Module 1G:
Creating a Circle-Based Cylindrical Sheet-metal Lateral Piece with an Overlaying Lateral Edge Seam And Dove-Tail Seams on the Top Edge

In *Module 1A*, we have explored the method of creating a cylindrical sheet metal part using the **Revolve** tool; with this method, a tiny gap is created along the seams of the lateral piece. In practical design, this method is suitable for situations where the folded-up lateral piece is to be welded (*Figure 1G-1A*); however, there are occasionally situations where the folded-up lateral piece is to be connected by an overlapping seam (*Figure 1G-1B*). In this Module, we will learn how to create a cylindrical sheet metal lateral piece with the **Contour Flange** tool. In addition, we will explore a "work-around" method of creating dove-tail seams on the top edge of the lateral piece.

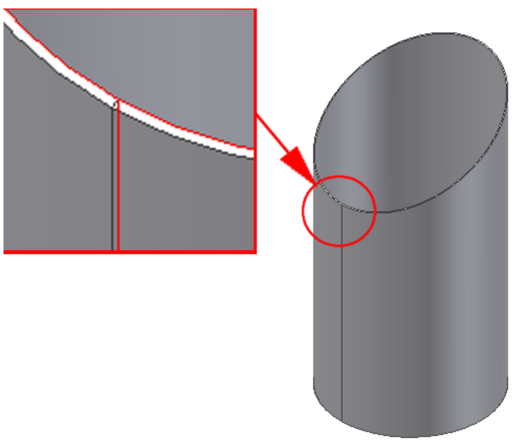


Figure 1G-1A: The tiny gap is created along the seams of the lateral piece when the creating a cylindrical sheet metal part using the Revolve tool.

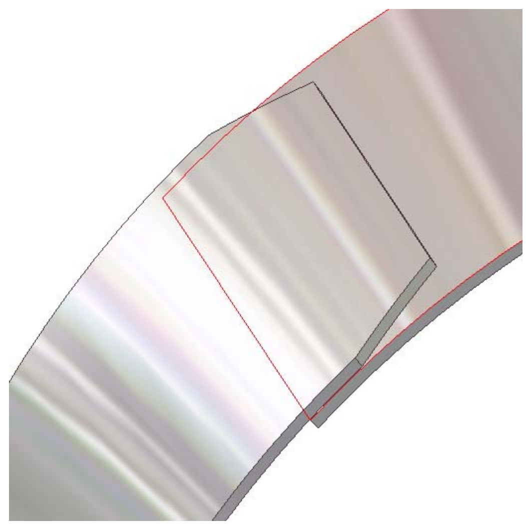


Figure 1G-1B: The overlapping Seam.

Step 1: Construct the circular base profile with an overlapping seam

First, we will create a sheet metal file with inch as units. Launch Autodesk Inventor. Create a new sheet metal file (go to the menu **File→New**, or click on the **New** icon on **Standard Tool Bar.** The Open window appears. Select the **English** tab; then select the **Sheet Metal (In).ipt** template, click **OK** button (*Figure 1G-2A*).

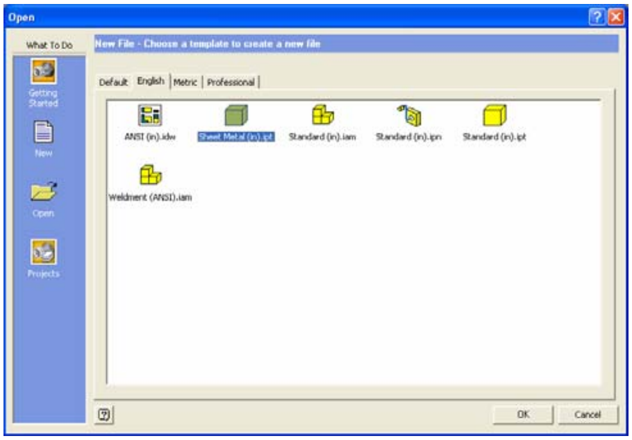


Figure 1G-2A: Starting a new Sheet Metal (In).ipt file.

