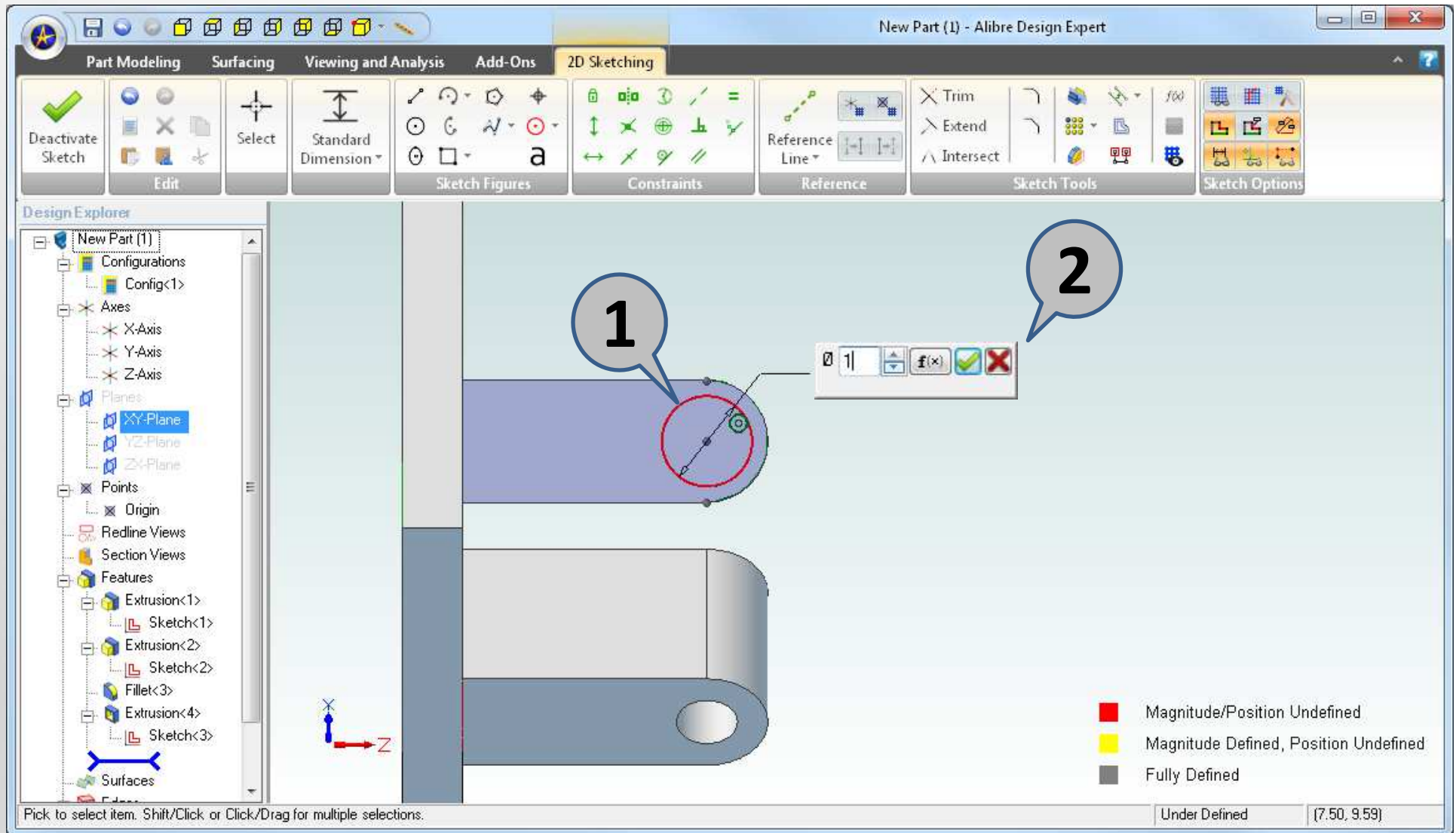


1. Select the **Concentric** sketch constraint.
2. Click on the perimeter of the circle.
3. Click on the circular edge shown in the screen capture.

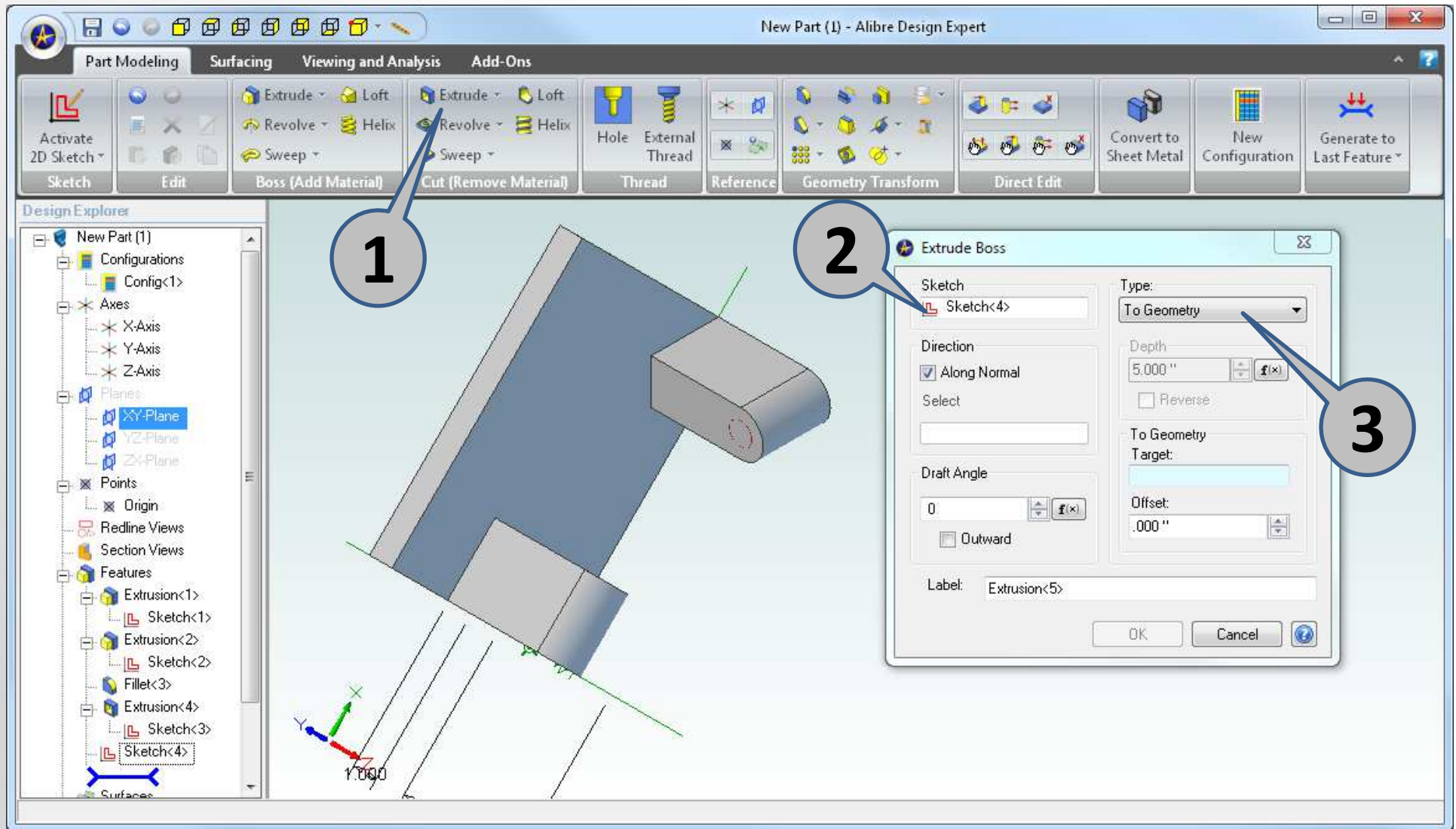


1. Confirm similar results for the **Concentric** constraint. Take note of the Concentric constraint symbol.
2. Enter into dimension mode.

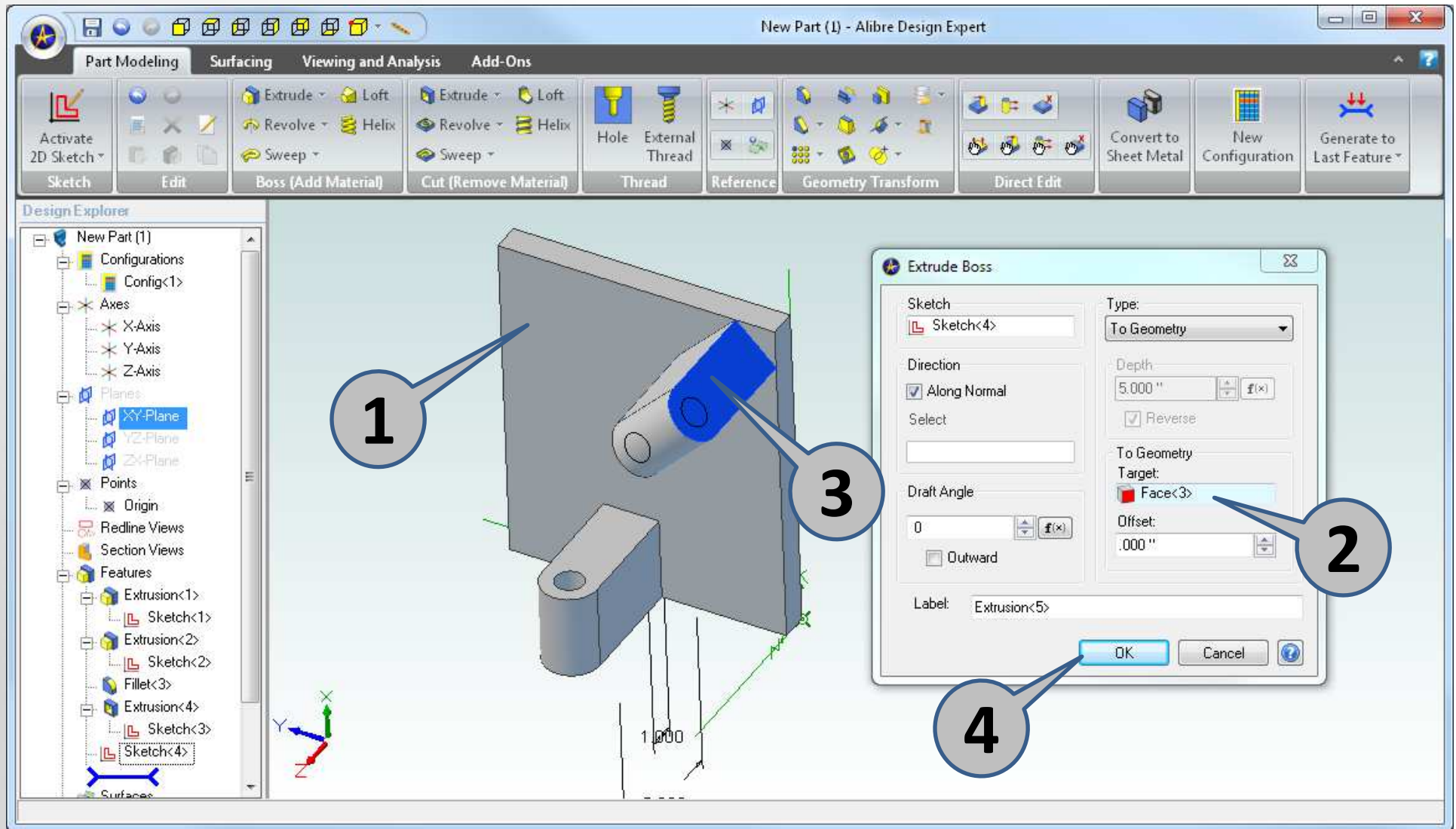
You can do this either by clicking on the **Standard Dimension** icon, or you can right click in the workspace and select **Dimension** from the popup menu.



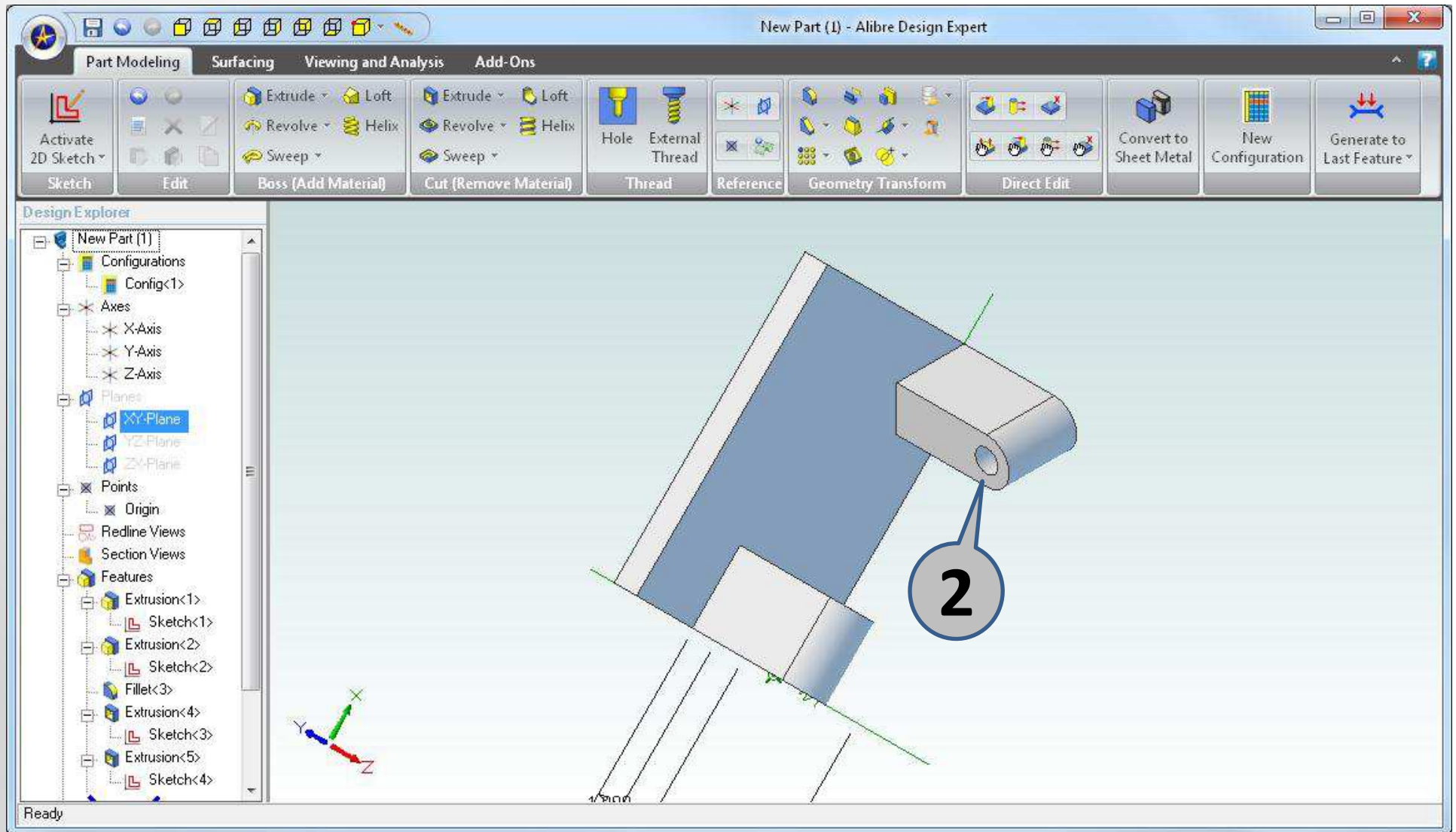
1. Click on the perimeter of the circle.
2. Position the dimension approximately as shown. Click and release to place the dimension.
3. Enter a value of **1** for the dimension. Click on the green checkmark or press the **Enter** key on the keyboard (not shown).



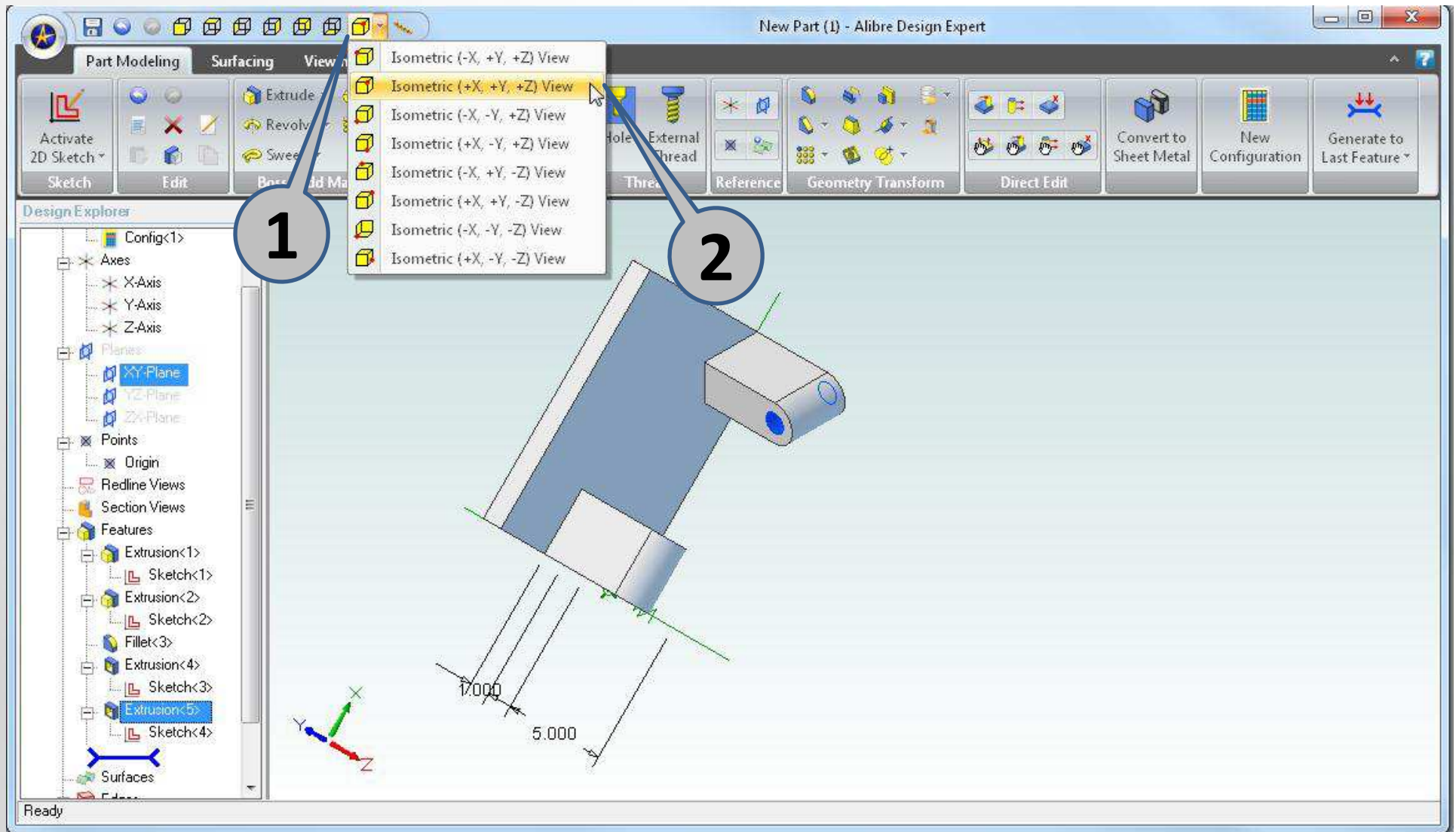
1. Click on the **Part Modeling** tab in the ribbon and click on the **Extrude Cut** icon.
2. Confirm that the **Sketch<4>** is the Sketch to be extruded.
3. Select **To Geometry** as the **Type** of extrusion.



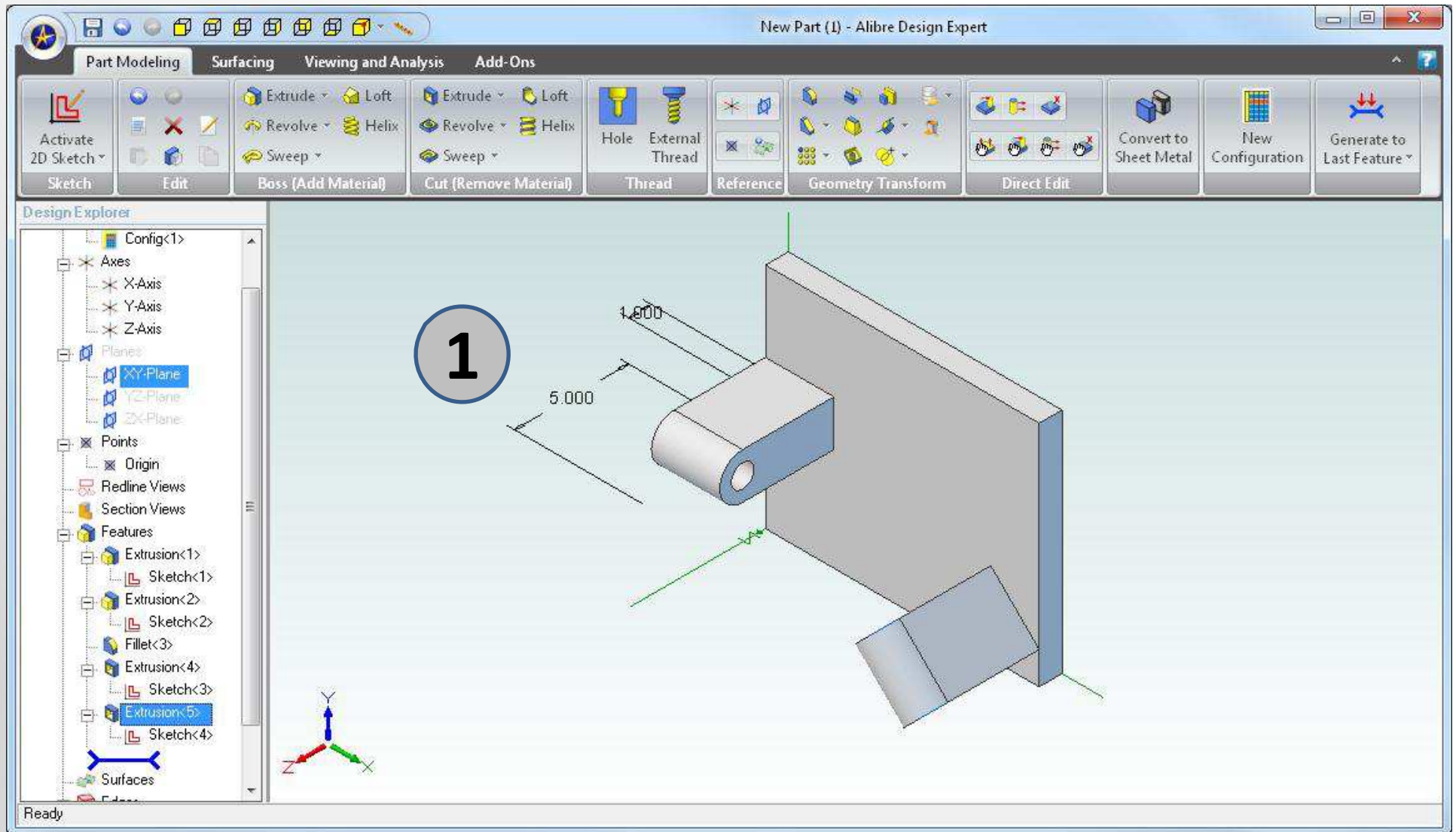
1. With the Extrude Cut dialog box still open, rotate the model (press and hold both mouse buttons while moving the mouse cursor) until it is in the approximate orientation.
2. Click in the Target textbox (it will highlight light blue).
3. Click on the face indicated above.
4. Click on the OK button.