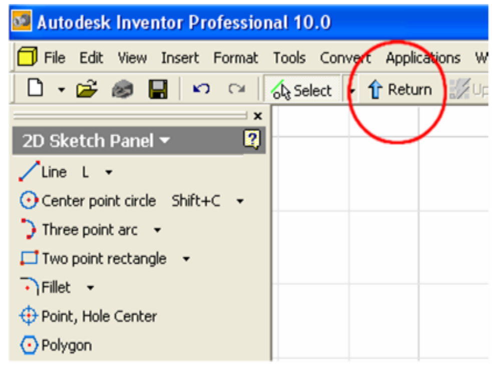


Figure 1G-2B: Dismissing the default Sketch1 by clicking the Return button.

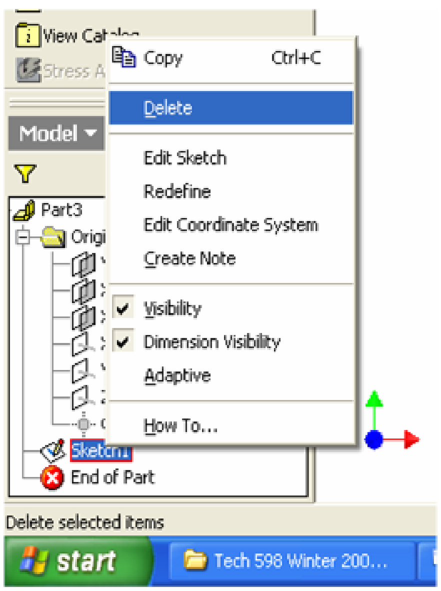


Figure 1G-2C: Deleting the default Sketch1.

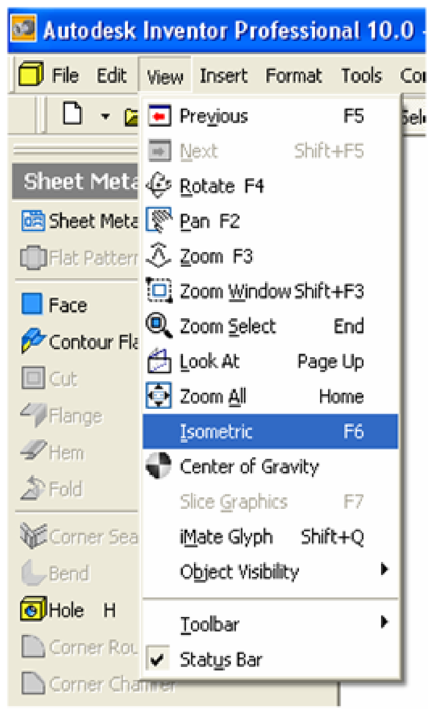


Figure IG-2D: Switching to Isometric View.

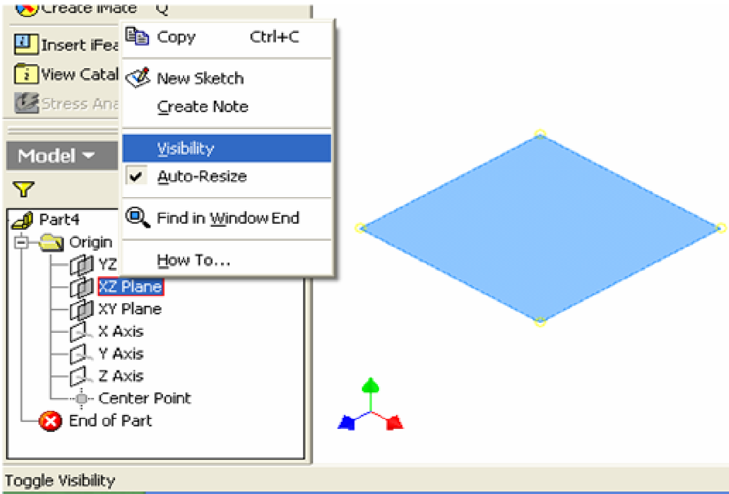


Figure IG-2E: Turning on the Visibility option.

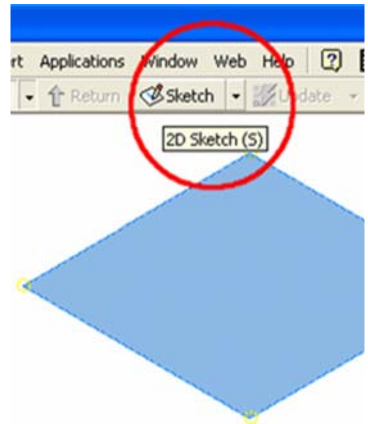


Figure IG-2F: Starting a new Sketch.

A new sheet metal file opens with two panels docked on the left side of the screen: **2D Sketch** (top) and **Model** (bottom). The **Sketch1** opens automatically on the screen; and **Sketch1** feature appears on the **Model** panel. Click the **Return** key in the **Inventor Standard** tool bar (*Figure 1G-2B*) to dismiss the **Sketch1** feature.

In the **Model** panel, click the cross on the left of the **Origin** folder to view the **Planes**, **Axis** and **Center Point** features; click-select the **Sketch1** feature and right-click for the shortcut menu and choose the **Delete** option to delete it (*Figure 1G-2C*). Next, go to the View→Isometric menu to switch to an isometric view (*Figure 1G-2D*). Next, select the **XZ Plane**, then right-click to open the short cut menu and check the **Visibility** option (*Figure 1G-2E*). Next, click-select the **XZ Plane** and then the **Sketch** button on the **Inventor Standard** tool bar to start a new sketch (*Figure 1G-1F*); **Sketch2** feature appears in the **Model** panel; highlight the **Sketch2** name and type *Base Profile* to rename the feature.

Next, click-select the **XZ Plane** and then the **Look At** tool button in the **Inventor Standard** tool bar (*Figure 1G-2G*, left) to switch to an orthographic normal view (*Figure 1G-2G*, right). Now, we are ready to start the elliptical base profile.

Click-select the **Project Geometry** tool from the **2D Sketch** tool panel and then the **Center Point** from the **Model** panel to project it onto the *Base Profile* **Sketch**; a green dot appears on the center of the *Base Profile* **Sketch**; next, select the **Center Point Circle** tool from the **2D Sketch** tool panel and create a horizontal line starting from the projected **Center Point**; use the **General Dimensions** tool to apply a 24-inch diameter dimension (*Figure 1G-3A*). Next, use the **Line** tool to create a horizontal line starting at the projected **Center Point** and ending at a convenient location on the right of the circle; and right-click for the shortcut menu and choose the **Restart** option (*Figure 1G-3B*) to create another horizontal line starting at the projected **Center Point** and ending at a convenient location on the left of the circle (*Figure 1G-3C*); next, select the **Trim** tool to trim off the lower half of the circle **(***Figure 1G-3D***)**; right-click for the shortcut menu and choose **Done [Esc]** option to exit the tool (*Figure 1G-3E*).

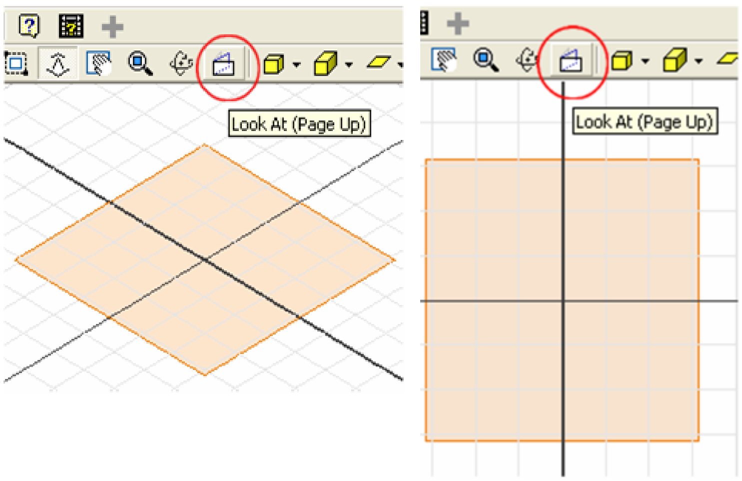


Figure IH-2G: Selecting the XZ Plane and clicking the Look At tool button (left); switching to an orthographic normal view (right).