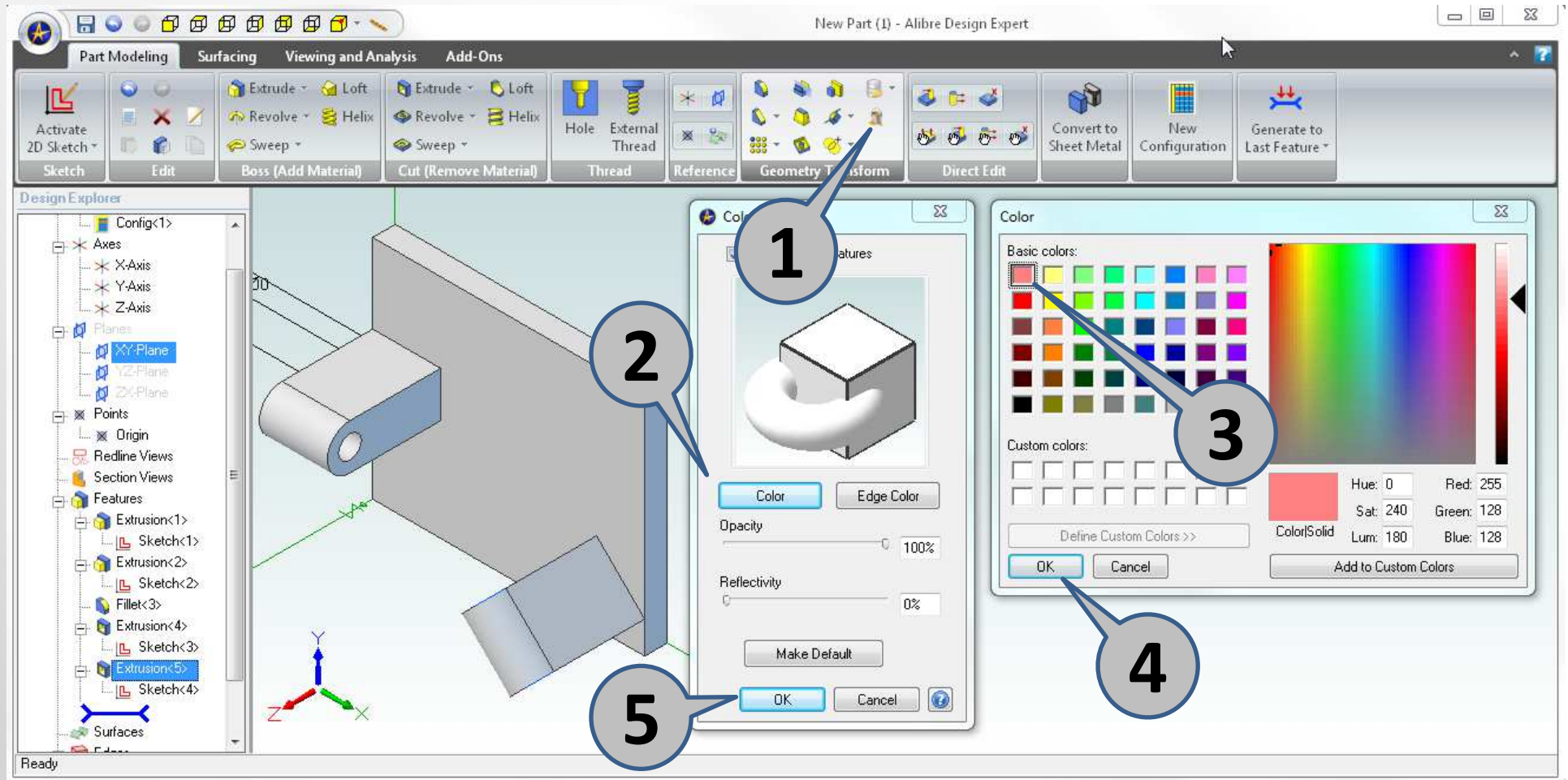
1. Rotate the model so it is in the approximate orientation as shown above.
2. Confirm similar results for the **Extrude Cut** operation.



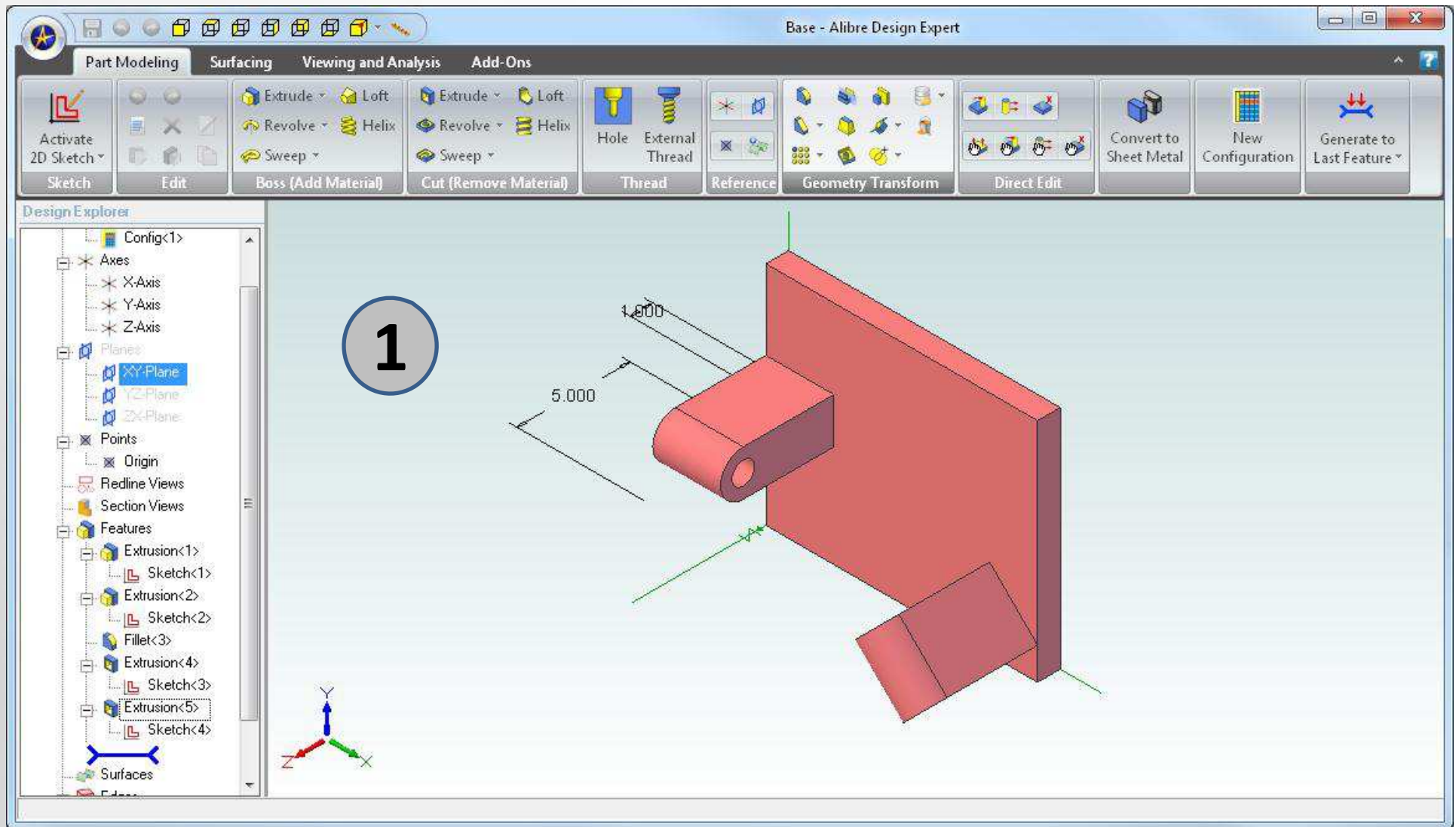
1. From the Quick Access toolbar
2. Select the second **Isometric** option.



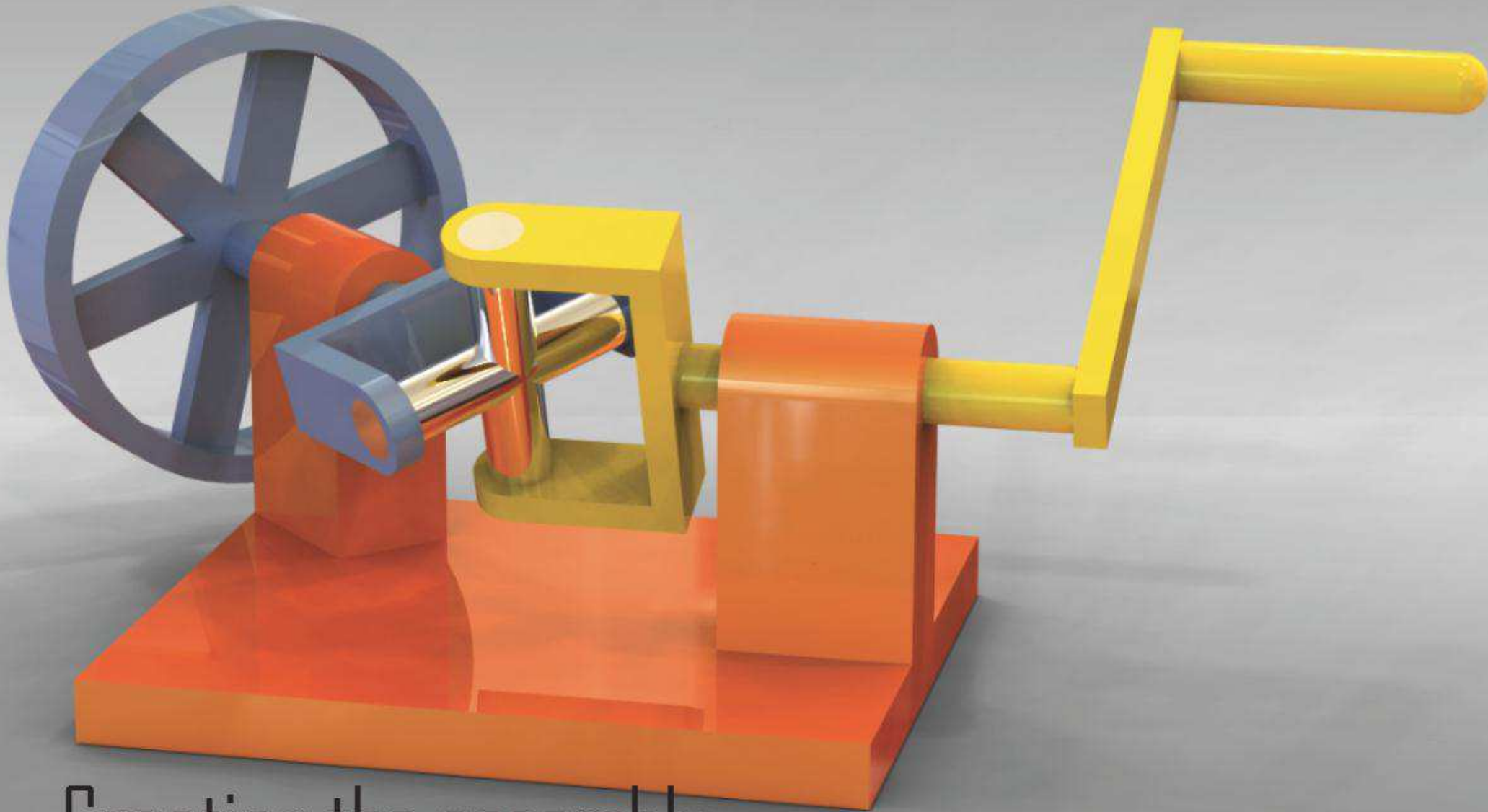
1. Confirm similar results for the view re-orientation.



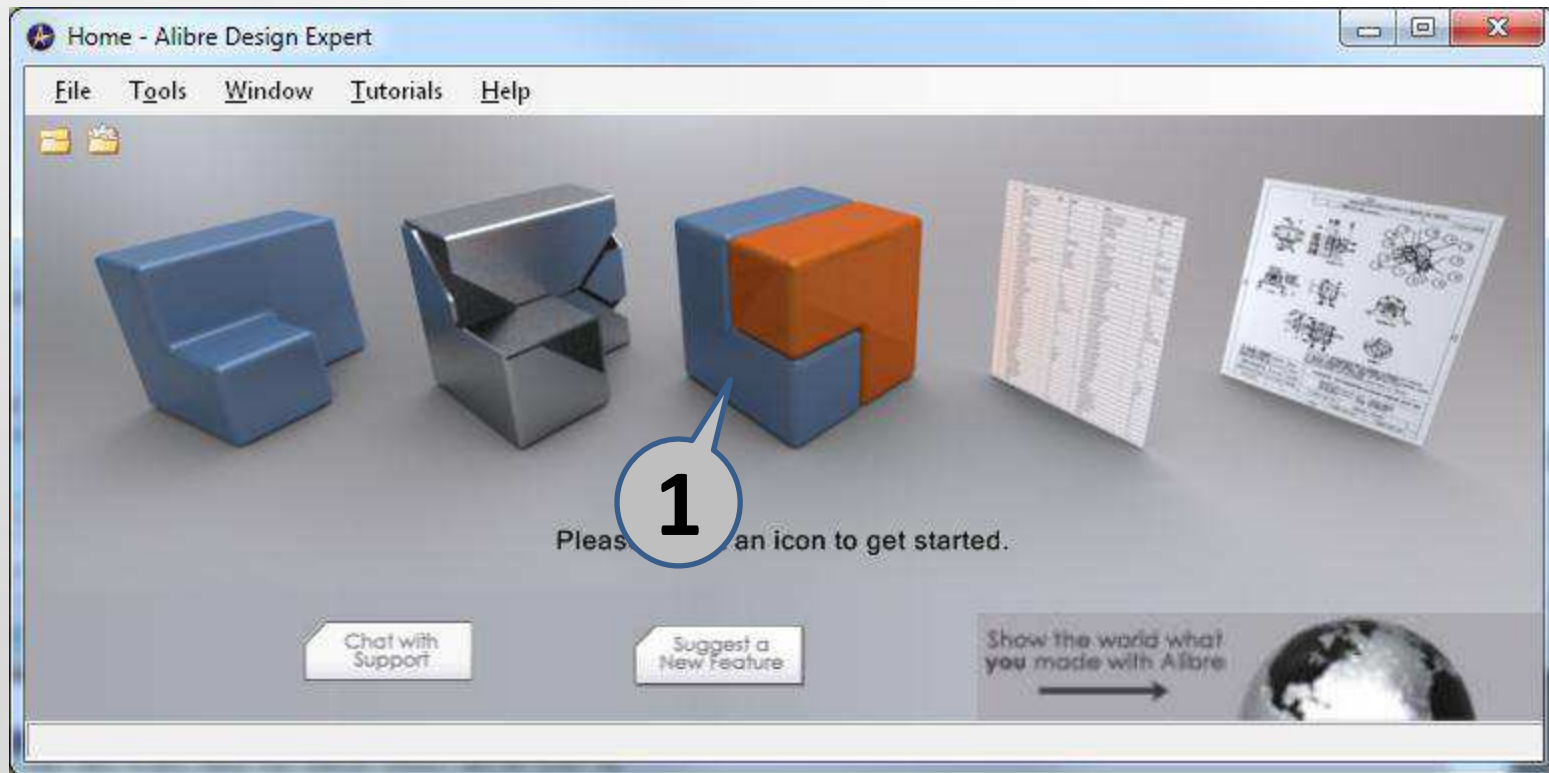
1. Select the **Part Color** icon.
2. Click on the **Color** button in the **Color Properties** dialog box.
3. Select a light red color in the **Color** dialog box.
4. Click **OK** in the **Color** dialog box.
5. Click **OK** in the **Color Properties** dialog box.



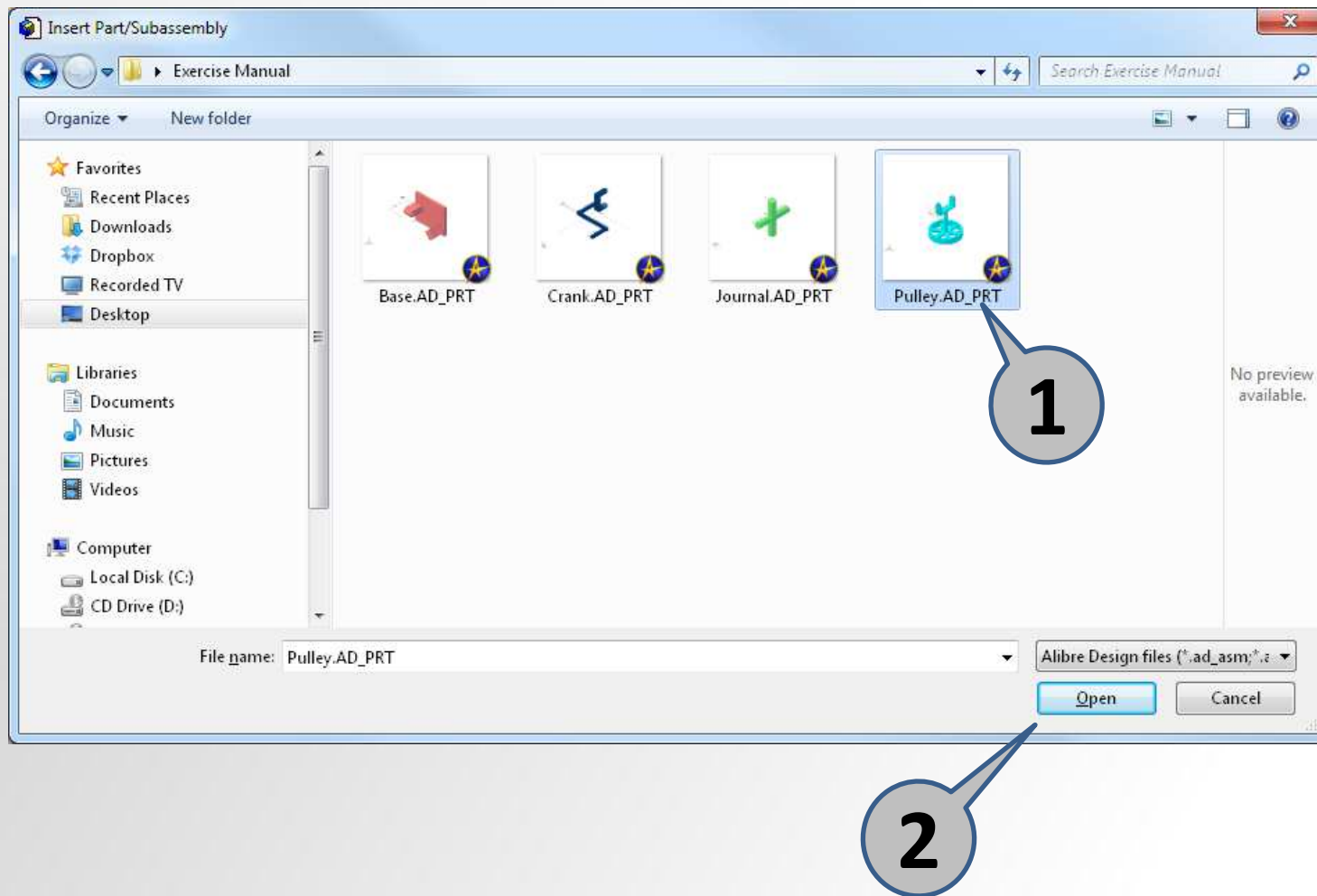
1. Confirm similar results for the color change.
2. **Save** the **Part** to the same location as the **Journal**, **Crank** and **Pulley**, with the name **Base** (not shown).
3. **Close** the **Part** workspace (not shown).