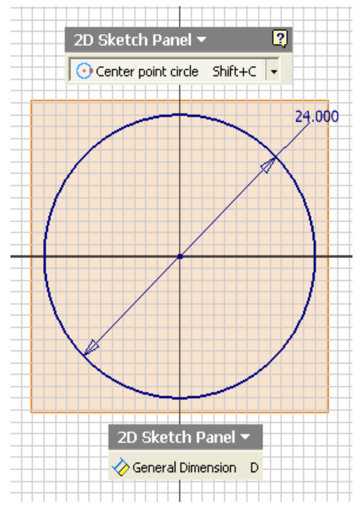


Figure 1G-3A: Creating the circle and applying a 24-inch diameter dimension.

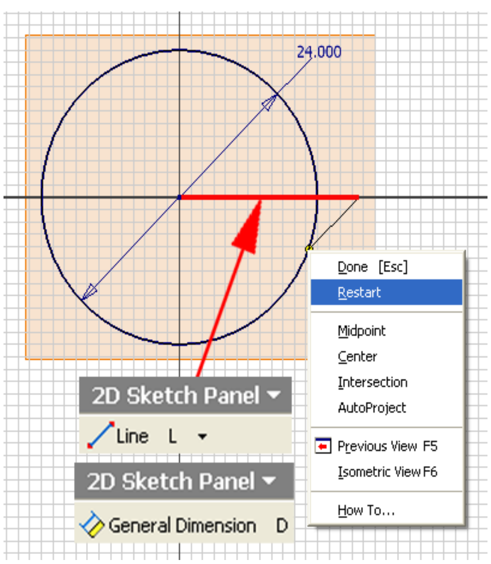


Figure 1G-3B: Creating a horizontal line starting at the projected Center Point and ending at a convenient location on the right of the circle; and selecting the Restart option to create a new line.

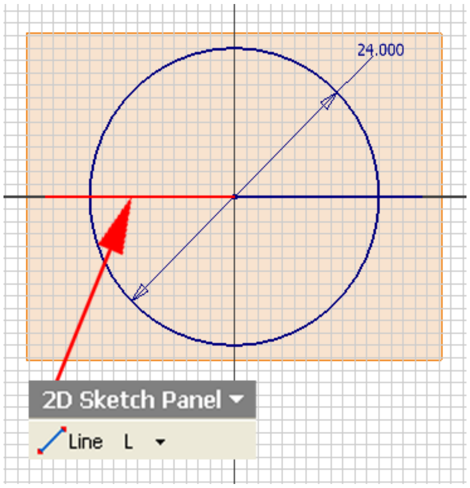


Figure 1G-3C: Creating the other horizontal line.

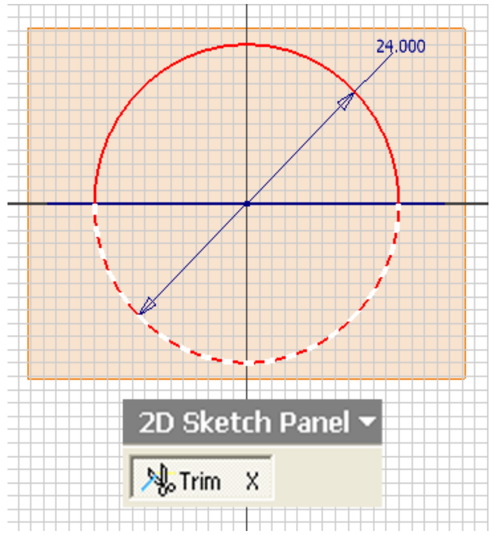


Figure 1G-3D: trimming off the lower half of the circle.

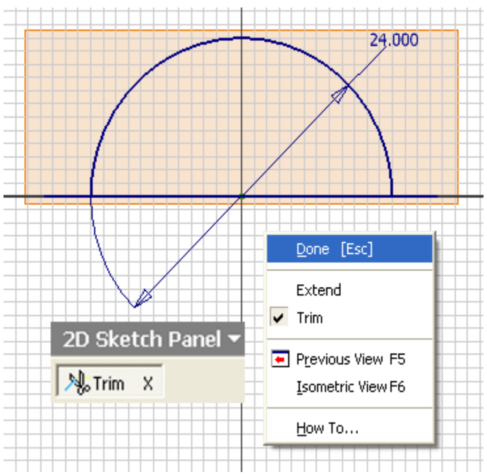


Figure 1G-3E: Exiting the tool.

Next, select the left horizontal line and press the **Delete** key on the keyboard to delete it (Figure 1G-3E). Next, select the **General Dimension** tool to apply a 0.066-inch (half of the thickness of the sheet-metal material plus a 0.001-inch gap allowance, or 0.120 ÷ 2 + 0.001 = 0.066) linear dimension to the short line (*Figure IH-3F*).

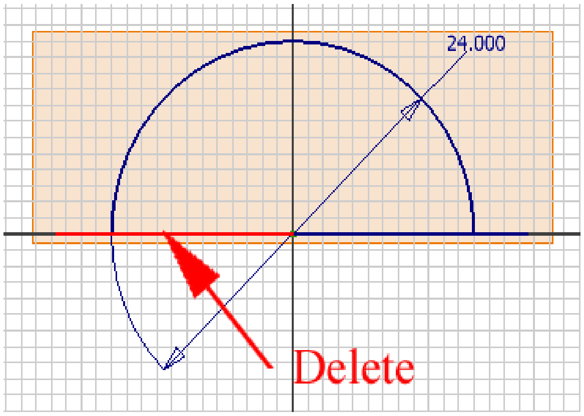


Figure 1G-3E: Selecting and deleting the left horizontal line.

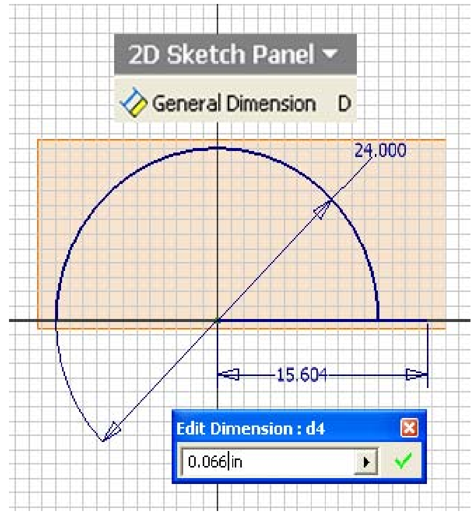


Figure IG-3F: Applying a 0.065-inch linear dimension to the short line.

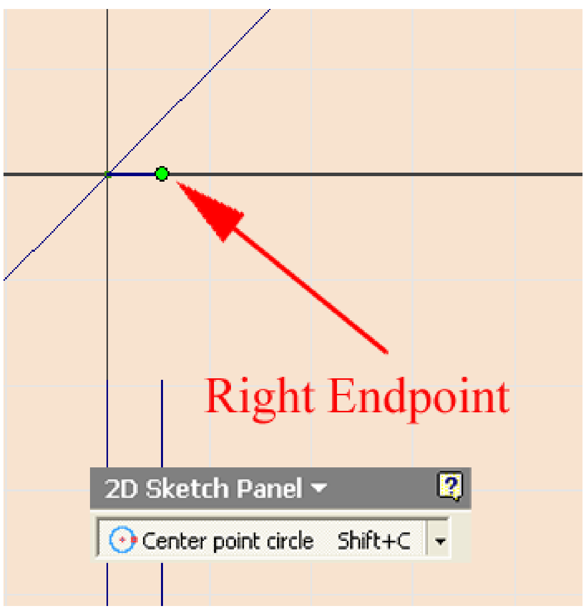


Figure 1G-3G: Clicking once at the right endpoint of the short line (point A) segment to establish the center of the circle.