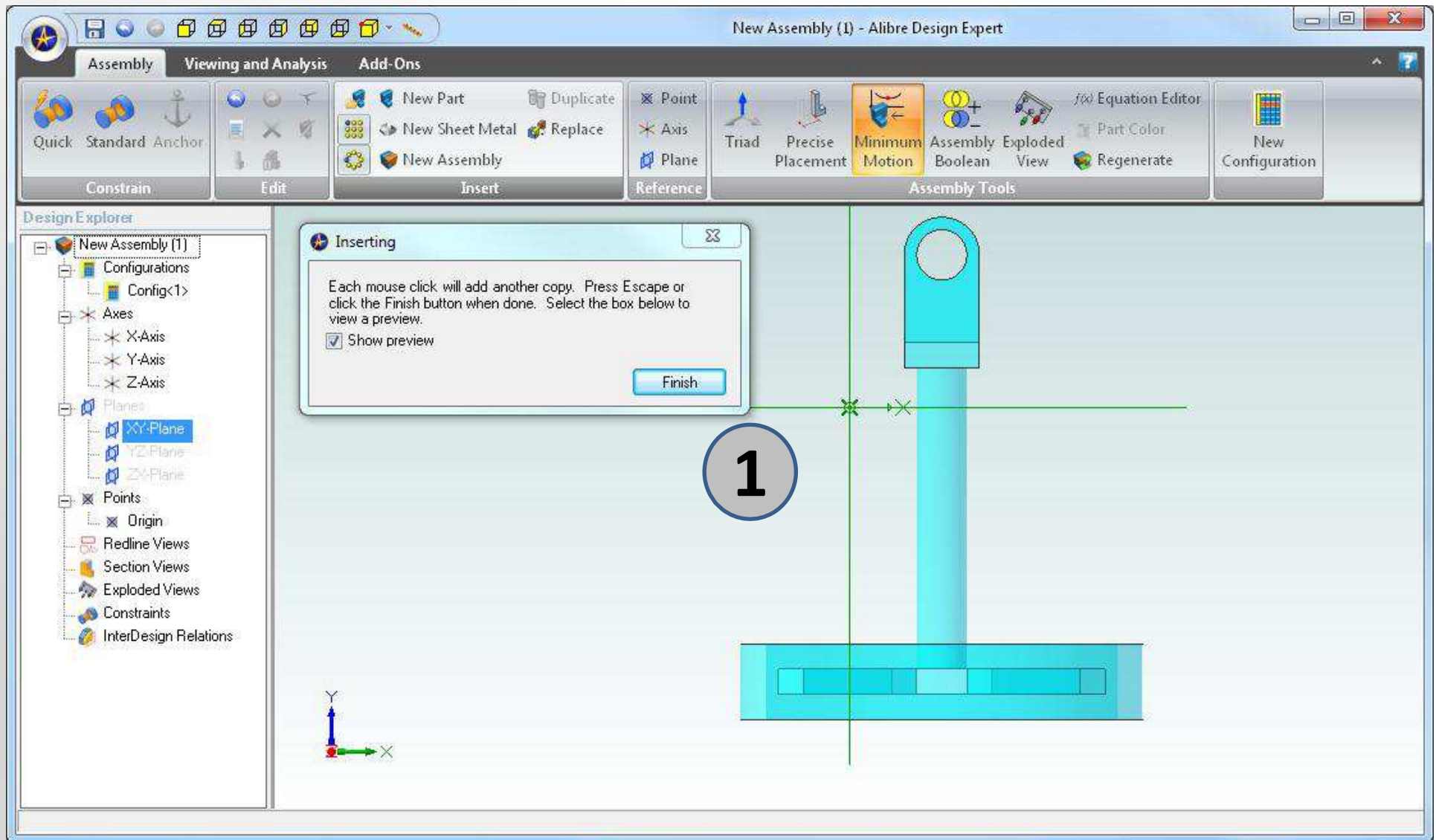
Creating the assembly



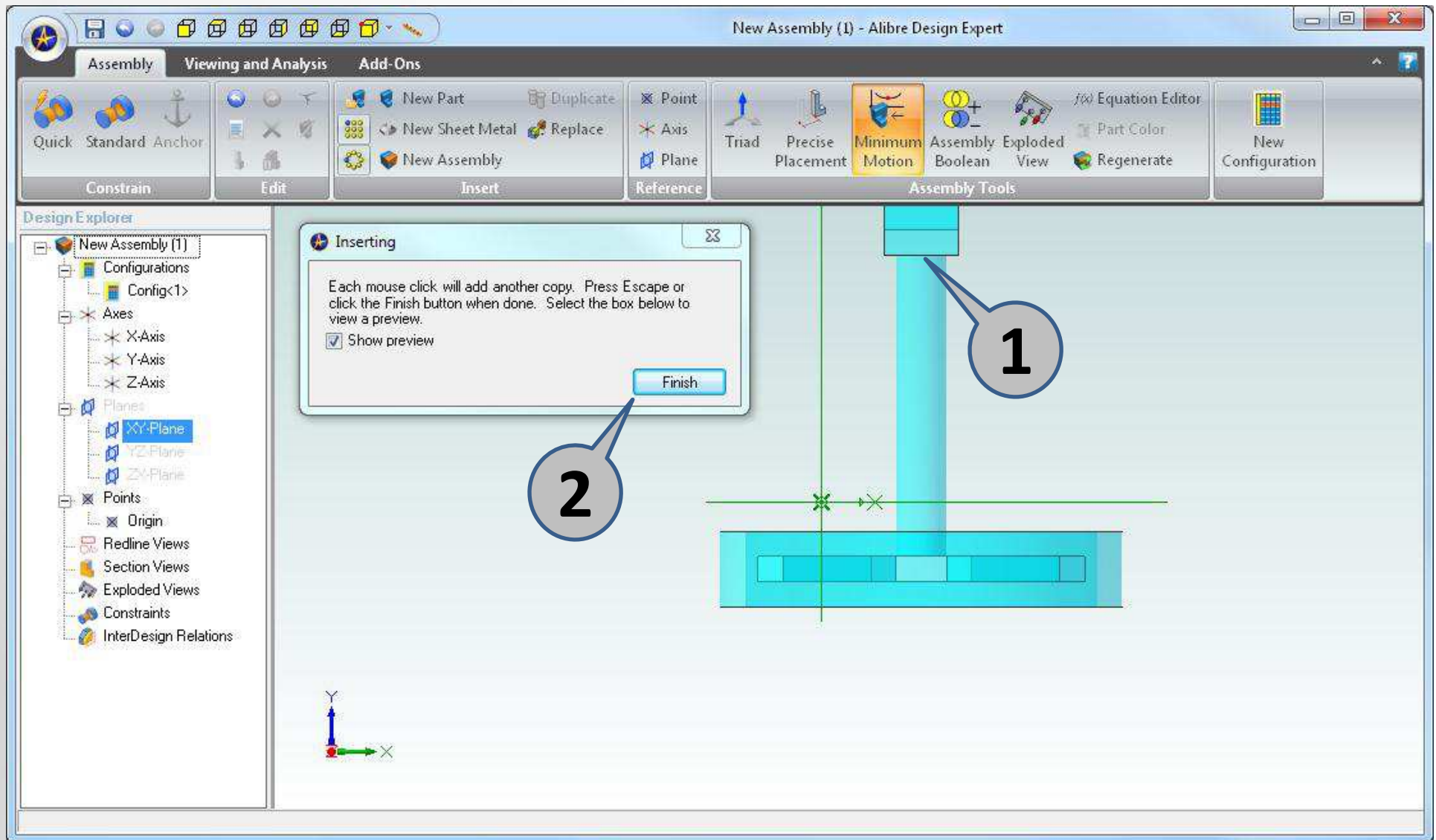
1. Open a new **Assembly** workspace from Alibre Design's **Home** window.



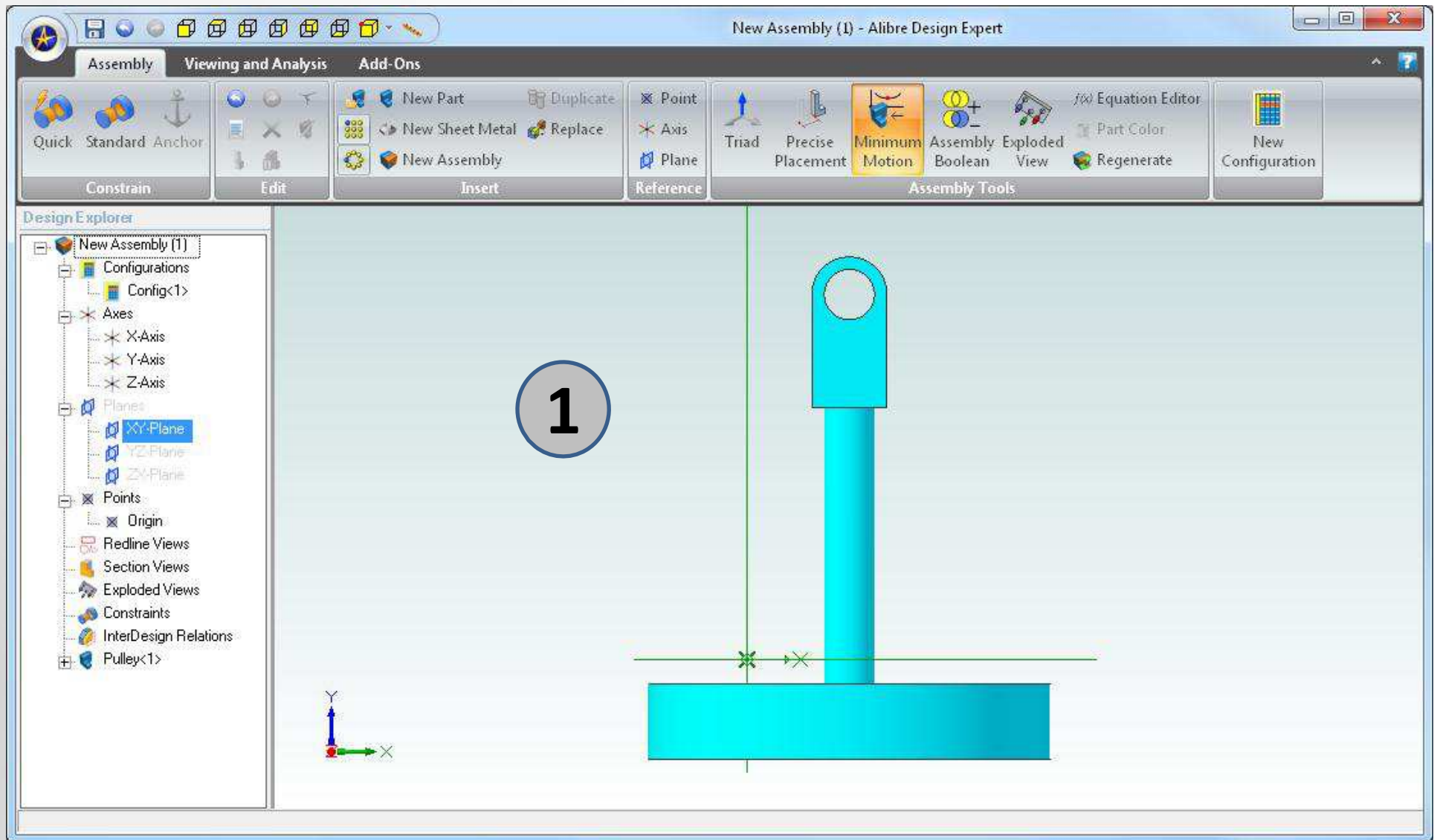
1. Open the **Insert** menu.
2. Click on the **Part/Subassembly...** option.
3. Select the **Pulley** in the **Insert Part/Subassembly** dialog box.
4. Click **OK**.



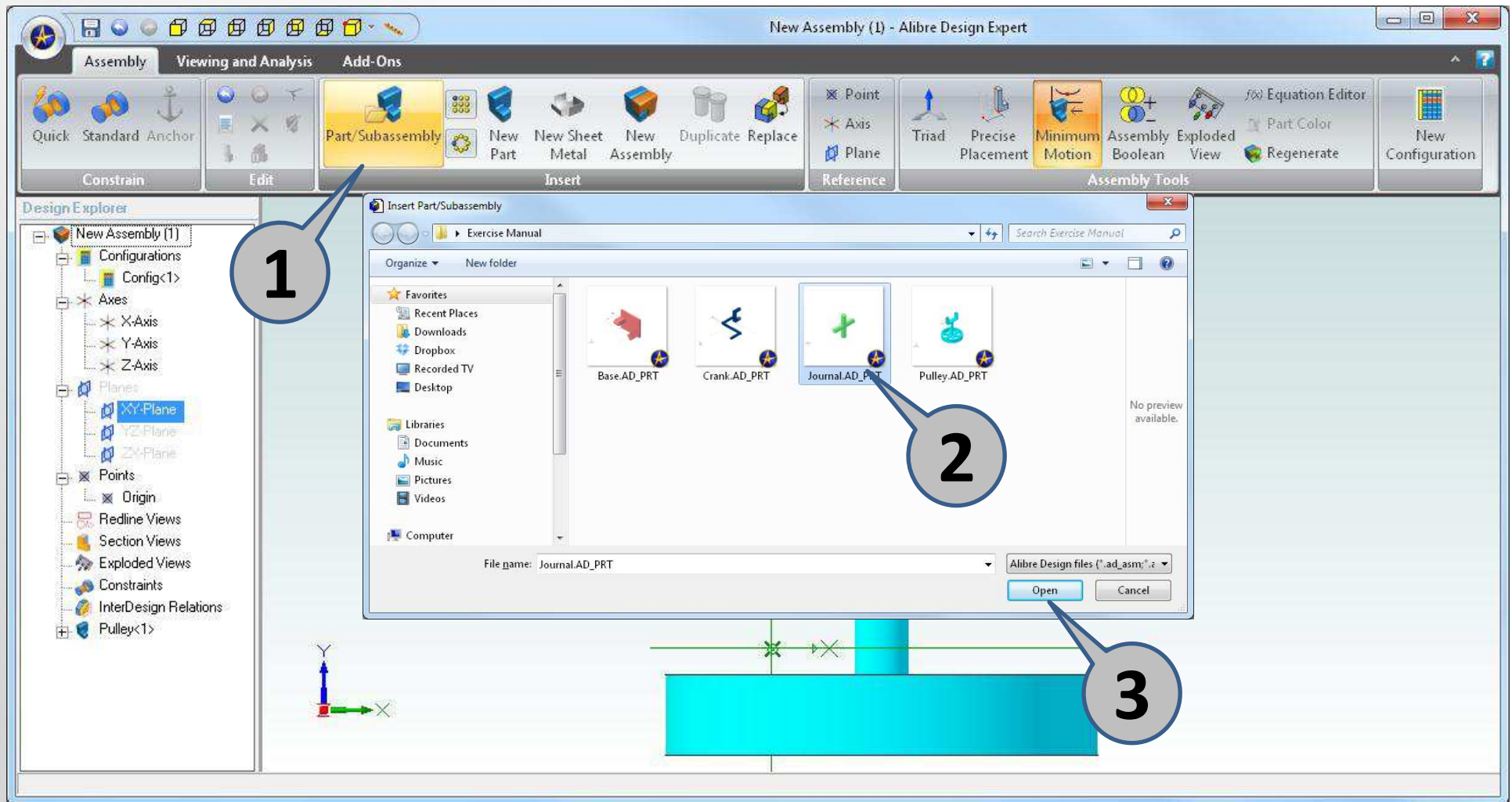
1. Do not click any mouse buttons yet, but move the mouse around. Take note of the transparent copy of the **Pulley** attached to the cursor.



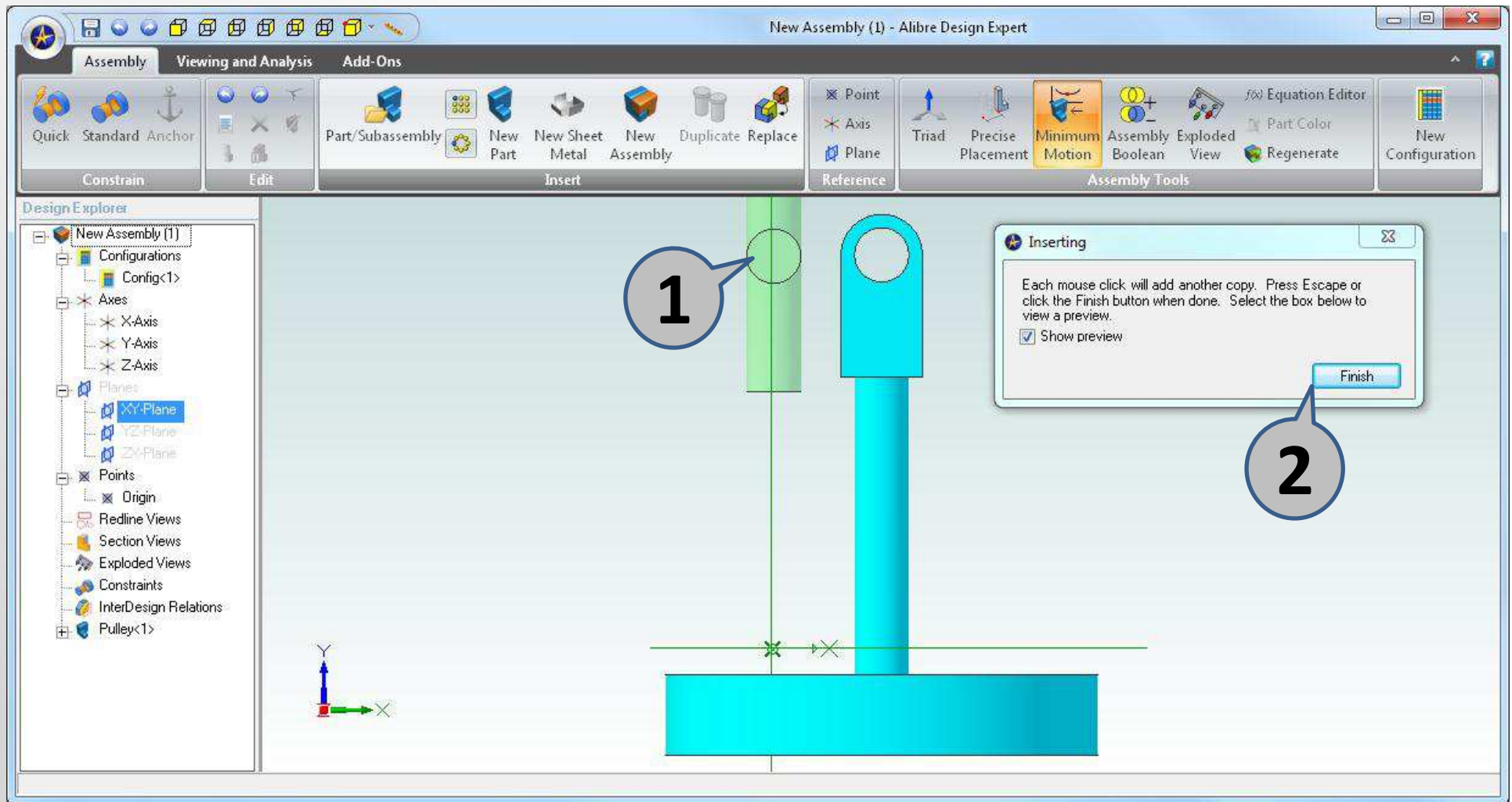
1. Position the mouse cursor as shown. Every mouse click in the workspace will result in an additional part being inserted into the assembly. Click and release once to create a single instance of the **Pulley** at the location indicated.
2. Click the **Finish** button in the **Inserting** dialog box.
3. Press the **Home** key on the keyboard (not shown).



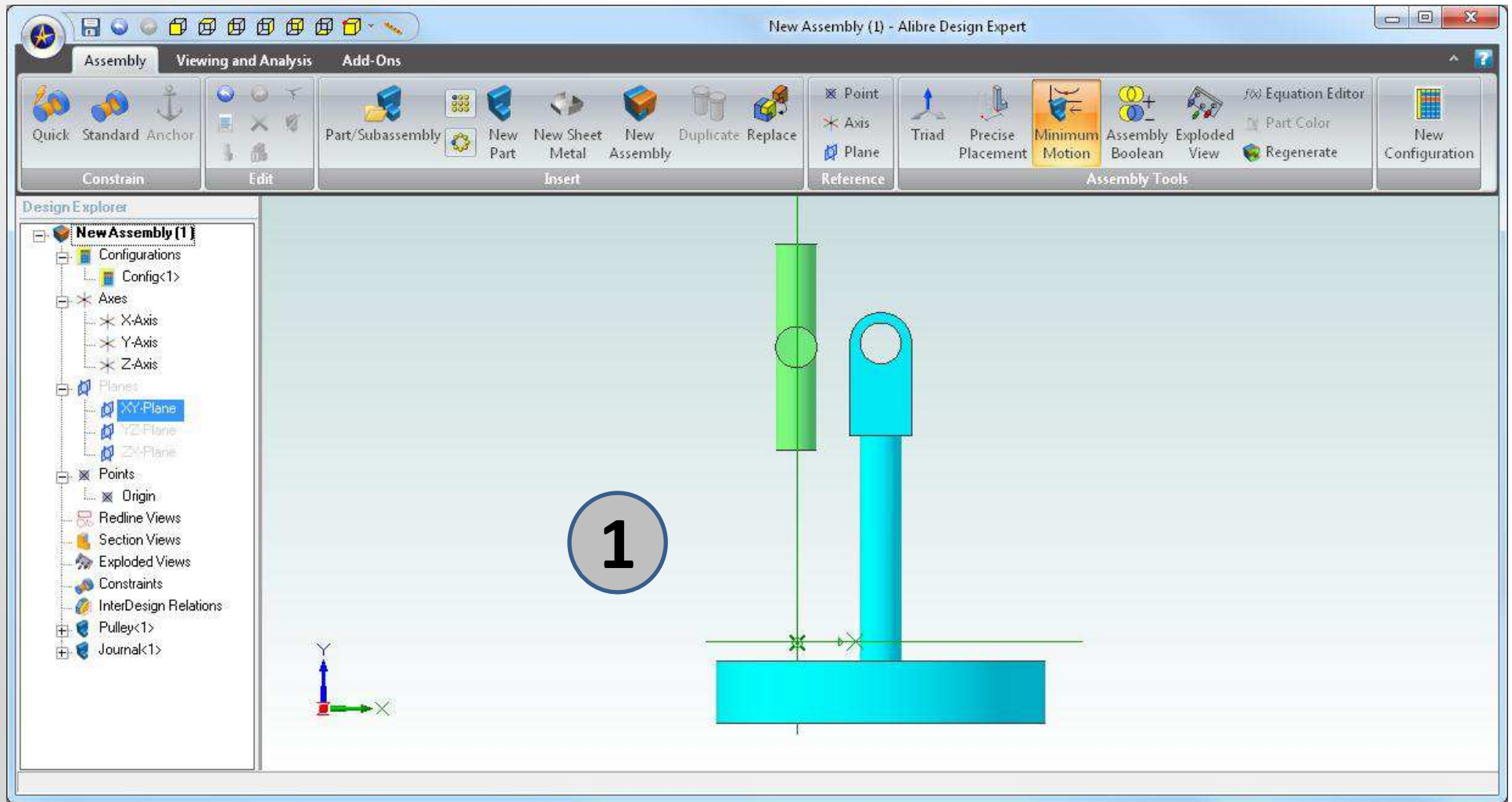
1. Confirm that the view of the workspace is similar to the screen capture.



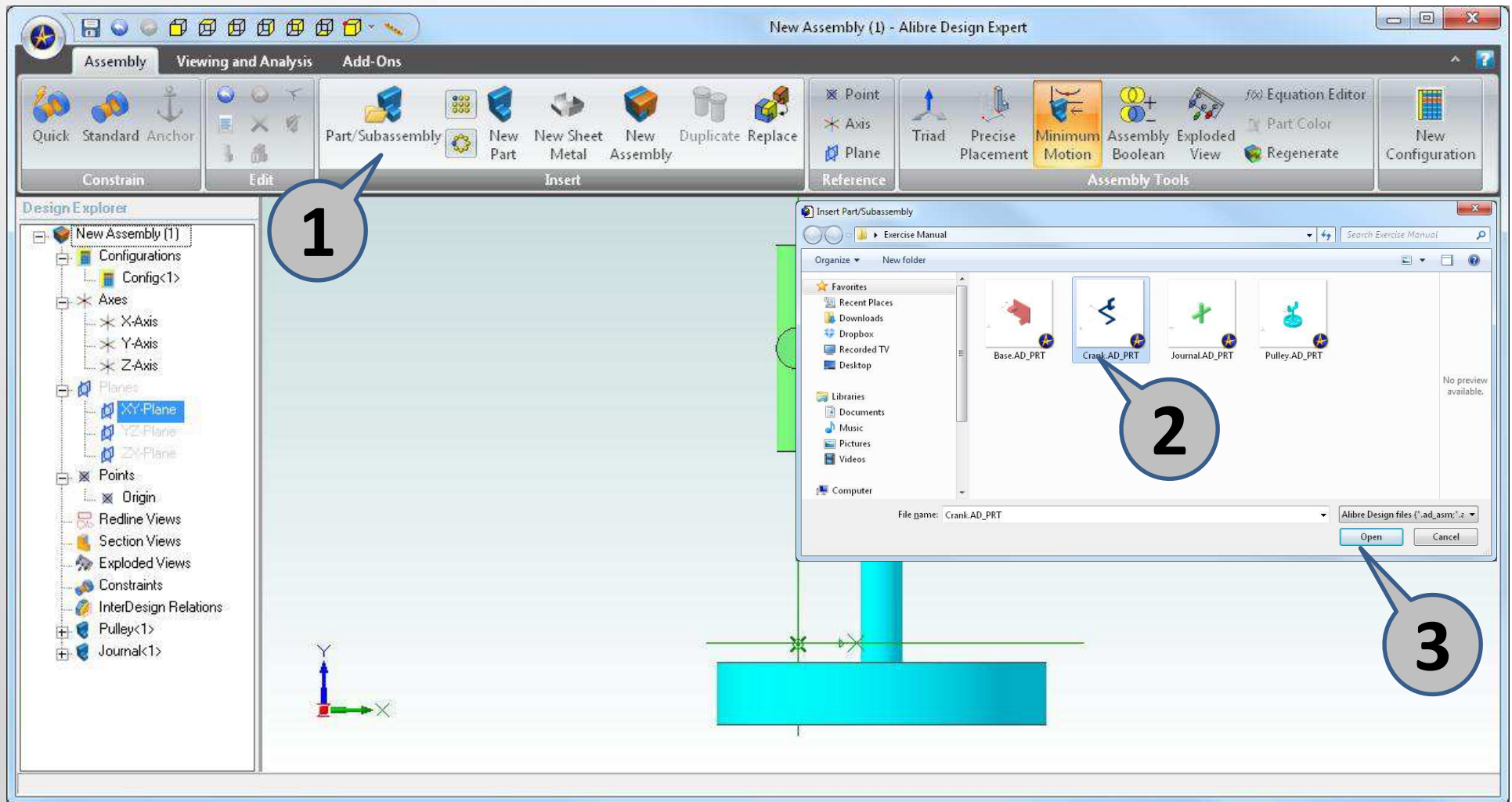
1. From the **Insert** group, select the **Part/Subassembly** icon.
2. Select the **Journal** in the **Insert Part/Subassembly** dialog box.
3. Click **Open**.