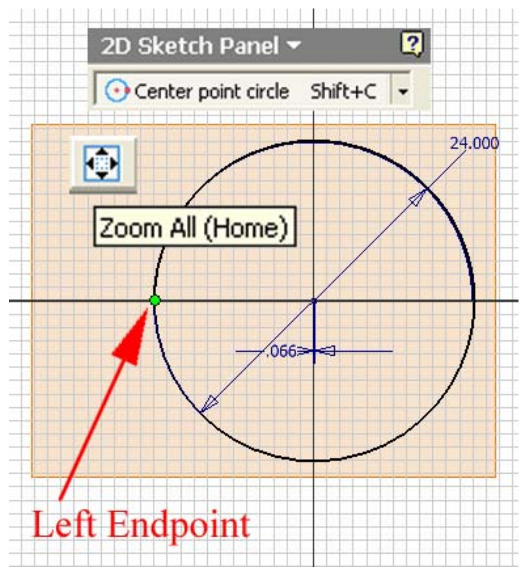
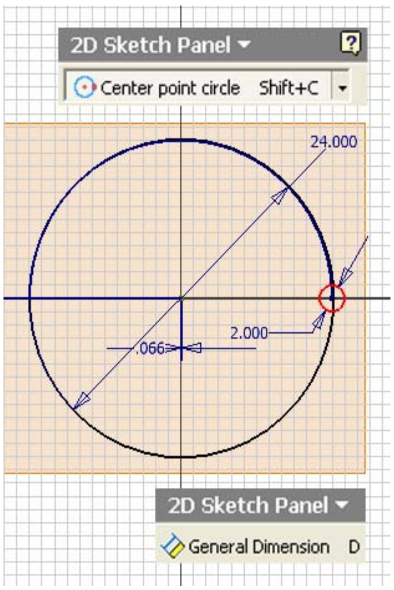


Figure 1G-3H: Clicking the Zoom All button to view the entire circle and clicking the left endpoint of the circular arc to establish the radius of the "overlapping circle."

Next, use the **Zoom Window** tool to zoom in the area enclosing the 0.066-inch short line segment; select the **Center Point Circle** tool; click once at the right endpoint of the short line segment to establish the center of the circle, at the appearance of the green dot endpoint snap indicator (*Figure 1G-3G*); click the **Zoom All** tool button on the **Inventor Standard** toolbar so as to view the entire circle (*Figure 1G-3H*), and click once at the left endpoint of the circular arc to establish the radius of the circle (*Figure 1G-3H*). Next, select the **Center Point Circle** tool and create a circle to establish the width of the overlapping seam; select the **General Dimension** tool to apply a 2.00-inch diameter dimension (corresponding to approximately 1.00-inch of the intended width for the overlapping seam (*Figure 1G-3J*). Next, select the **Trim** tool to trim off the upper half of the “overlapping circle” (*Figure 1G-3K*).



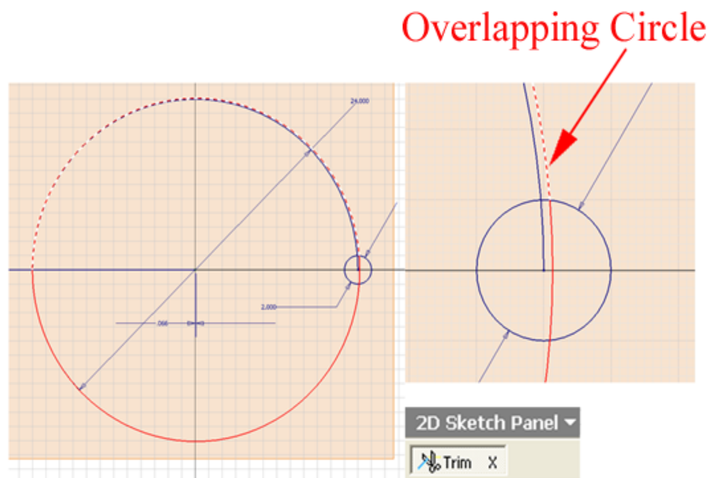


Figure 1G-3K (above): Trimming off the upper half of the “overlapping circle.”

Figure 1G-3J (left): Creating the “seam width” construction circle.

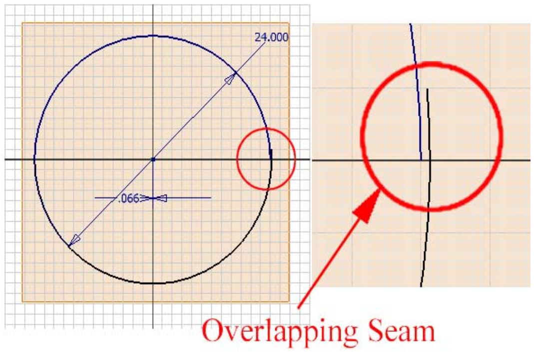


Figure 1G-3L: The profile of the cylindrical sheet-metal piece with overlapping seam.

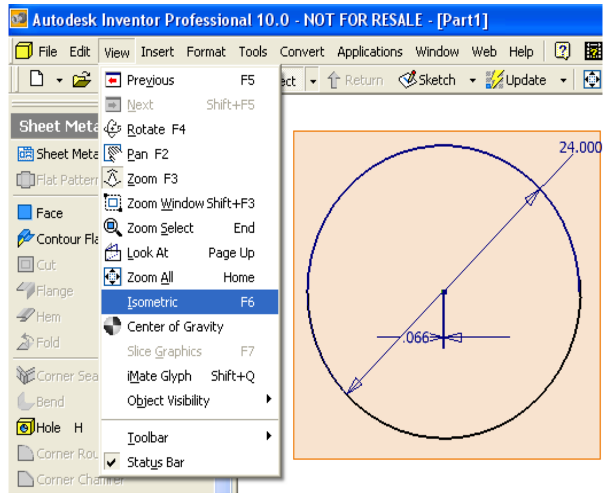


Figure 1G-3M: Switching to an isometric view with the View→Isometric menu.

The profile of the cylindrical sheet-metal piece with overlapping seam is complete. Click the **Return** button on the **Inventor Standard** tool bar to exit the **Sketch** mode. Next, go to the **View→Isometric** menu to switch to an isometric view (*Figure 1G-3M*). In the **Model** panel, select the **Sketch1** feature, highlight the name and retype *Base Profile* to rename it. Now, start saving the file. Press **Ctrl** and **s** keys simultaneously to save the file as *Tut- Cylinder with Overlapping Seam.ipt* with **Part File (*ipt)** for **Save As Type** field, in a convenient directory location, in a new folder to be created and named *Tut-Cylinder with Overlapping Seam* in the **Save As** window in a convenient directory location (*Figure 1G-3N* through *Figure 1G-3Q*). Save often, at least at the end of each step.

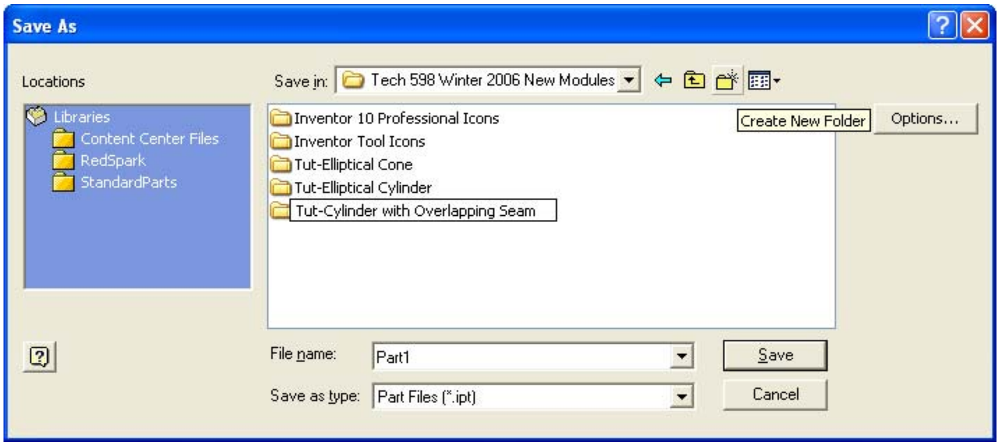


Figure 1G-3N: Creating the Tut-Cylinder with Overlap-ping Seam folder.