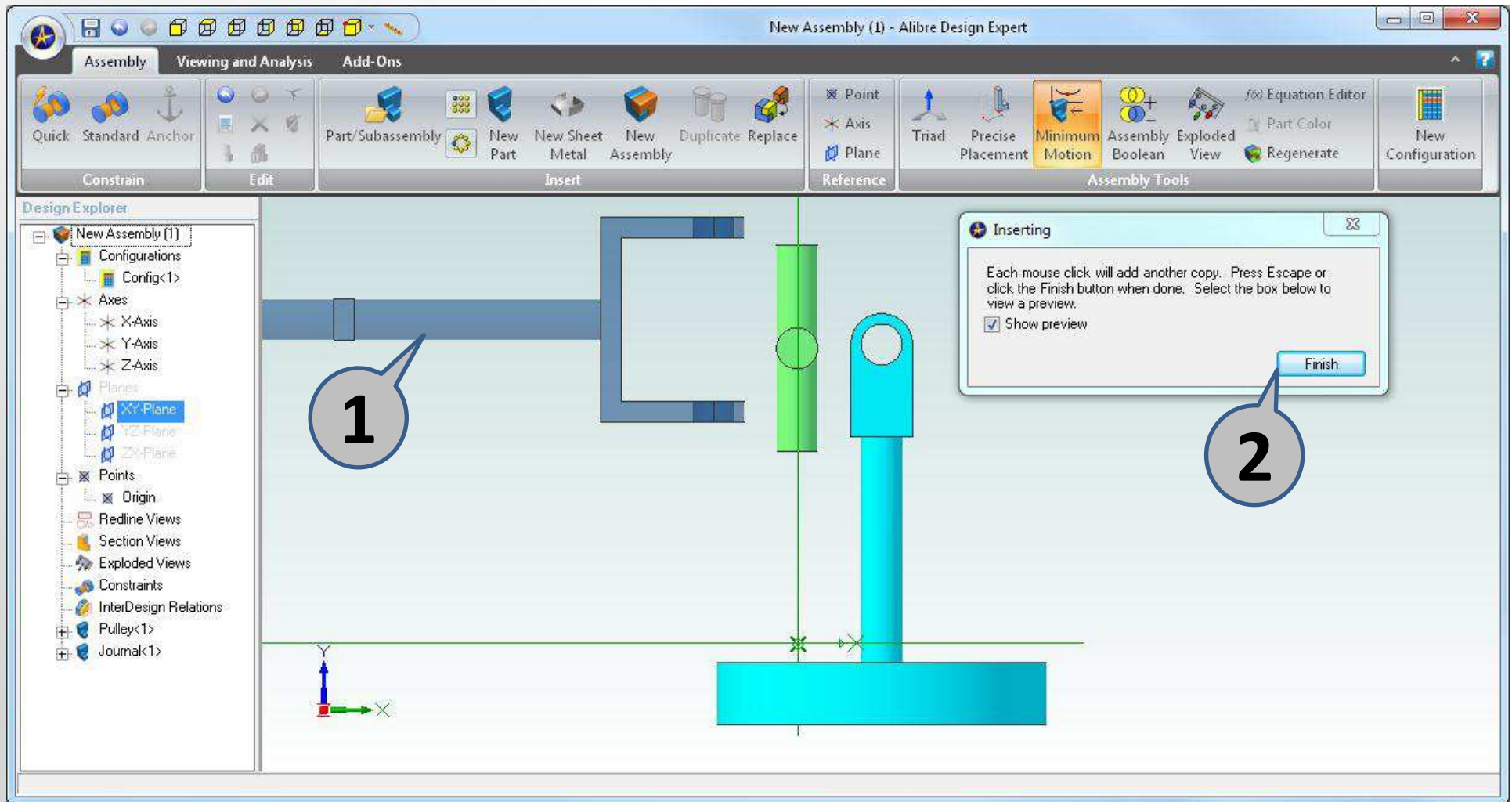
1. Position the mouse cursor as shown. Click and release once to create a single instance of the **Journal** at the location indicated.
2. Click the **Finish** button in the **Inserting** dialog box.
3. Press the **Home** key on the keyboard (not shown).



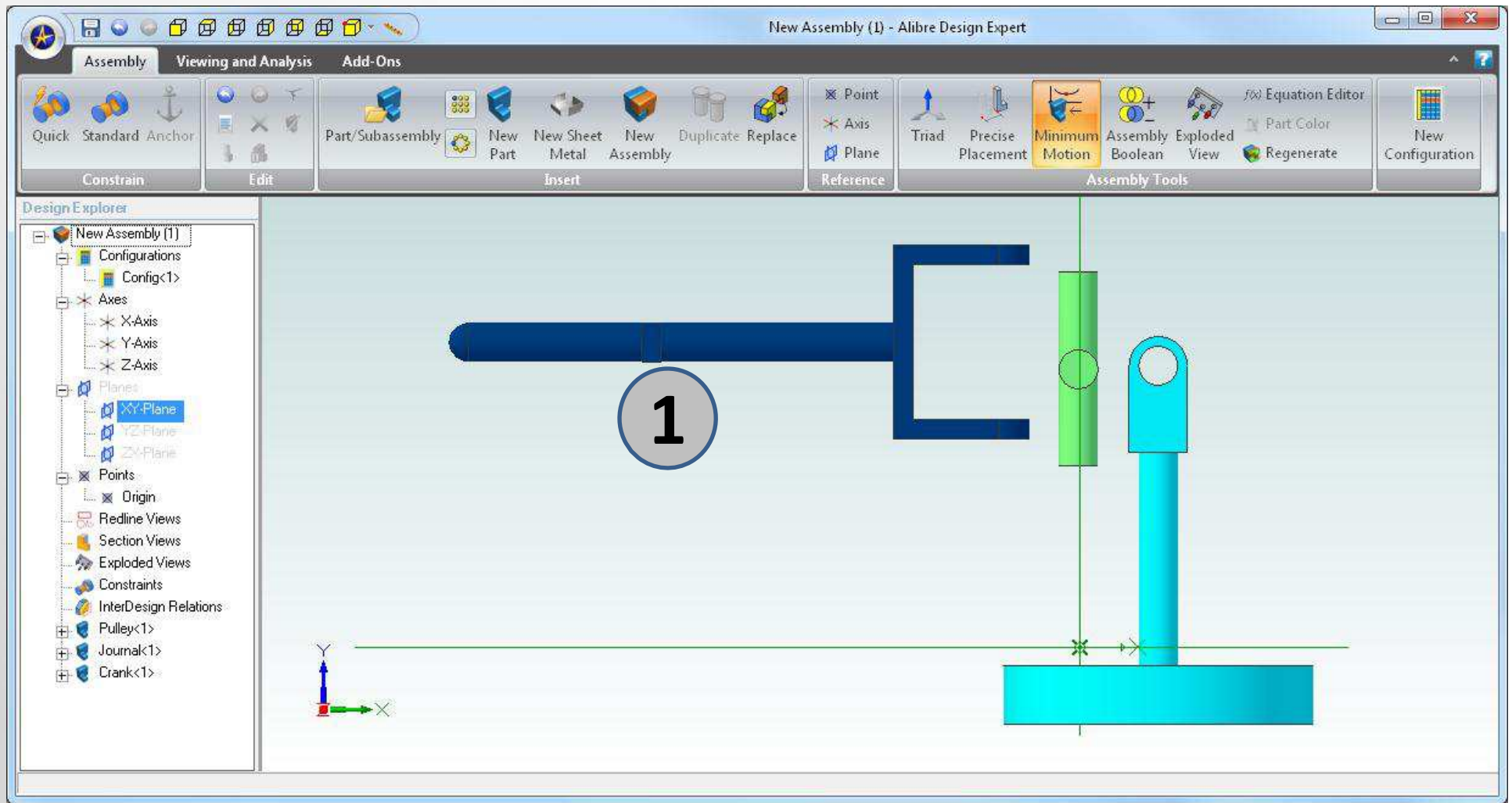
1. The view of the work area should be similar to the screen capture.



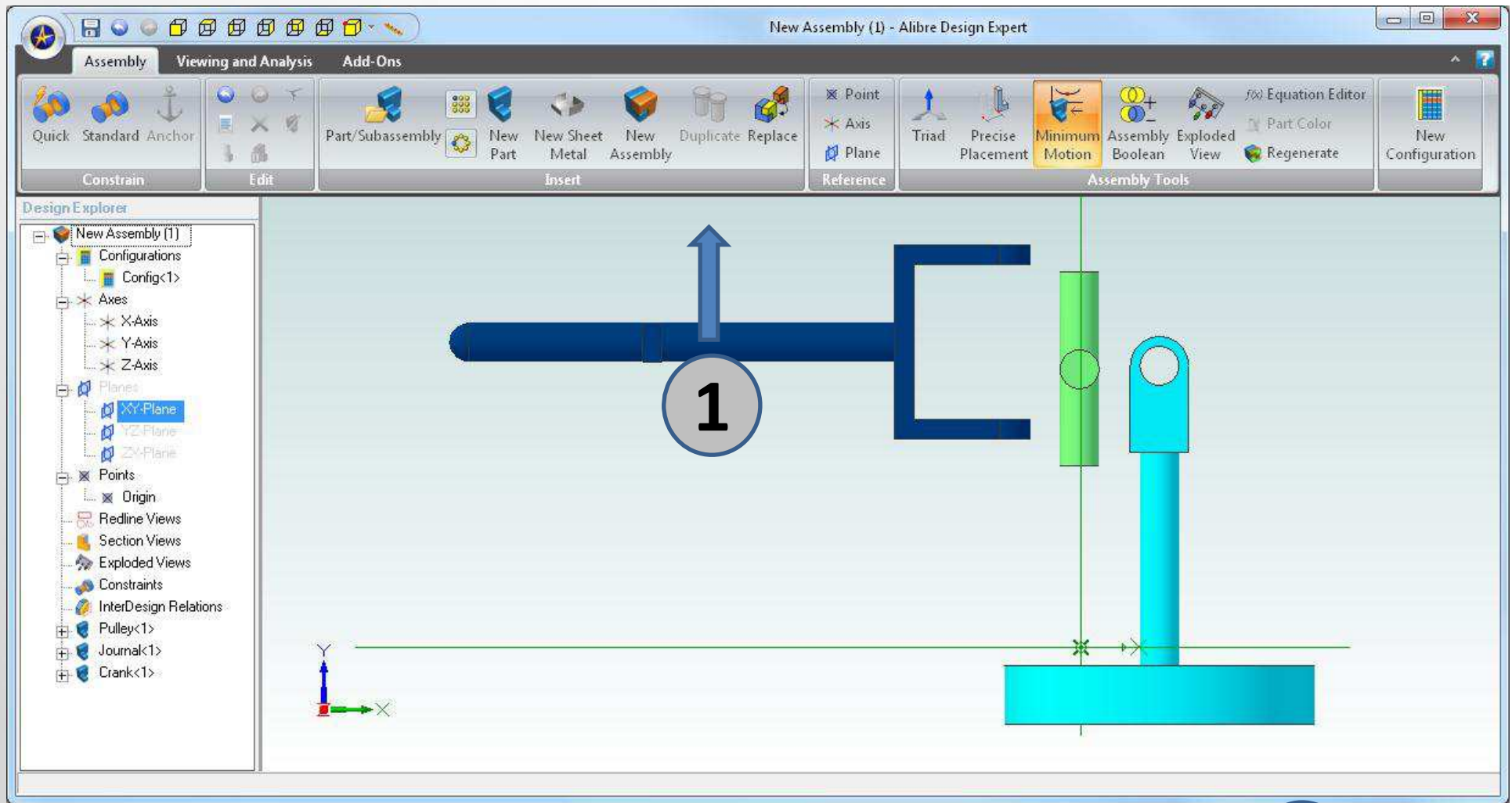
1. From the **Insert** group, select the **Part/Subassembly** icon.
2. Select the **Crank** in the **Insert Part/Subassembly** dialog box.
3. Click **Open**.



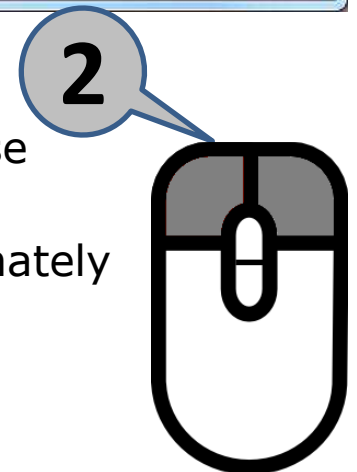
1. Position the mouse cursor as shown. Click and release once to create a single instance of the **Crank** at the location indicated.
2. Click the **Finish** button in the Inserting dialog box.
3. Press the **Home** key on the keyboard (not shown).

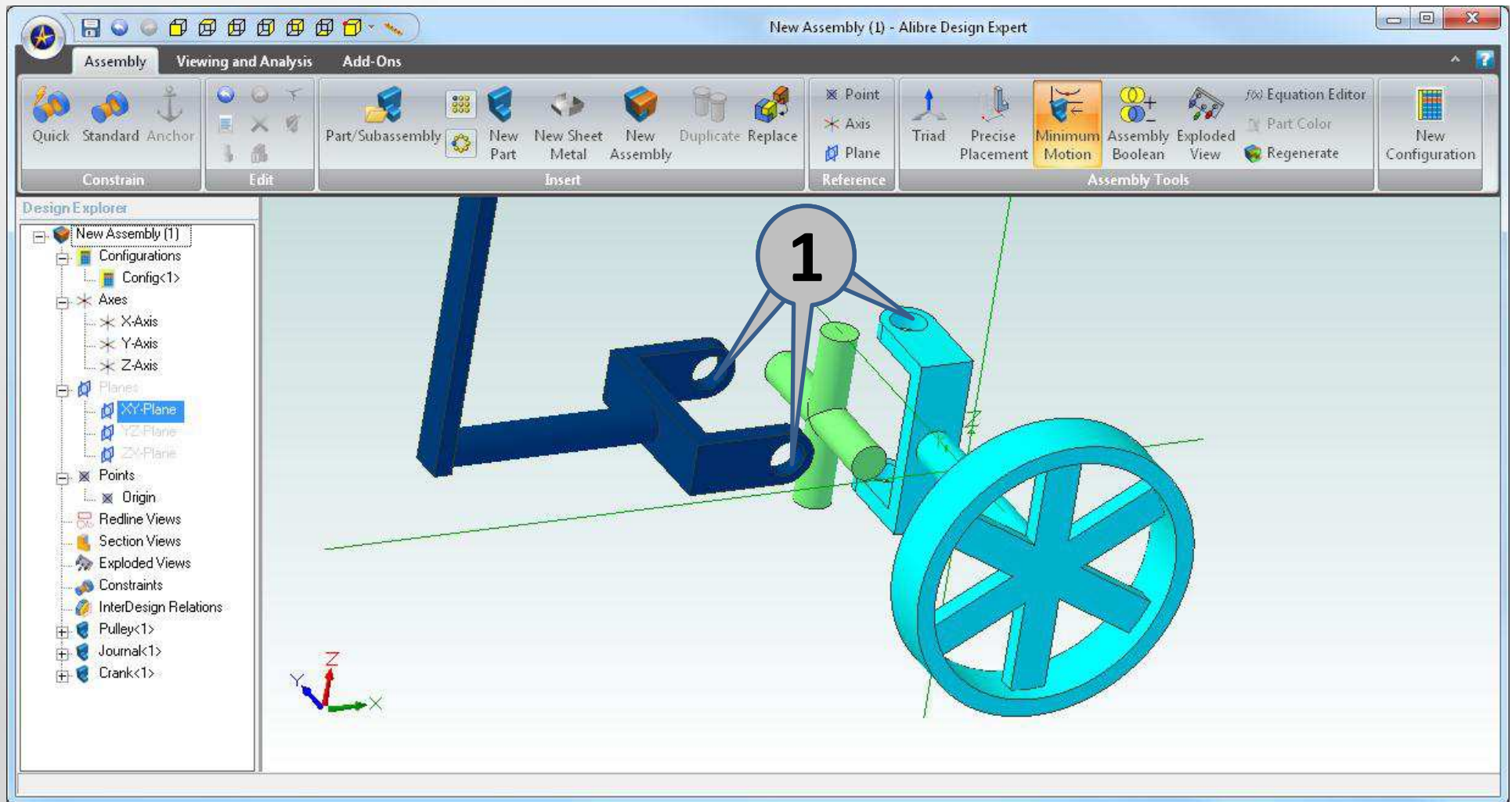


1. The view of the work area should be similar to the screen capture.

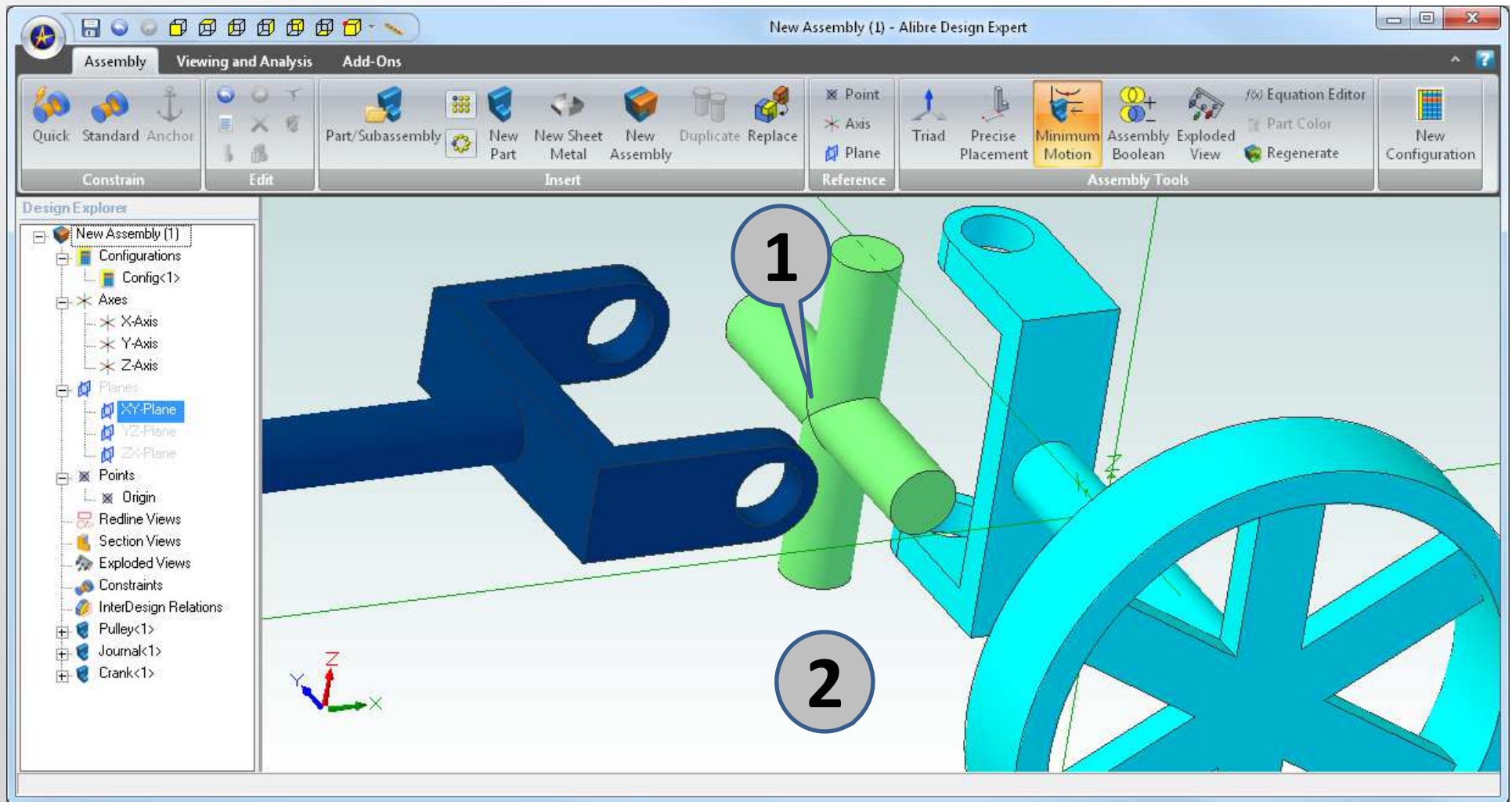


1. Position the mouse cursor as shown. Hold down the left and right mouse buttons together over the model.
2. Click and hold the mouse button while moving the mouse up (approximately as shown). Release the mouse button.

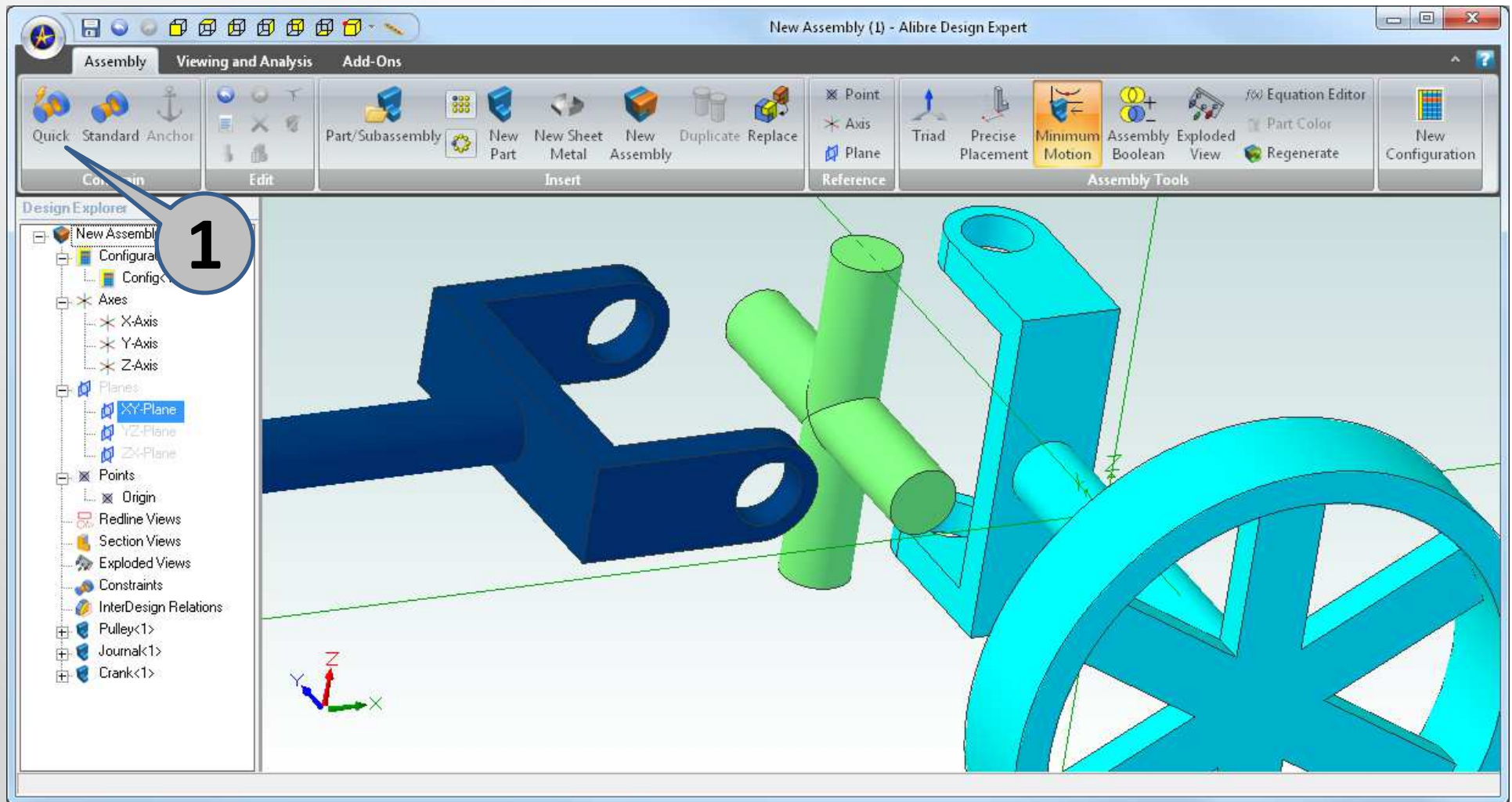




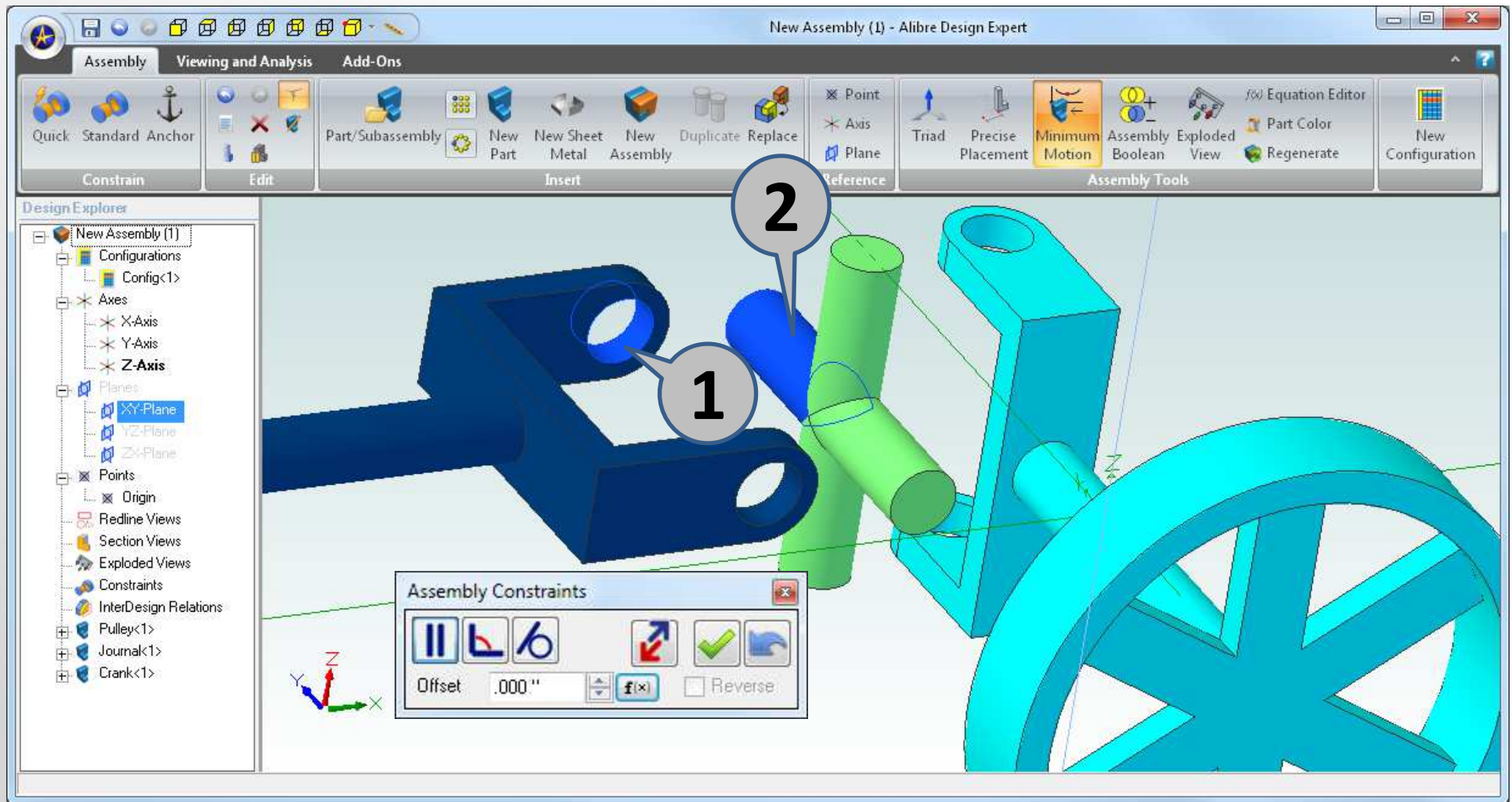
1. Confirm (or continue) the view-rotation such that the indicated faces are visible: the inner faces of both cylindrical holes, and the flat face of the crank (as shown).



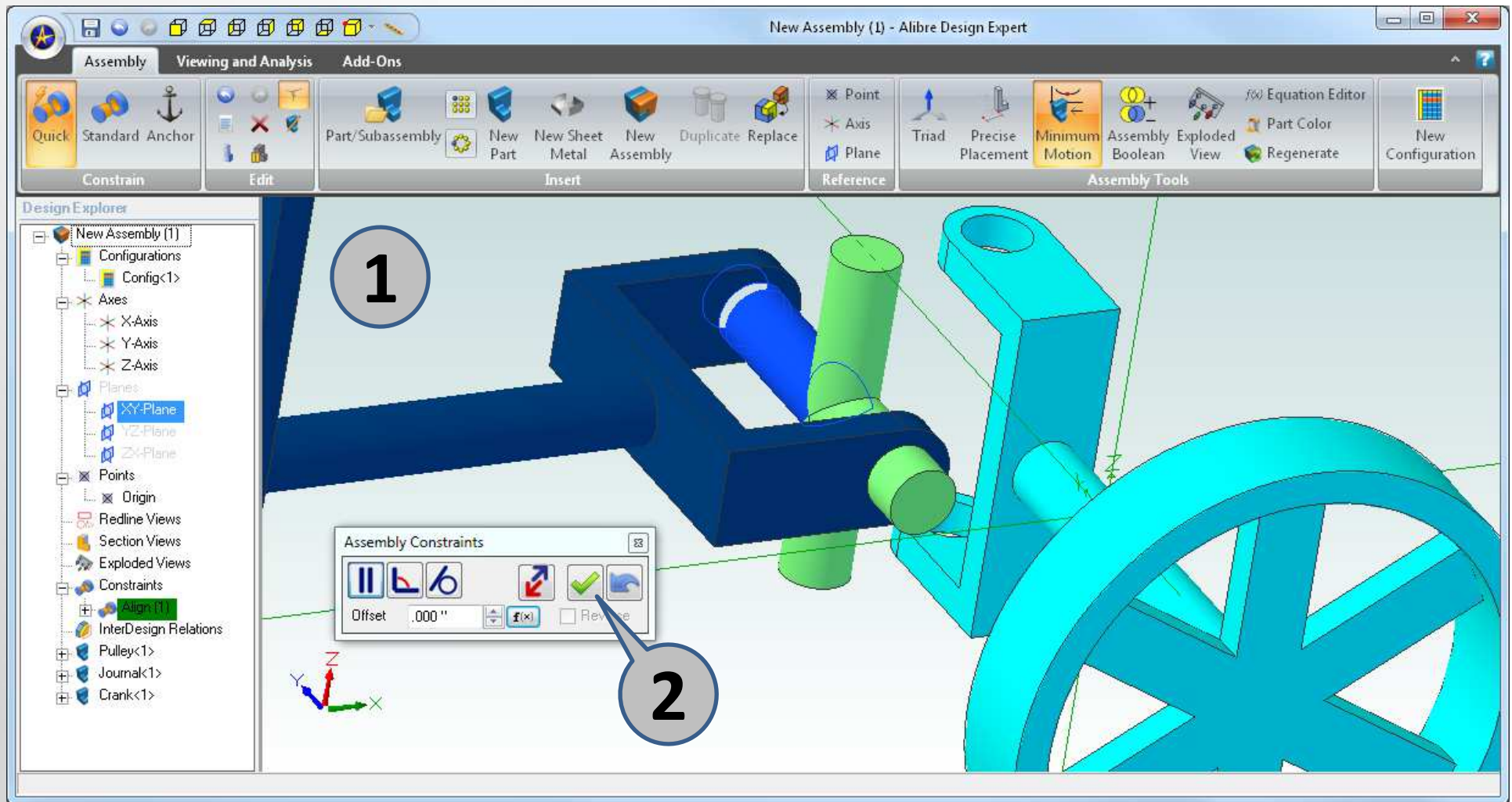
1. Position the mouse cursor over the center of the Journal part, and roll the mouse wheel forward to zoom in until the visible geometry roughly doubles in size.
2. The view of the work area should be similar to the screen capture.



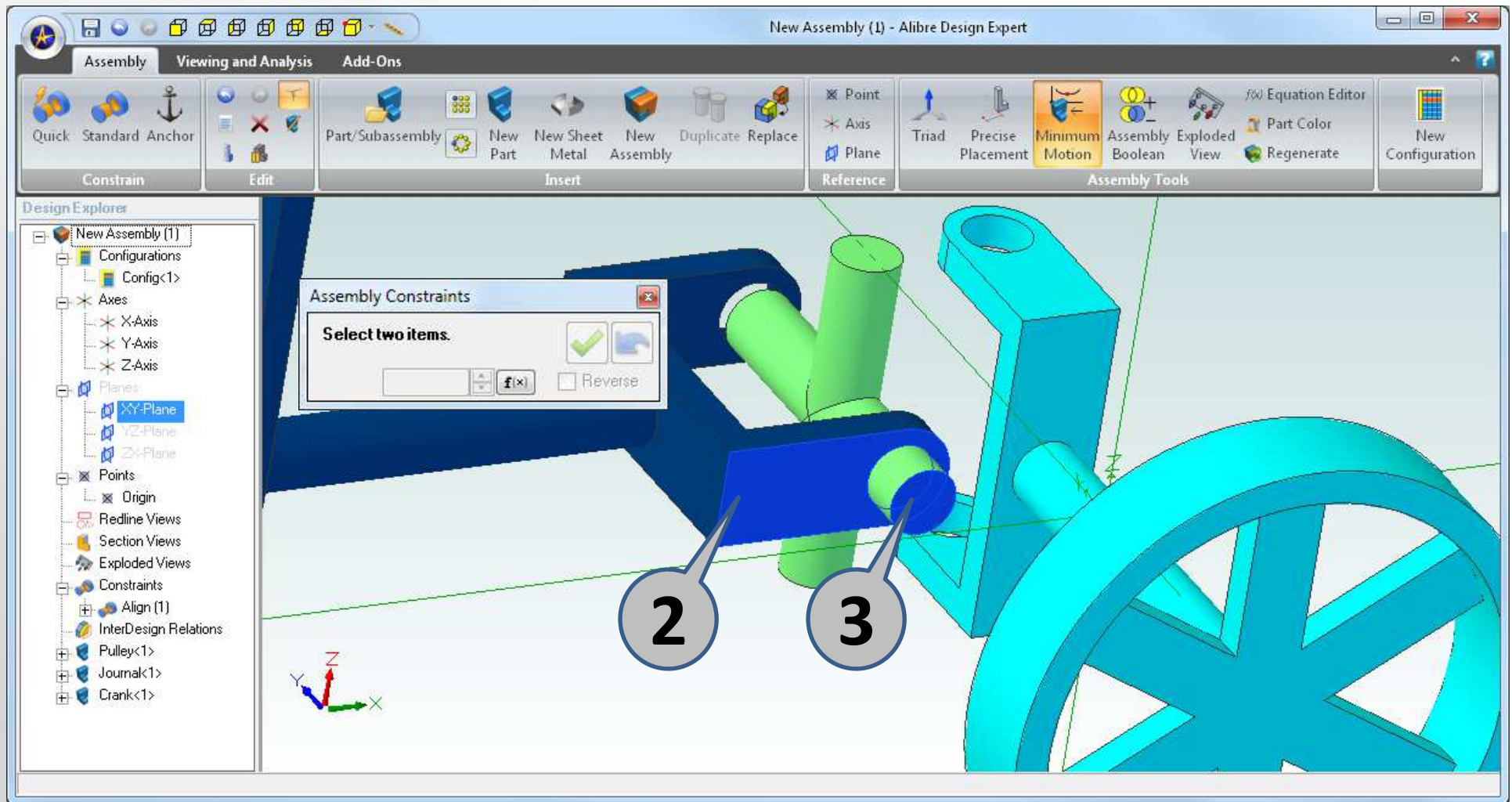
1. From the **Constrain** group, select the **Quick** command.



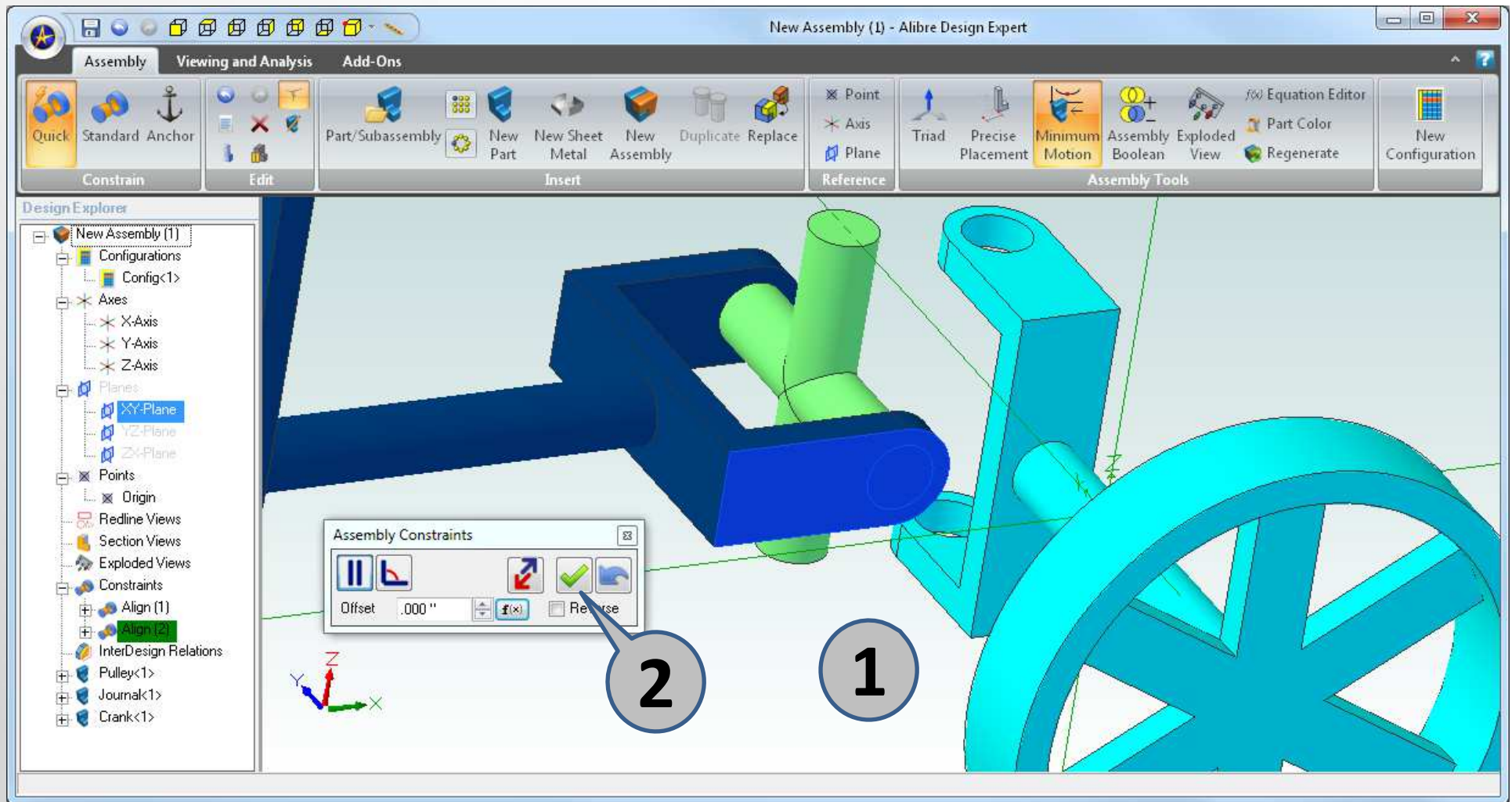
1. Click on the indicated face
2. Click on the indicated face.



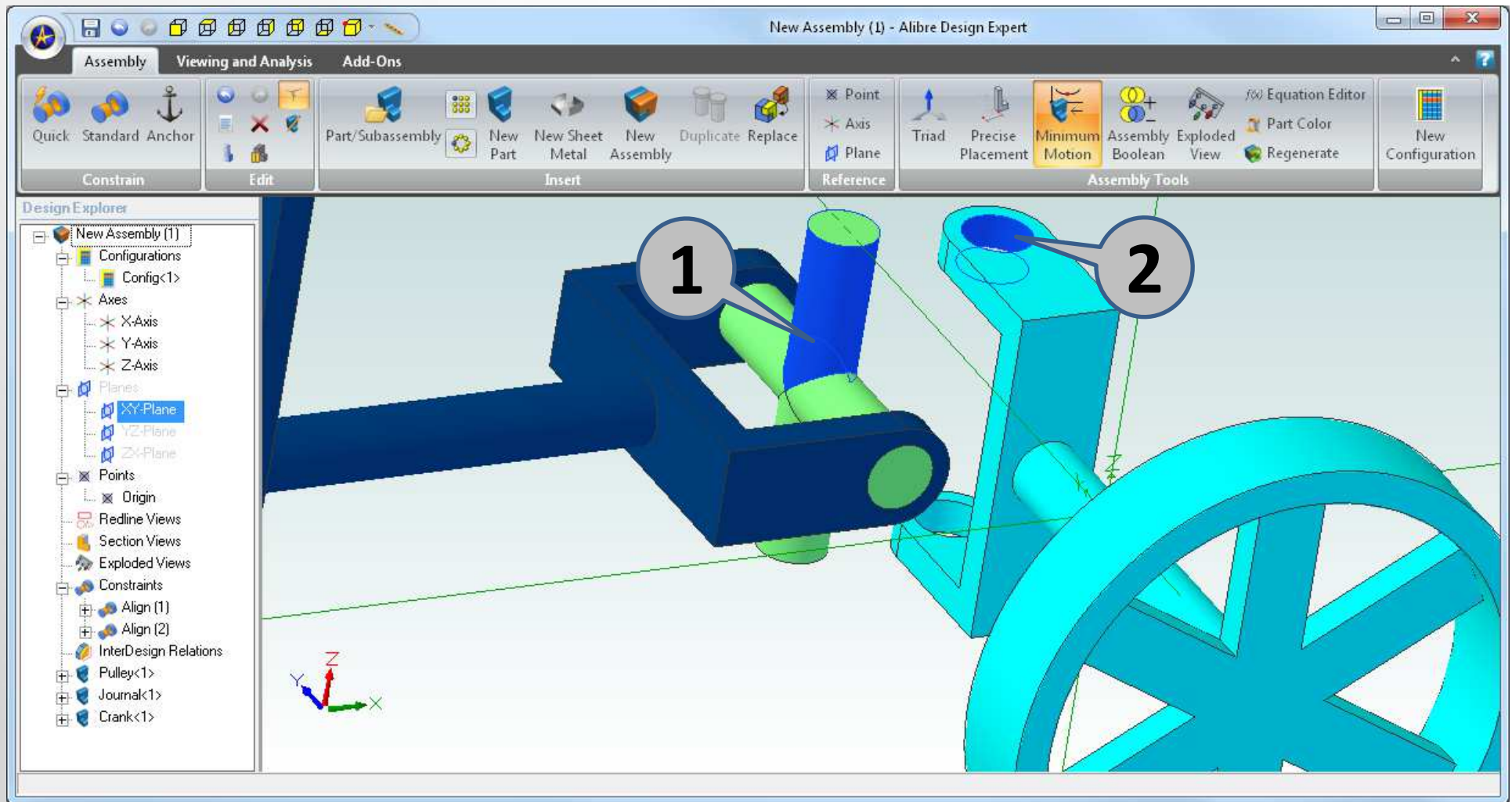
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.



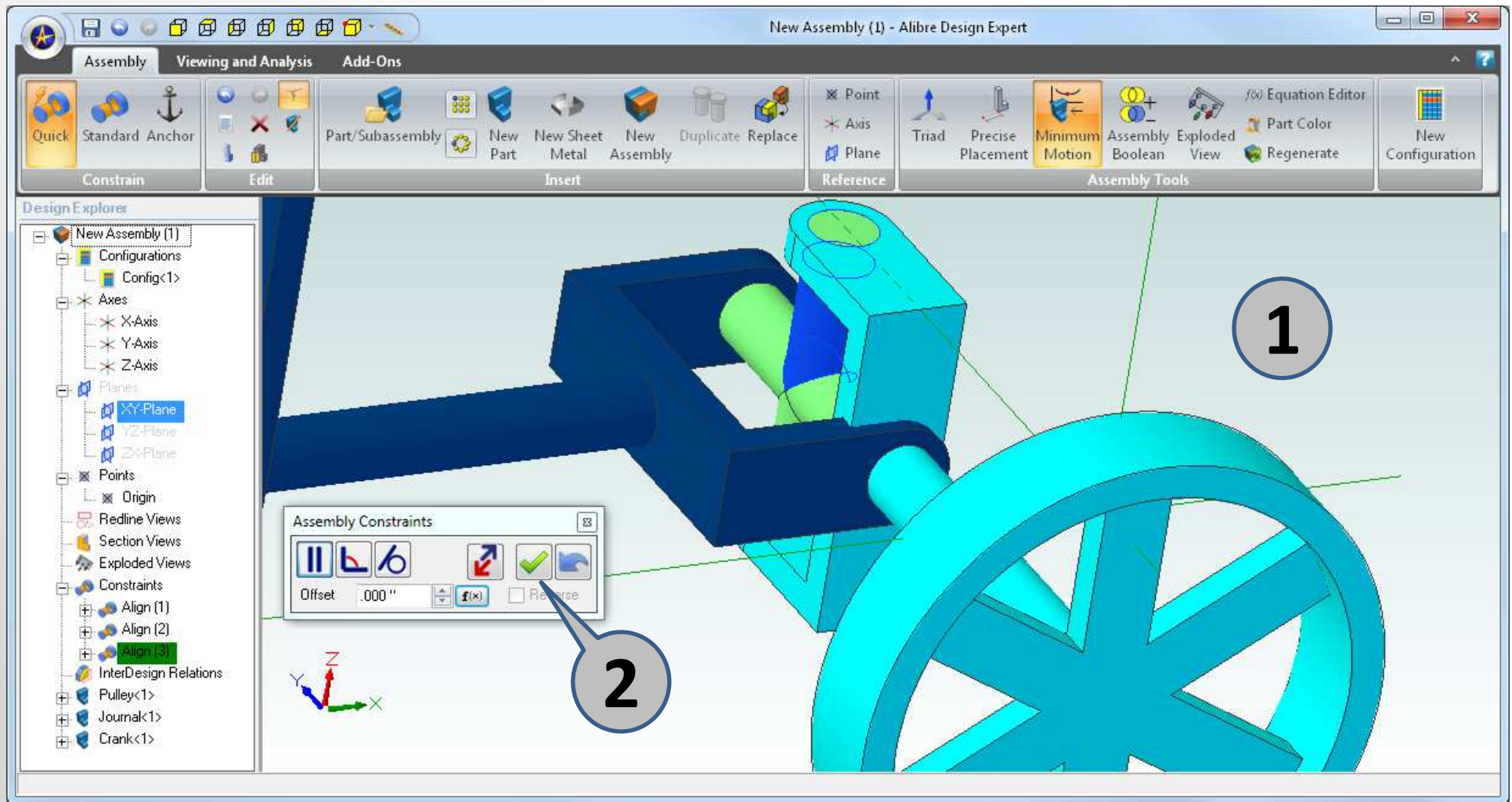
1. At this point the Quick Constraint dialog box should still be open.
2. Click on the indicated face
3. Click on the indicated face.



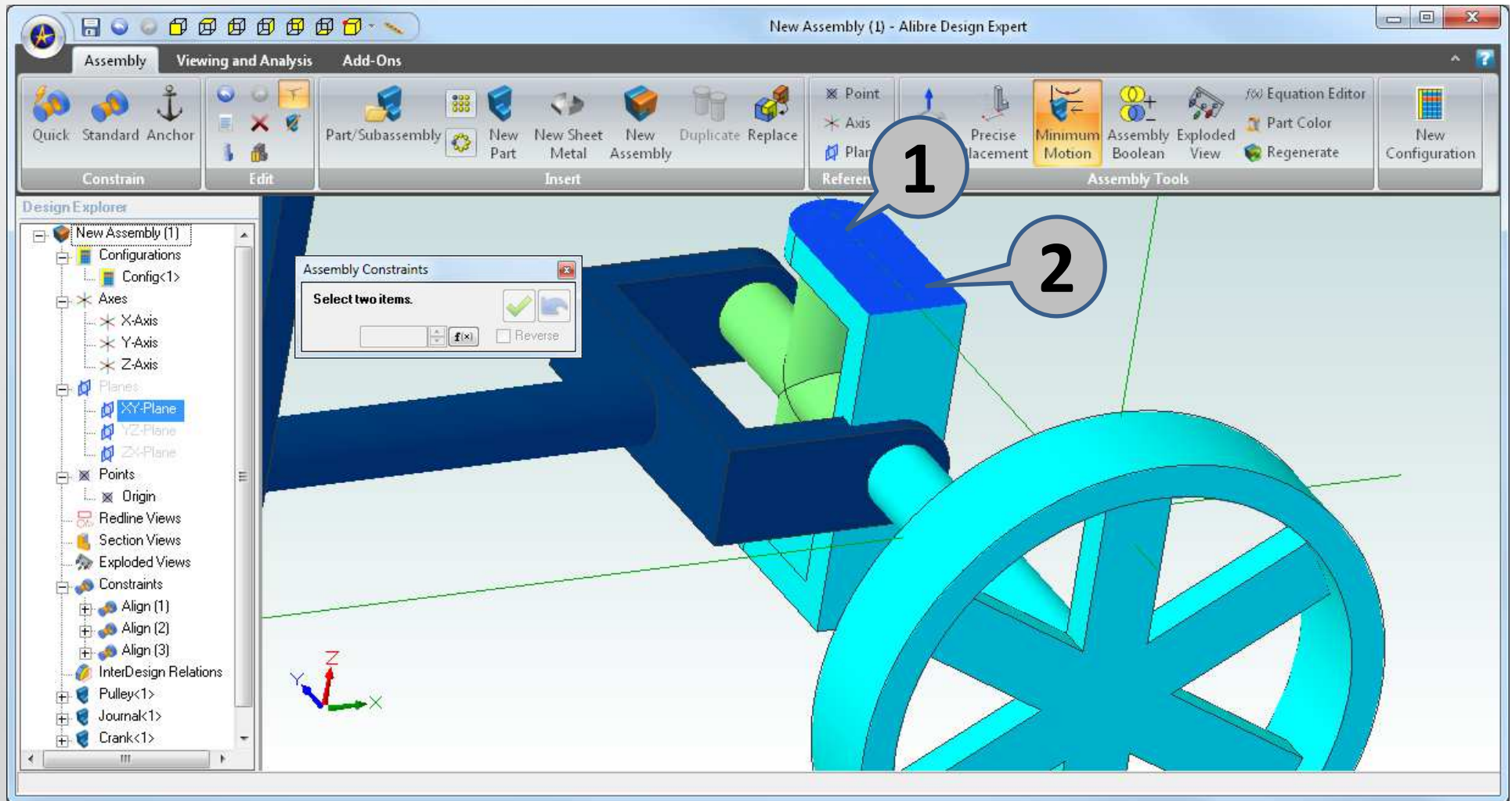
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.



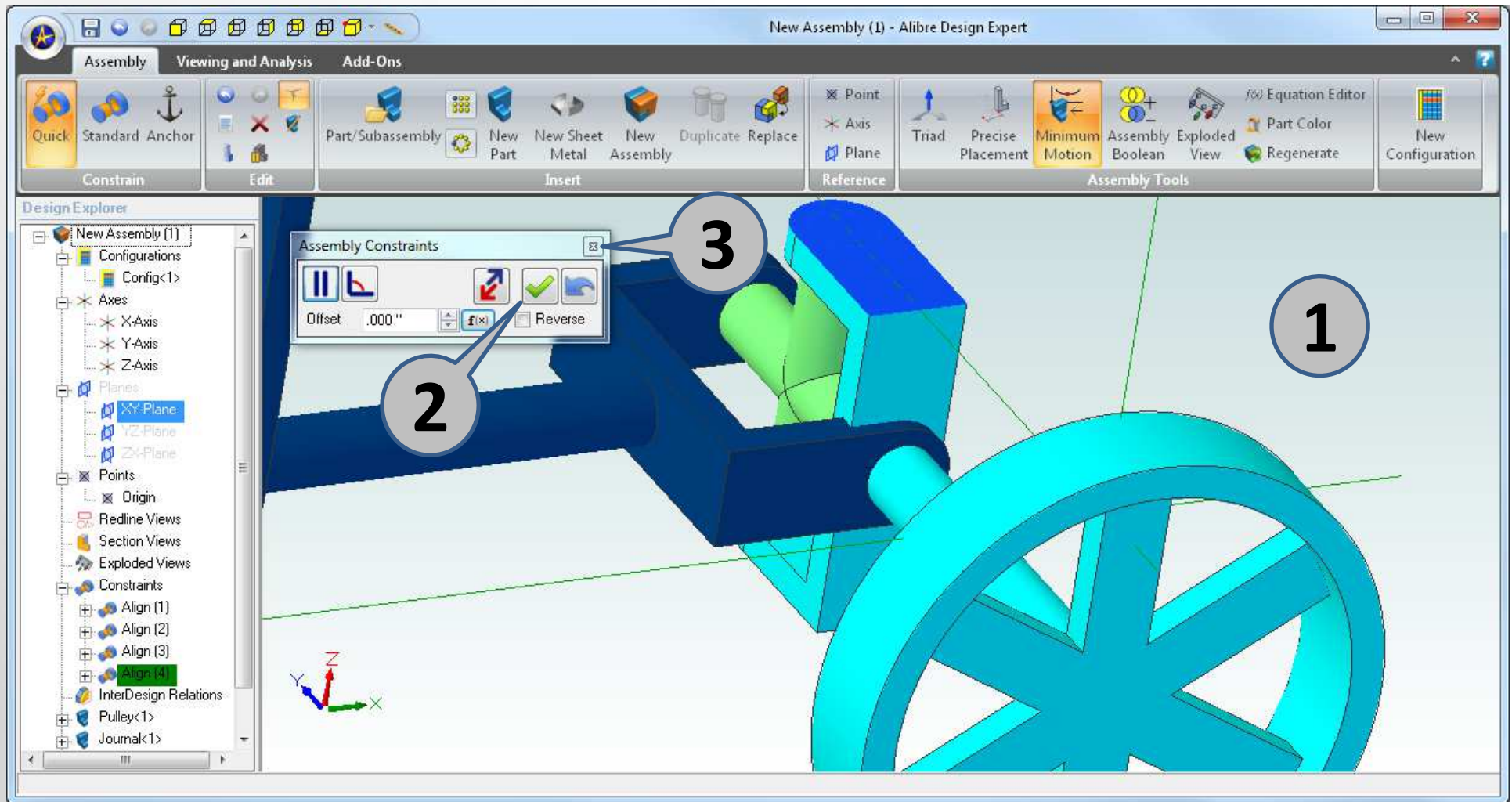
1. Click on the indicated face
2. Click on the indicated face.



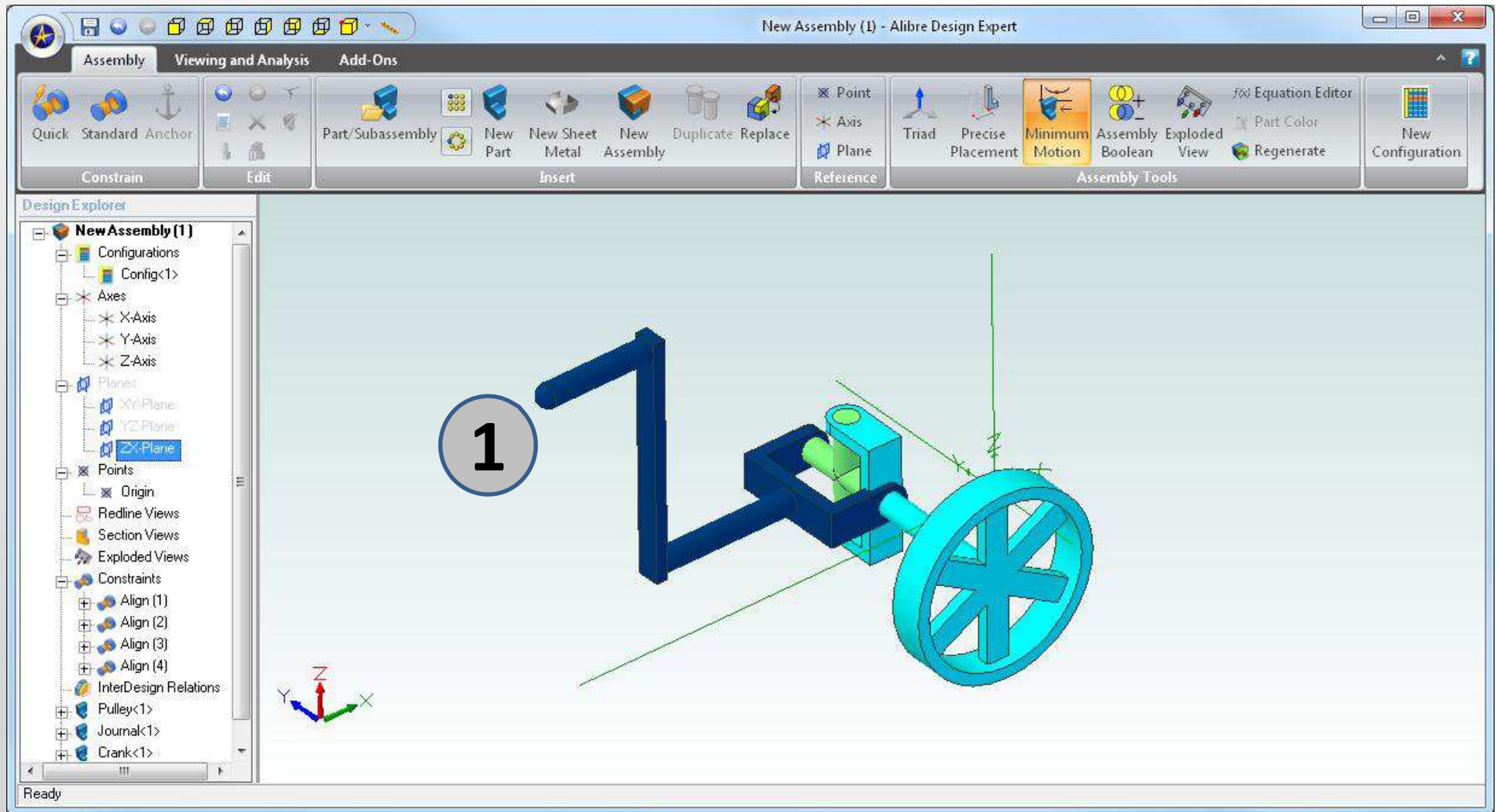
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.



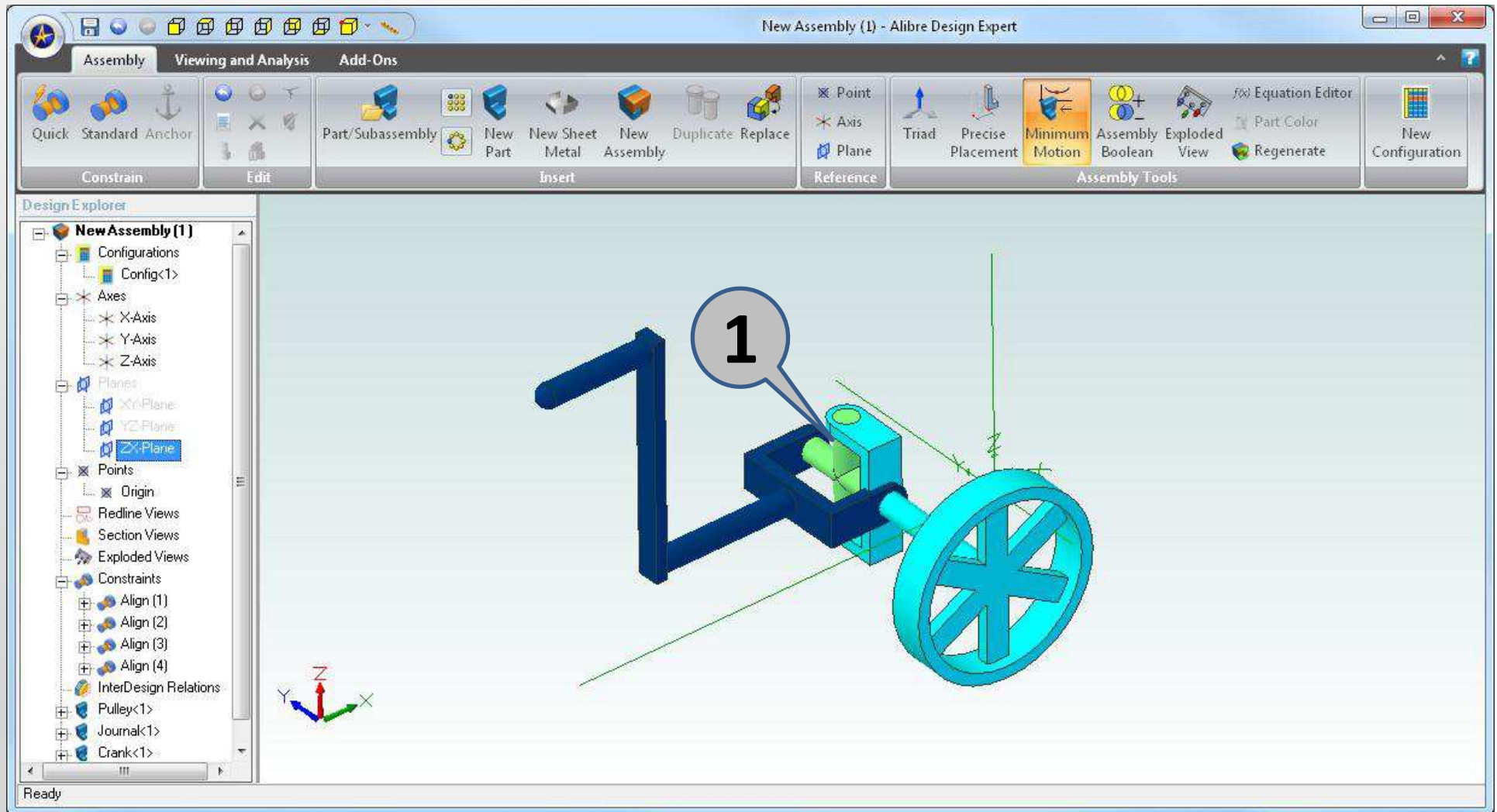
1. Click on the indicated face.
2. Click on the indicated face.



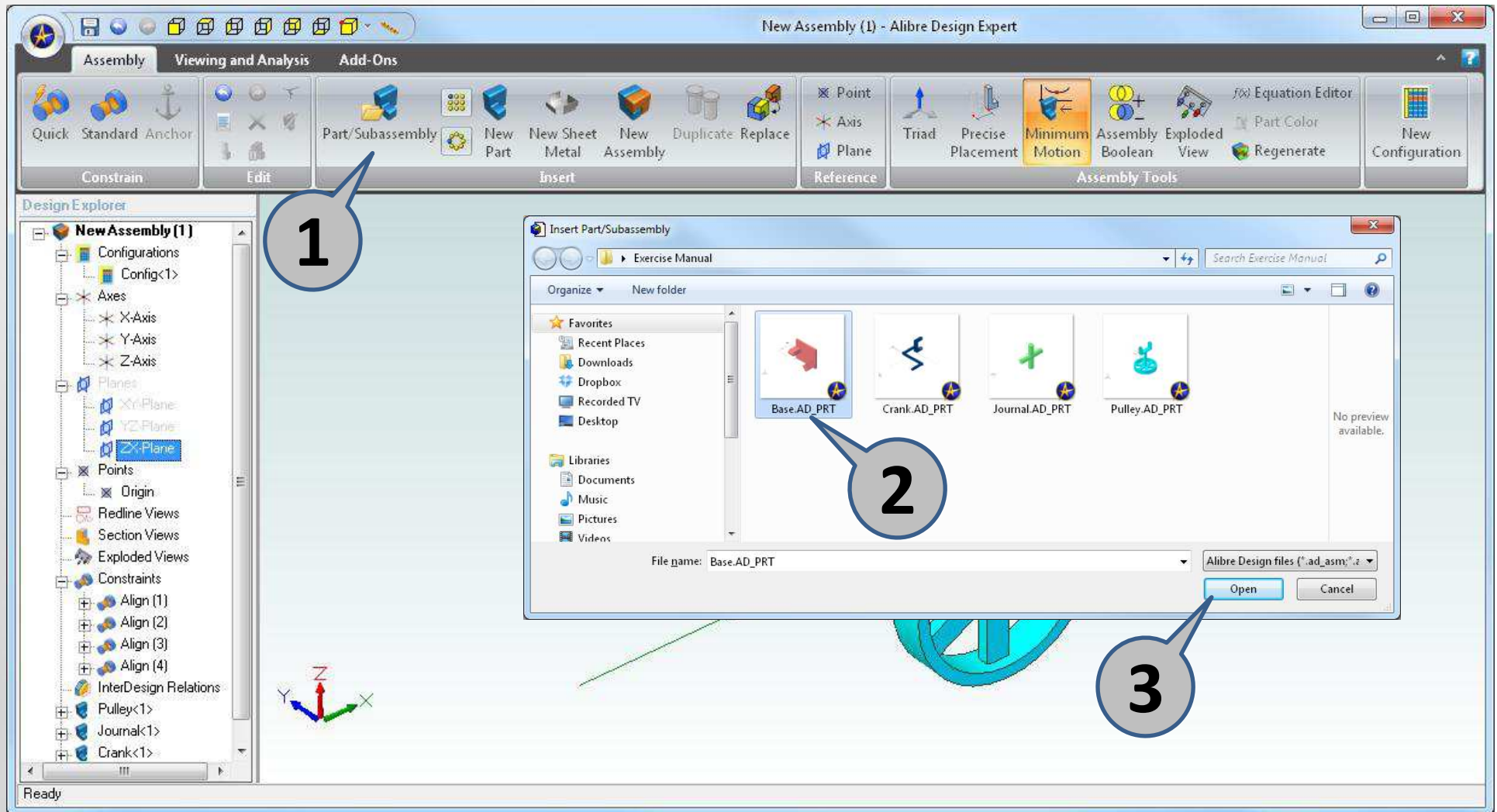
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
3. Close the **Assembly Constraints** dialog box.
4. Press the **Home** key on the keyboard (not shown).



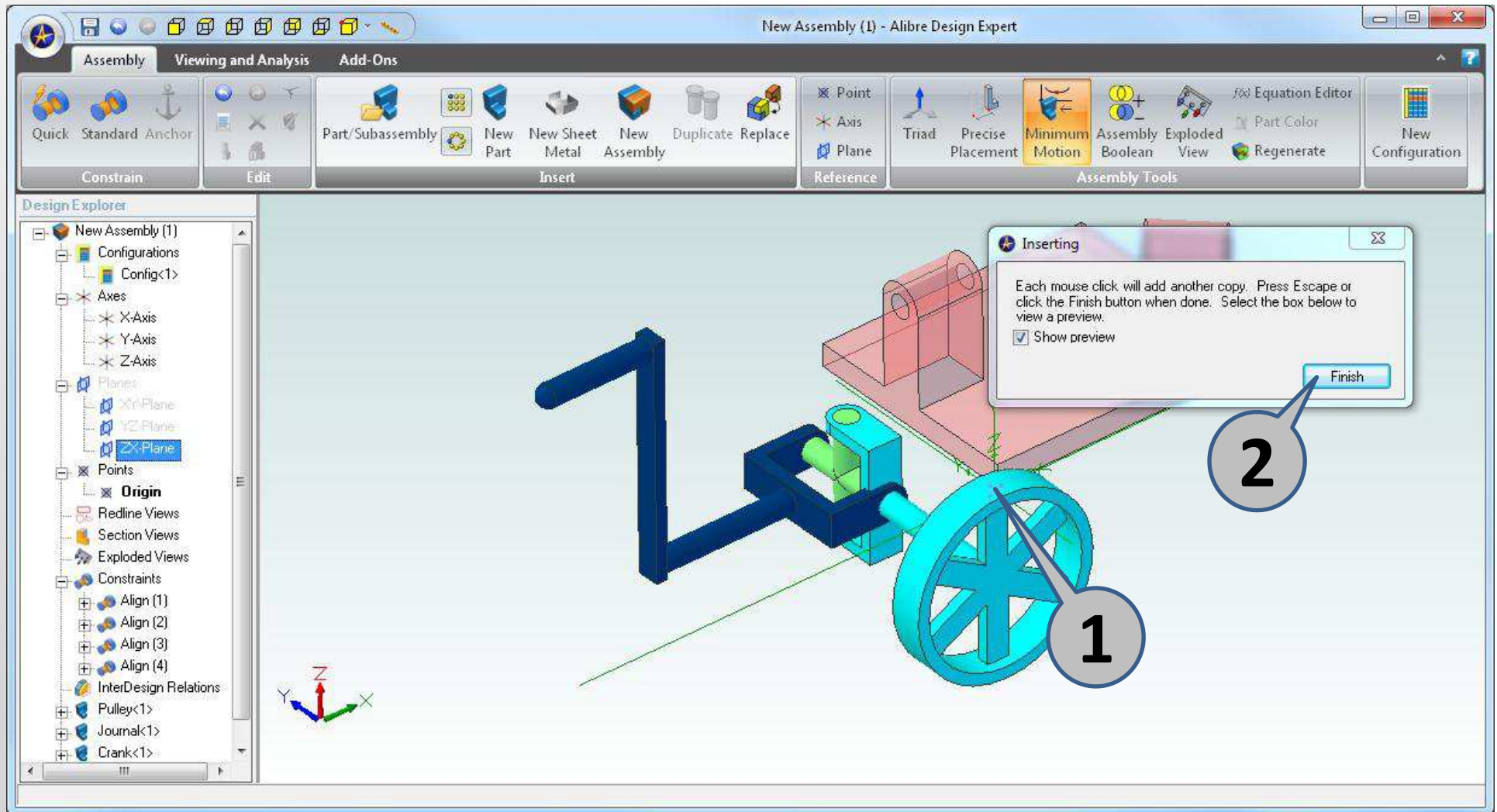
1. The view of the work area should be similar to the screen capture.



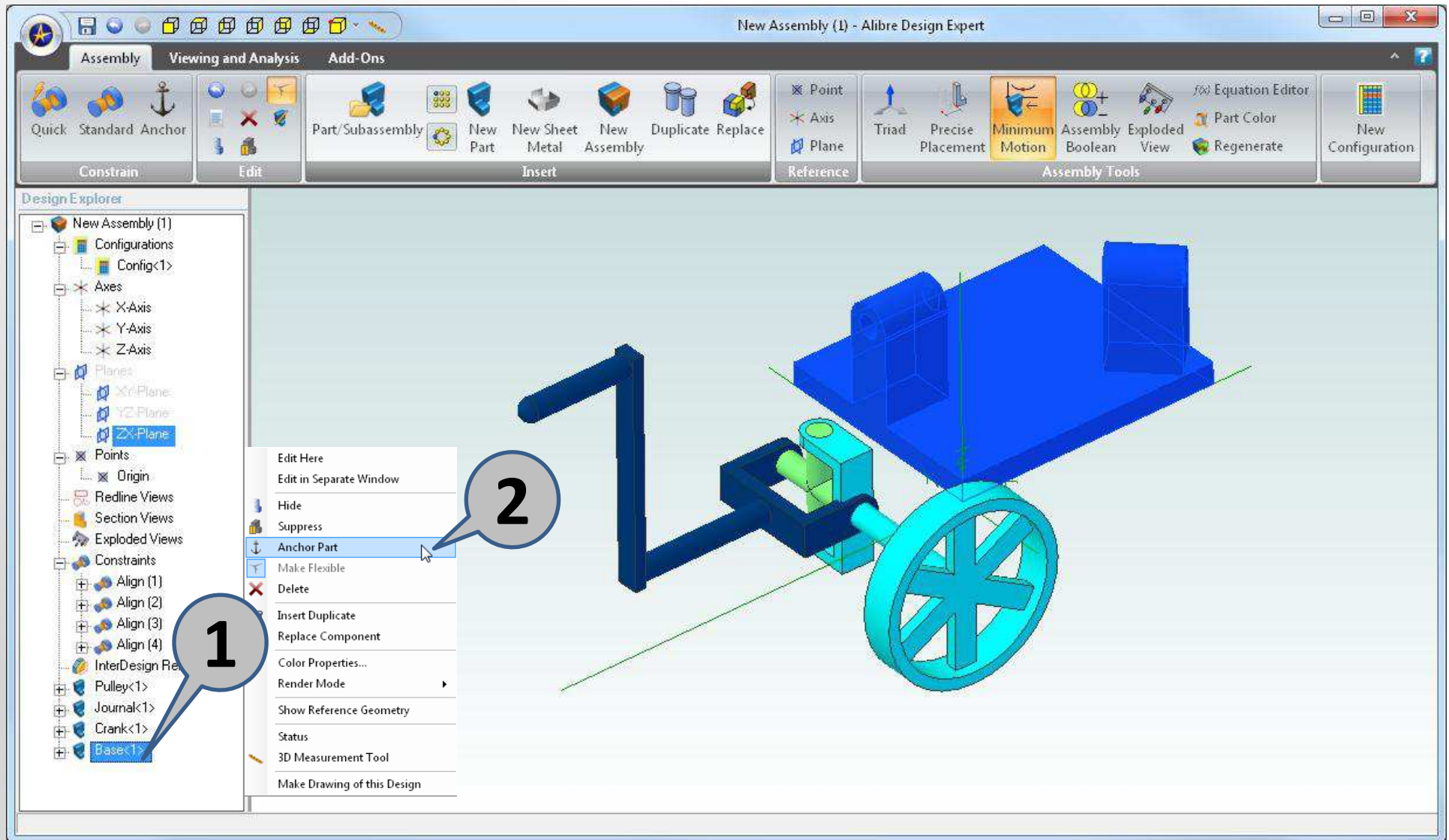
1. Position the mouse cursor over the **Journal** and roll the mouse wheel forward to zoom in. Achieve a similar result to the screen capture.



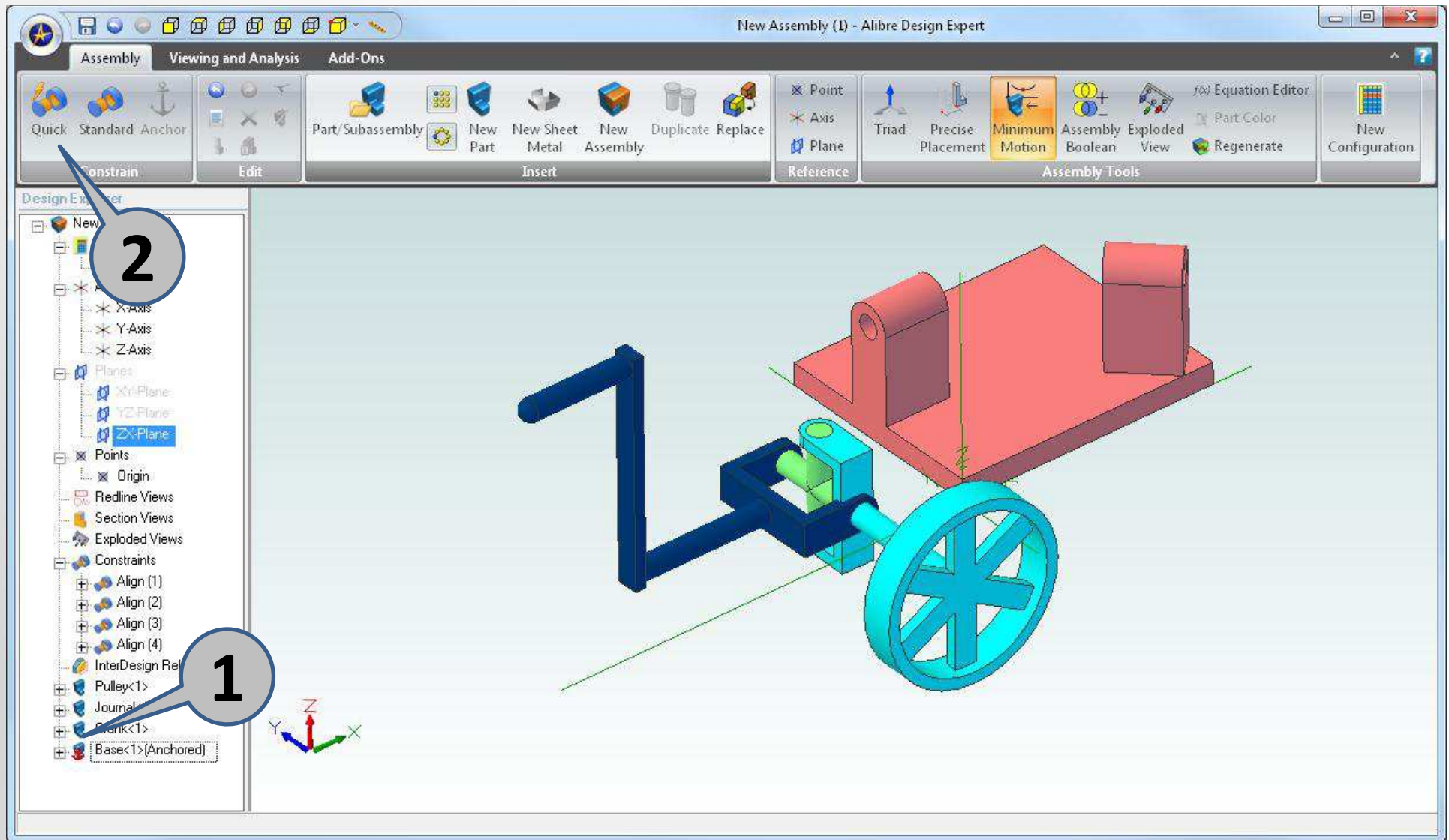
1. From the **Insert** group, select the **Part/Subassembly** icon.
2. Select the **Base** in the **Insert Part/Subassembly** dialog box.
3. Click **Open**.



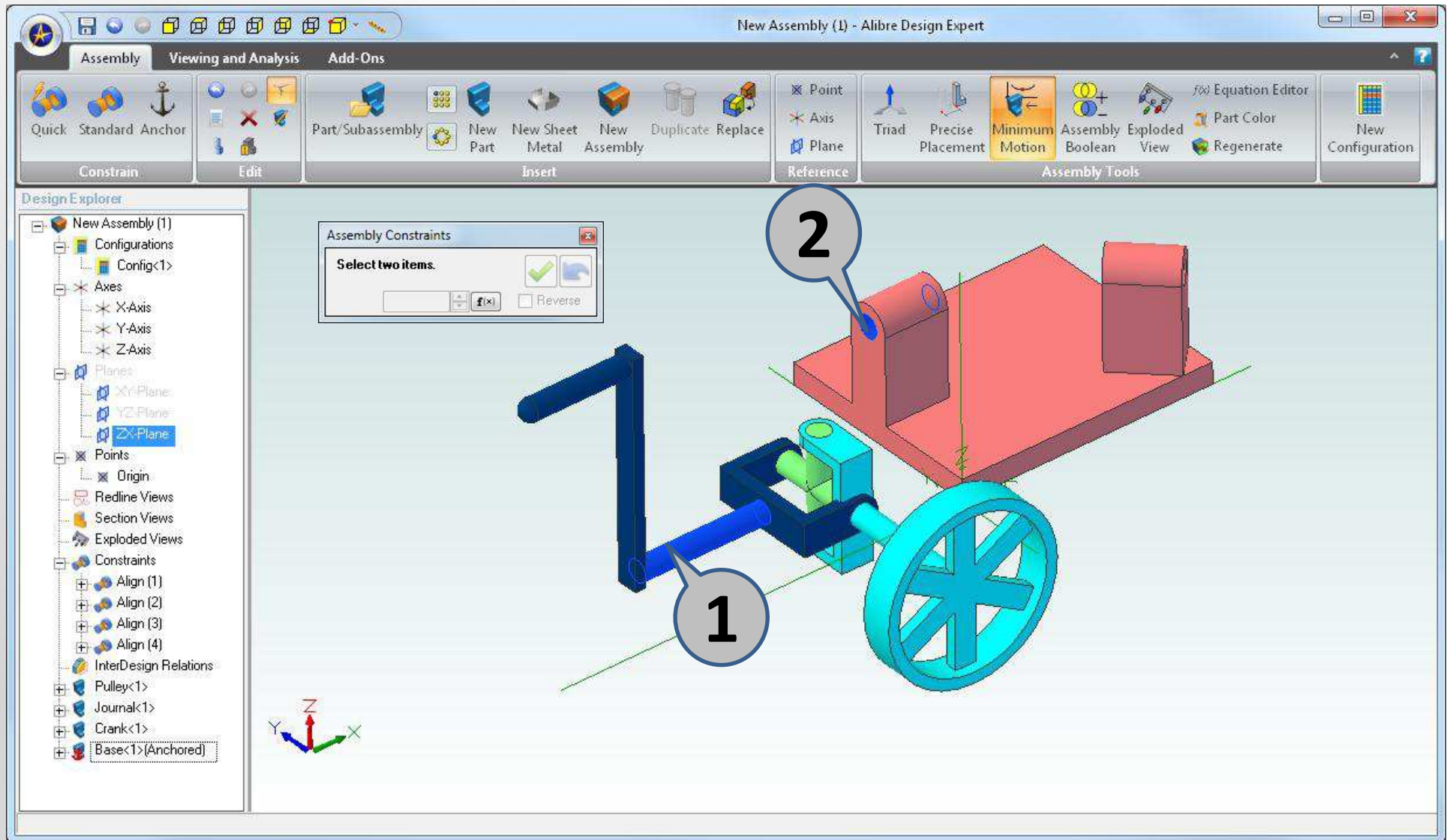
1. Position the mouse cursor (with the transparent preview of the base) over the **Origin point**. Proper positioning is confirmed by the color change of the Origin point. Click and release once to create a single instance of the part.
2. Click on the **Finish** button on the **Inserting** dialog box.



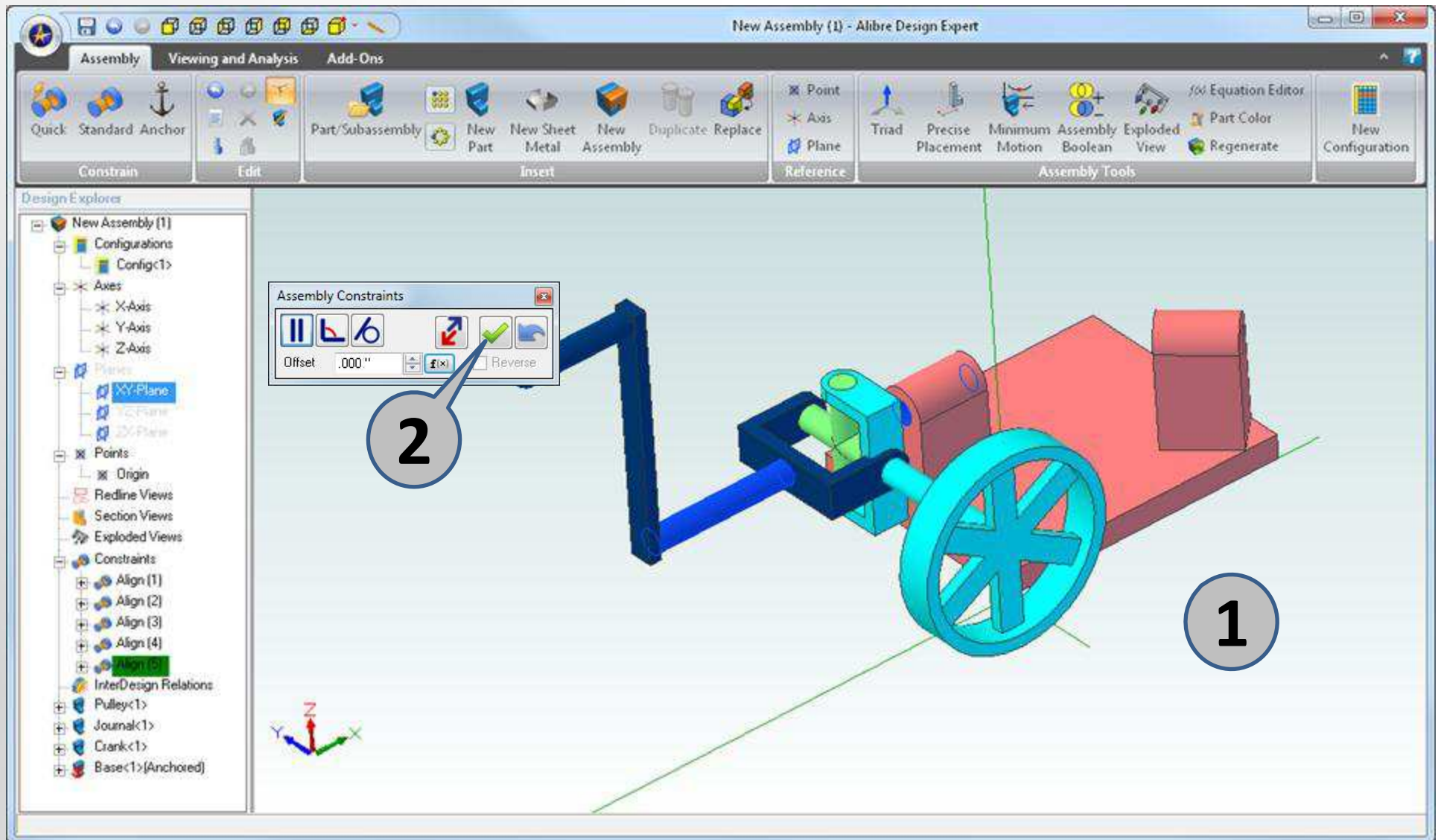
1. Right click on the **Base** item that is listed in the **Design Explorer**.
2. Select the **Anchor Part** command.



1. Confirm that the **Base** part is anchored by the small, red anchor graphic that now appears on the icon; and the (Anchored) description that's appended to its **name** in the **Design Explorer**.
2. Select the **Quick Constraint** command.

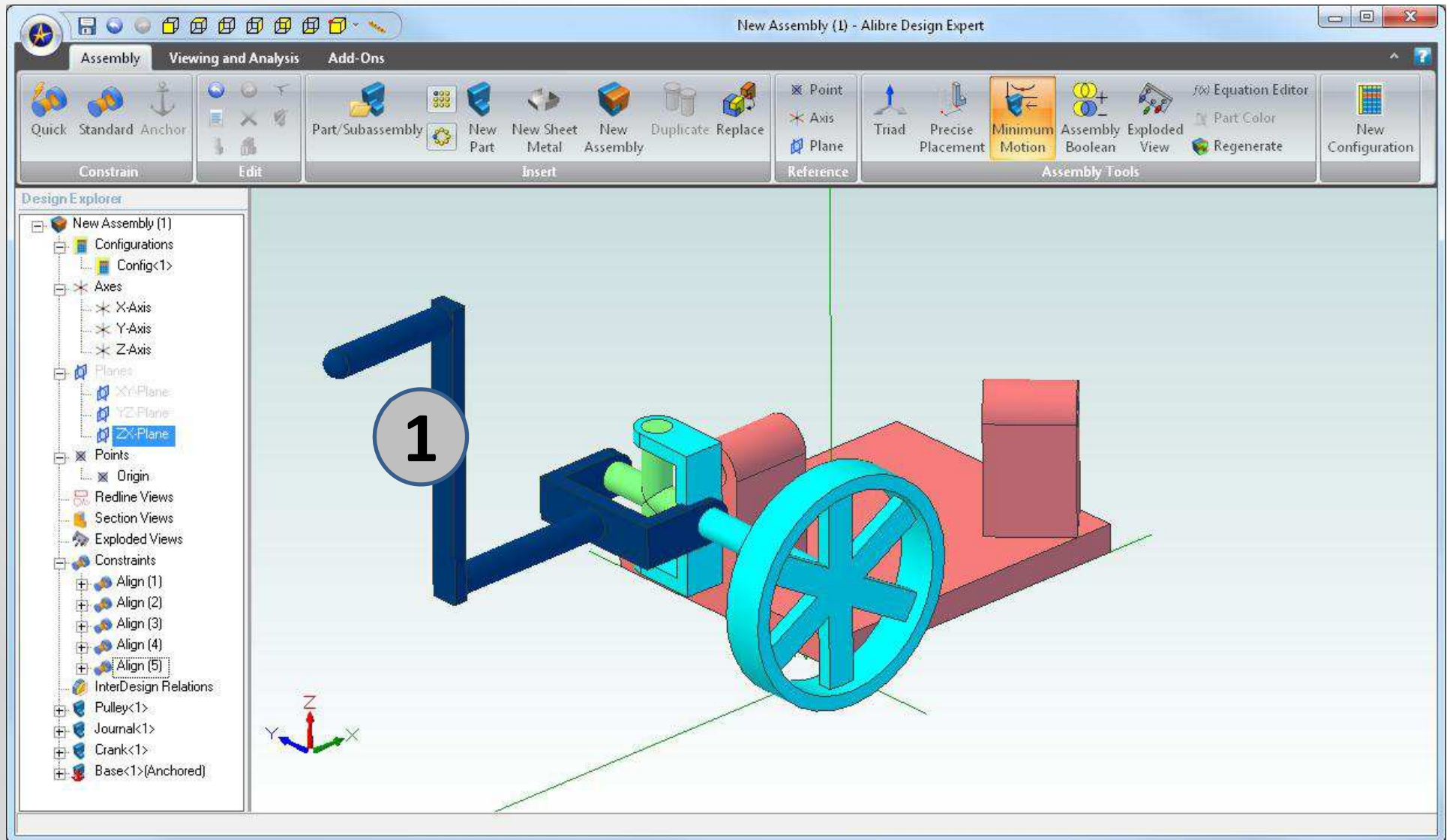


1. Click on the indicated face
2. Click on the indicated face.

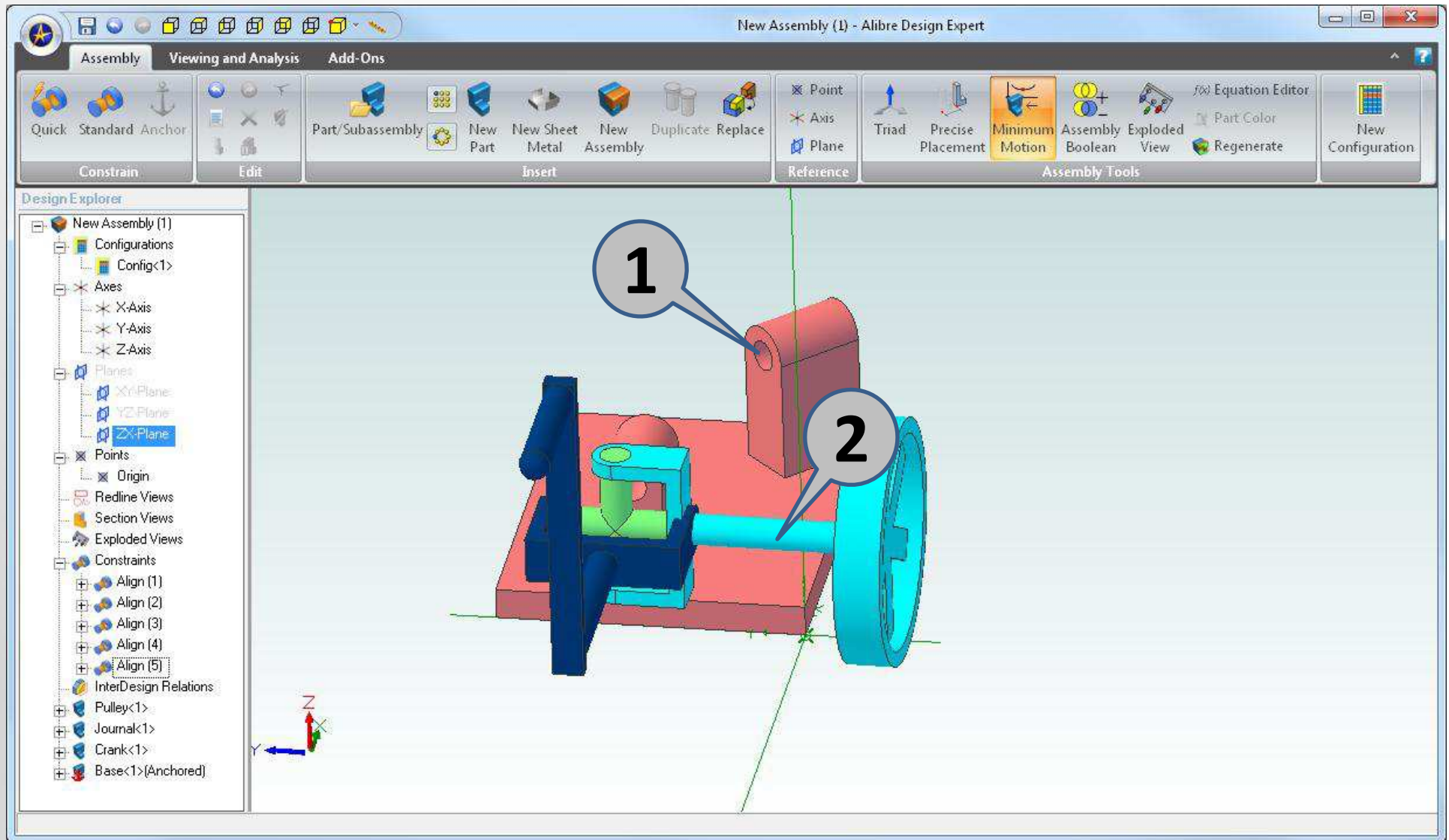


1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
3. Press the **Home** key on the keyboard (not shown).

Since the **Base** is anchored it cannot move. In this case, the **Crank** will always move in relation to the **Base**; the order of selection is irrelevant.

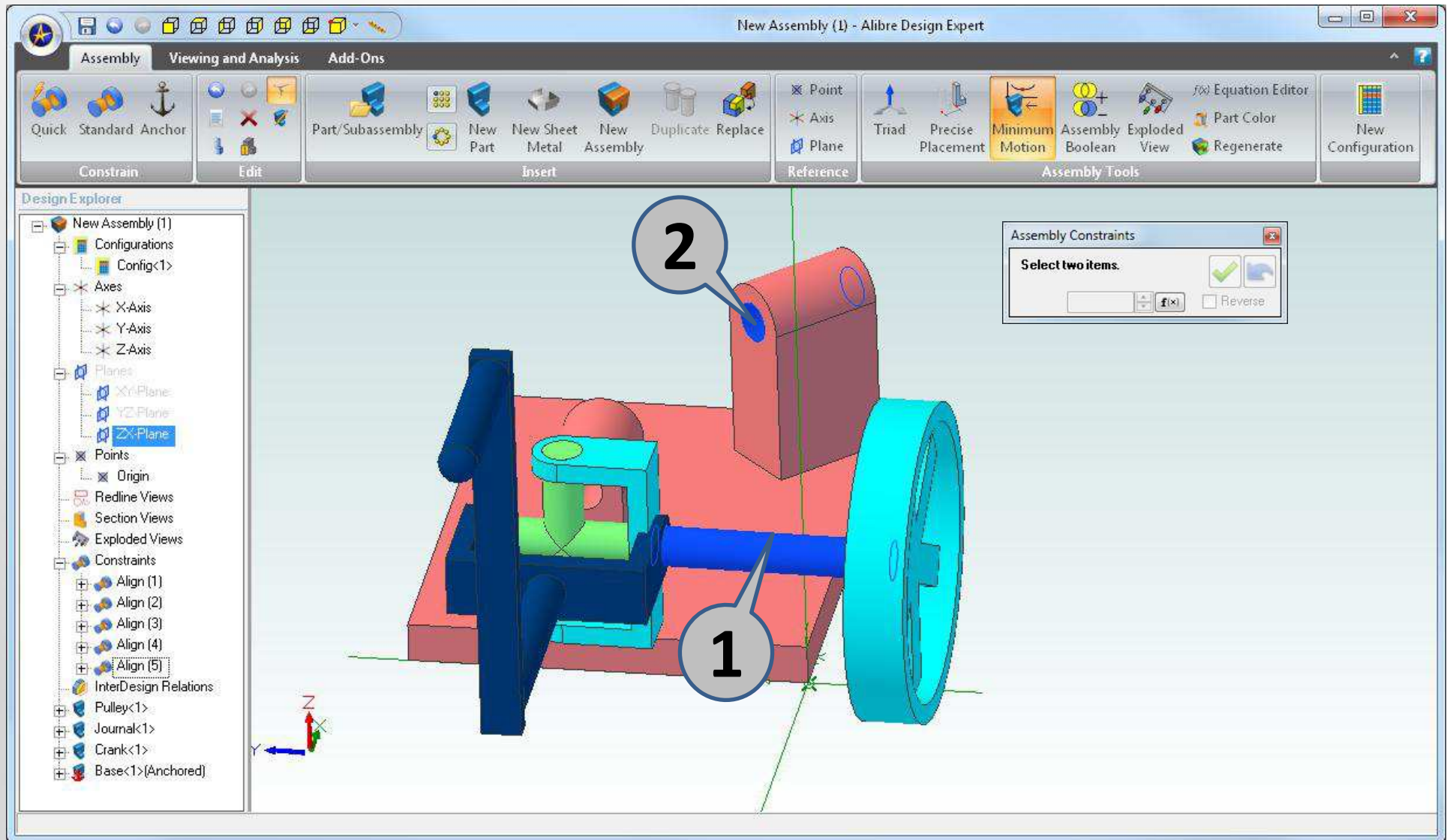


1. The view of the work area should be similar to the screen capture.

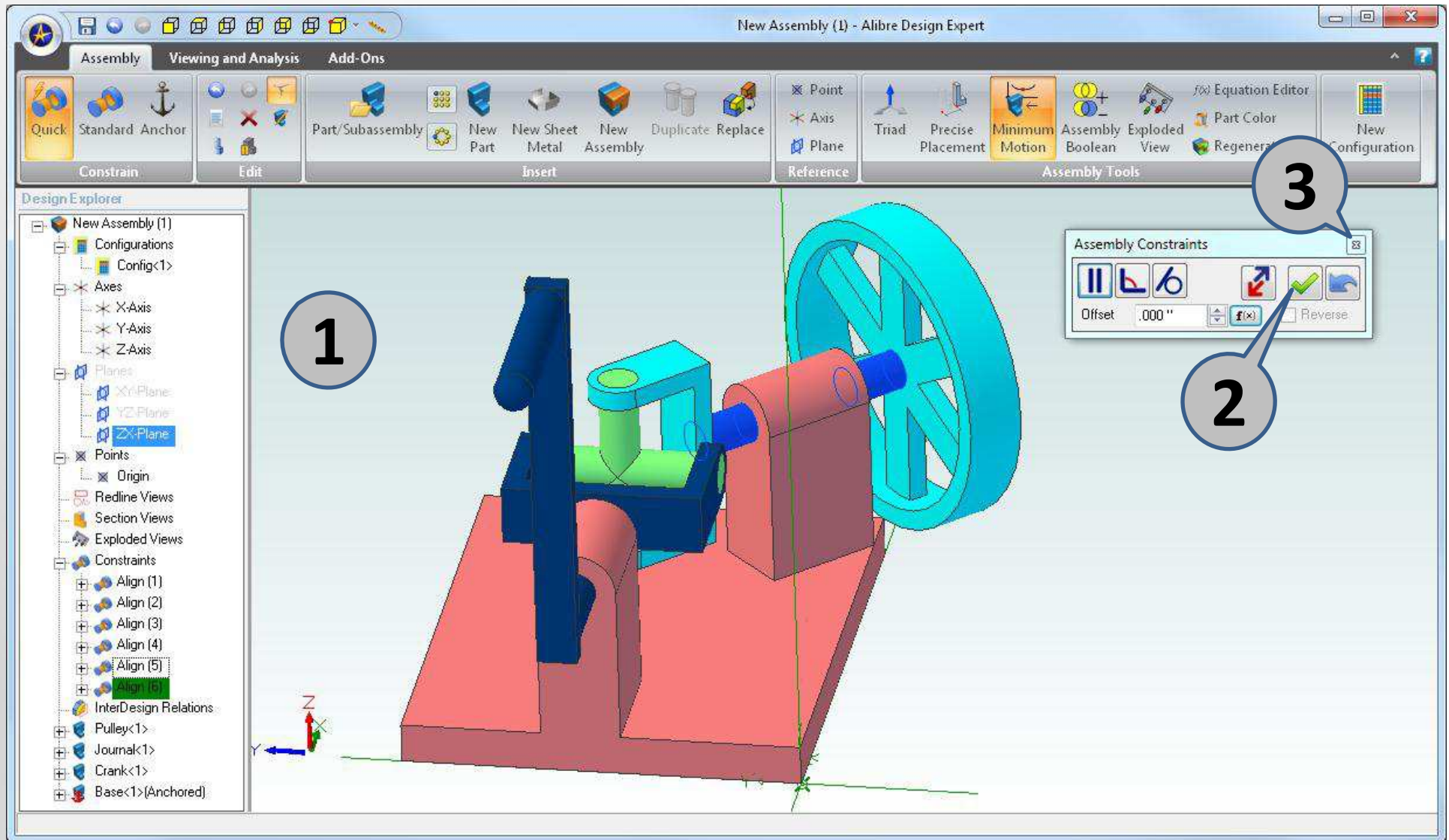


Rotate and Zoom the workspace so you can easily see the following items.

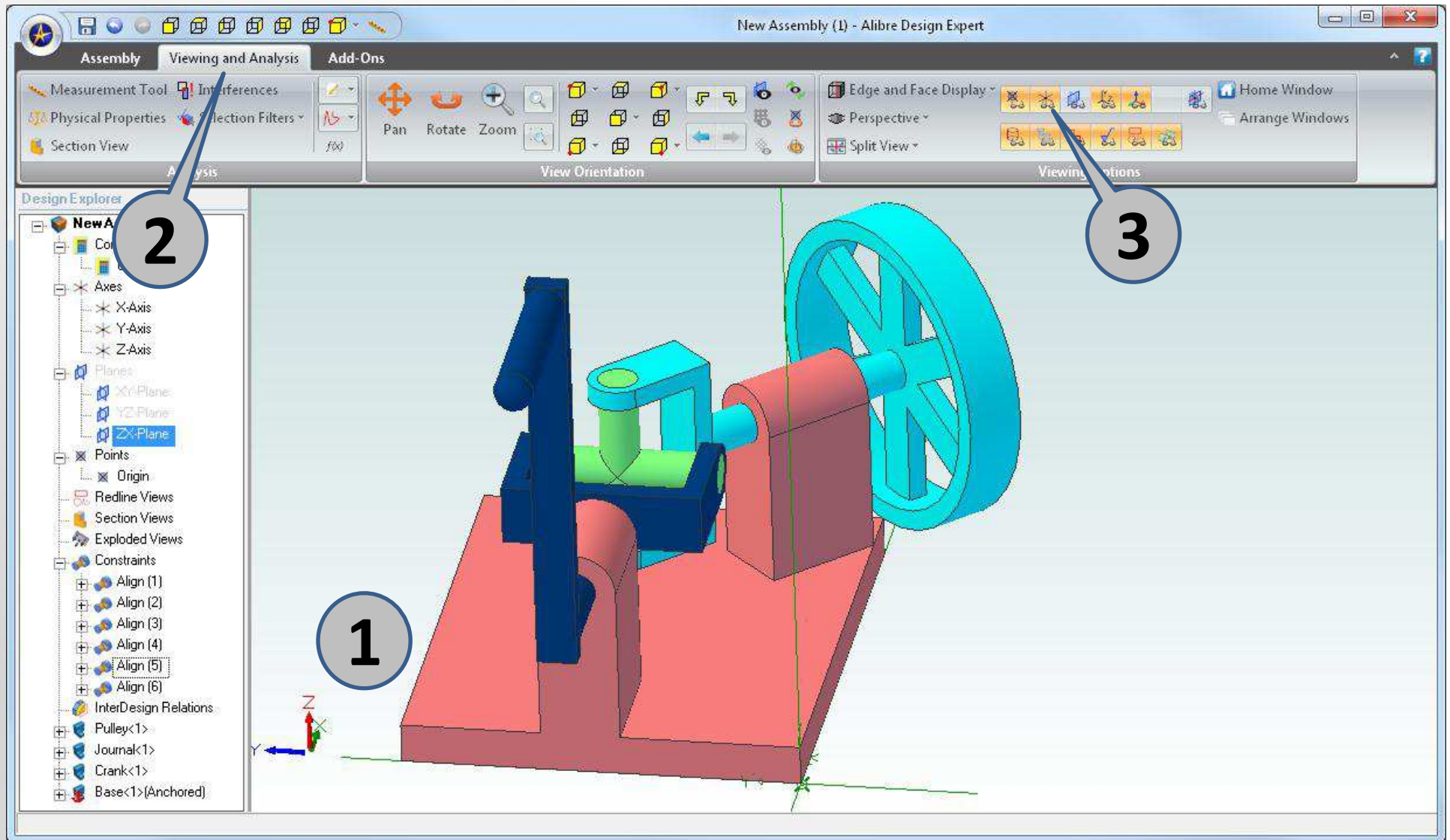
1. The inside surface on the Base.
2. The cylindrical shaft on the Crank.



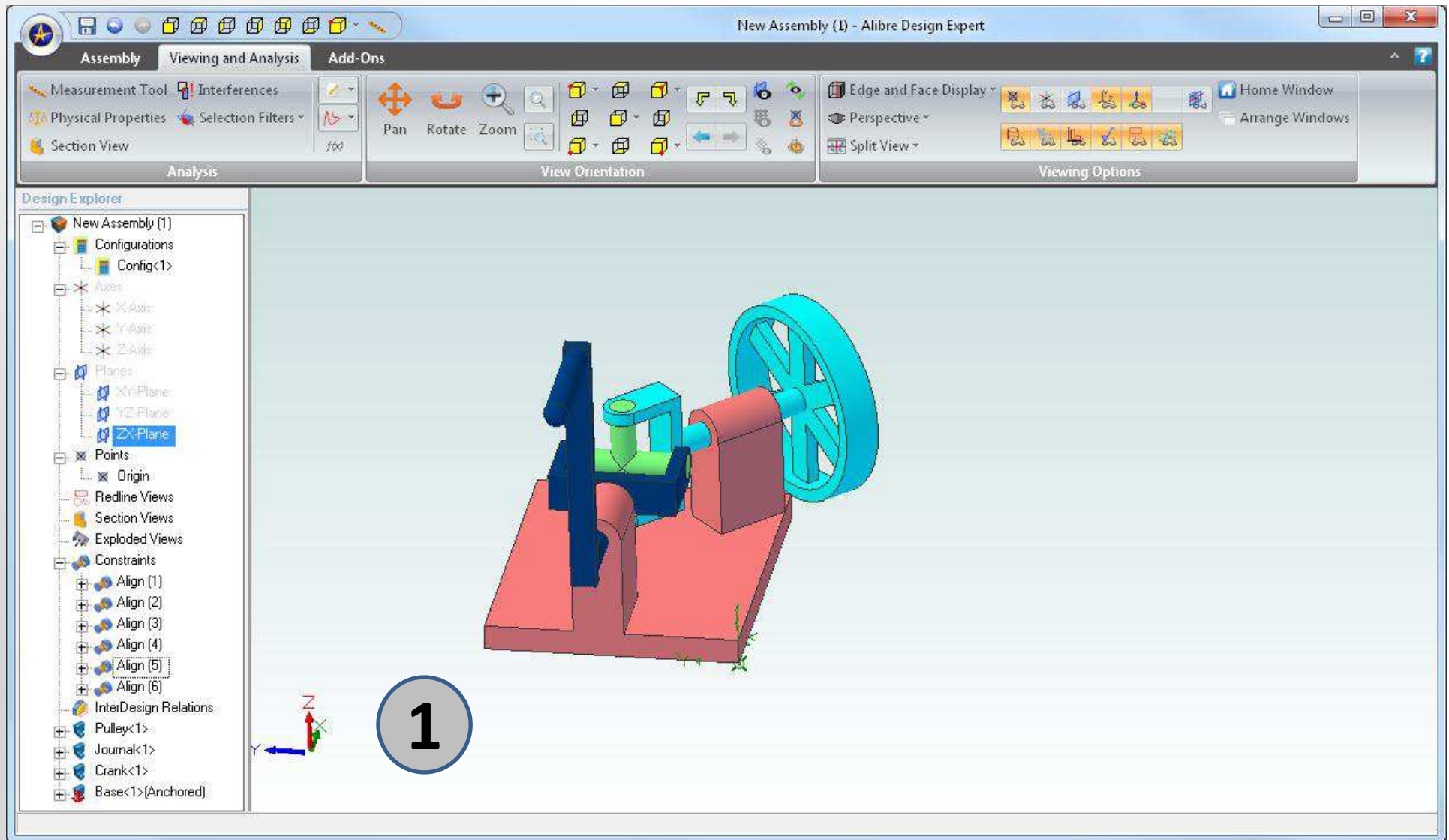
1. Click on the indicated face
2. Click on the indicated face.



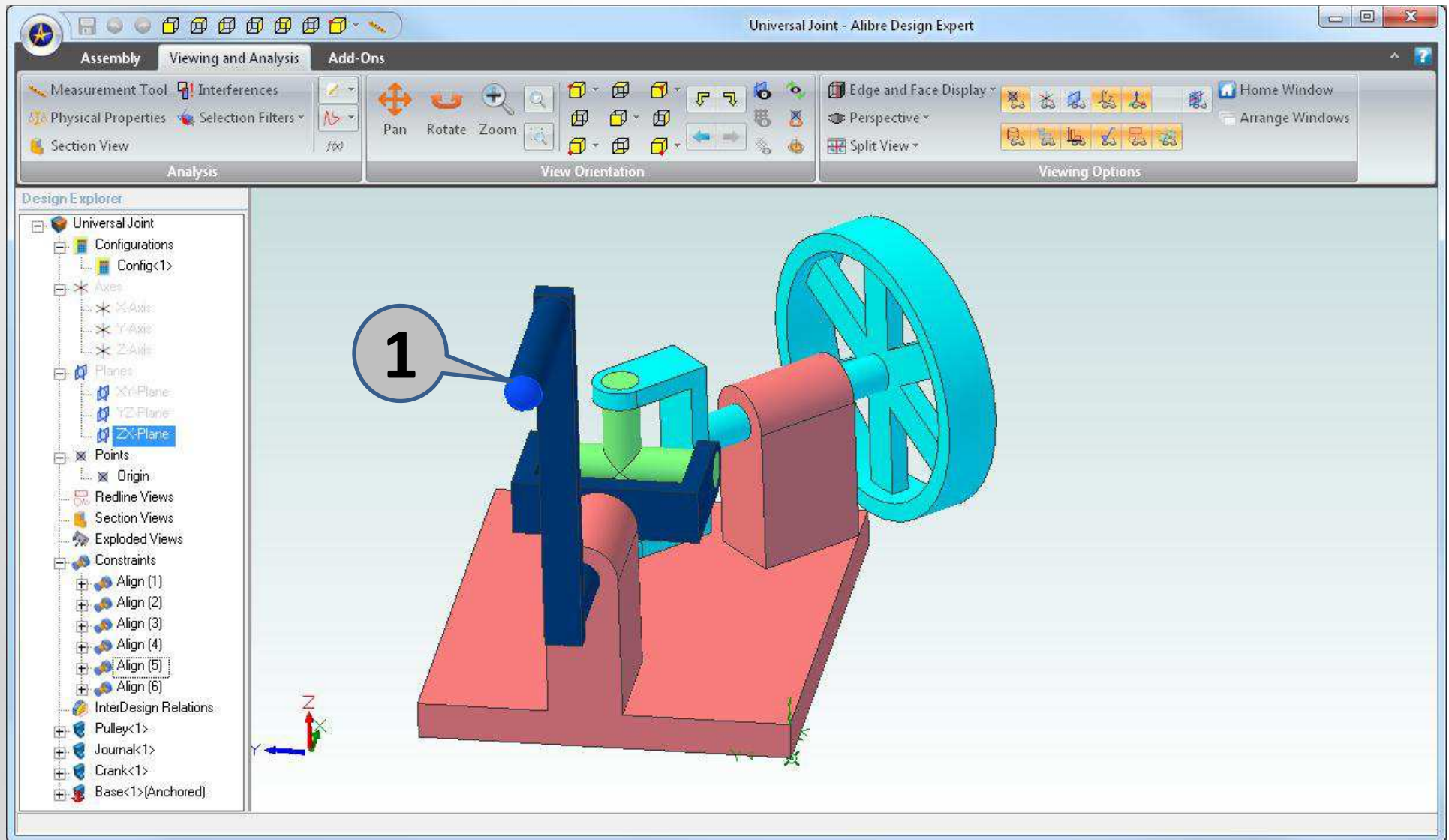
1. Confirm the preview of the results.
2. Click on the button with the green check-mark icon to accept the constraint.
3. Close the **Assembly Constraints** dialog box.
4. Press the **Home** key on the keyboard (not shown).



1. The view of the work area should be similar to the screen capture.
2. Click on the **Viewing and Analysis** group.
3. Click on the **Toggle Axes** icon.



1. Position the mouse cursor over the **Journal** part, and roll the mouse wheel backward to zoom out slightly, until the view is similar to the screen capture.



1. Position the mouse cursor over the rounded end-face of the **Crank** part. Click and hold the mouse button; then move the mouse around to simulate turning the crank by hand. Notice that the mechanism responds as it would in the real world.
2. **Save** this Assembly to the same location as the part files with the name: Universal Joint. (not shown)
3. **Close** the **Assembly** workspace (not shown).
4. This concludes the Tutorial.