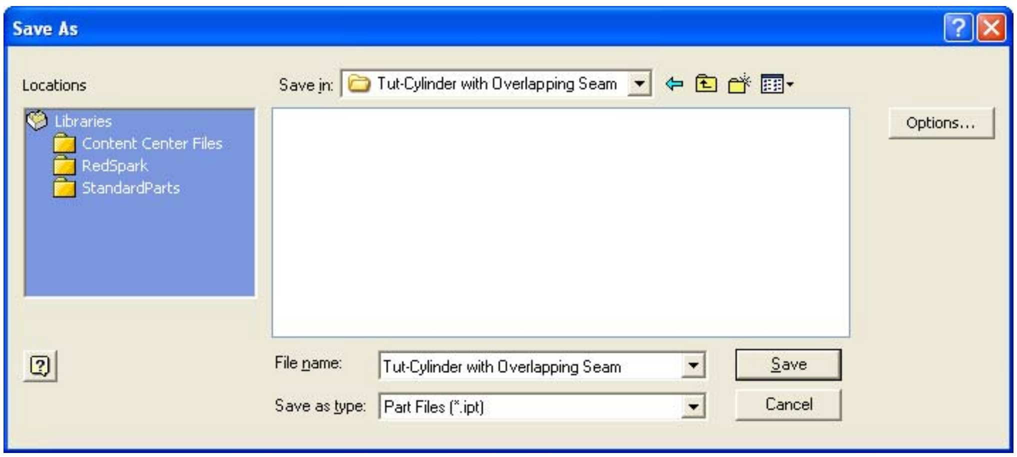


Figure 1G-3P: Saving the file asTut- Cylinder with Overlapping Seam.ipt inside the Tut-Cylinder with Overlap-ping Seam folder.

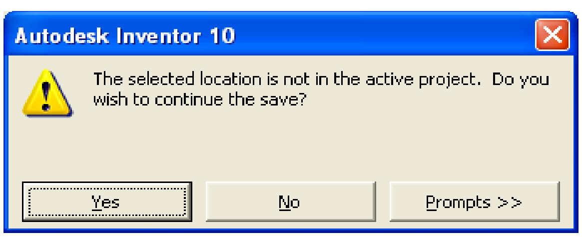


Figure 1G-3Q: Clicking the OK button to dismiss the warning message if it appears and to save the file.

Step 2: Create the circular cylindrical sheet-metal lateral piece with an overlapping seam

Next, switch to the **Sheet metal Features** tool panel and select the **Contour Flange** tool; click the **Profile** button in the tool's dialog window and click-select the open circular profile on the screen; the green geometry outlines of the **Contour Flange** appears on the screen (*Figure 1G-4A*); use the **Zoom Window** tool to zoom in the area with the overlapping seam as shown in *Figure 1G-4B*; click the **Offset** button in the tool's dialog window so that the **Contour Flange**'s points inward (*Figure 1G-4B*); click the **OK** button to create the **Contour Flange** feature, and rename it *Cylinder* in the **Model** panel. Select the **XZ Plane** on the screen and right-click for the shortcut menu to uncheck the **Visibility** option; the **XZ Plane** disappears from view (*Figure 1G-4C*). Next, select the **Rotate** tool and rotate the model to a convenient location where the overlapping seam can be easily seen (*Figure 1G-4D*).

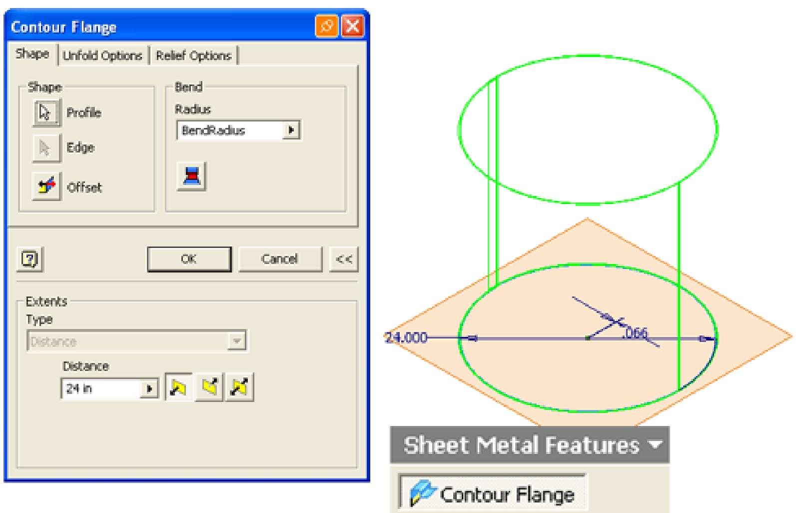


Figure 1G-4A: Click-selecting the open circular profile on the screen.

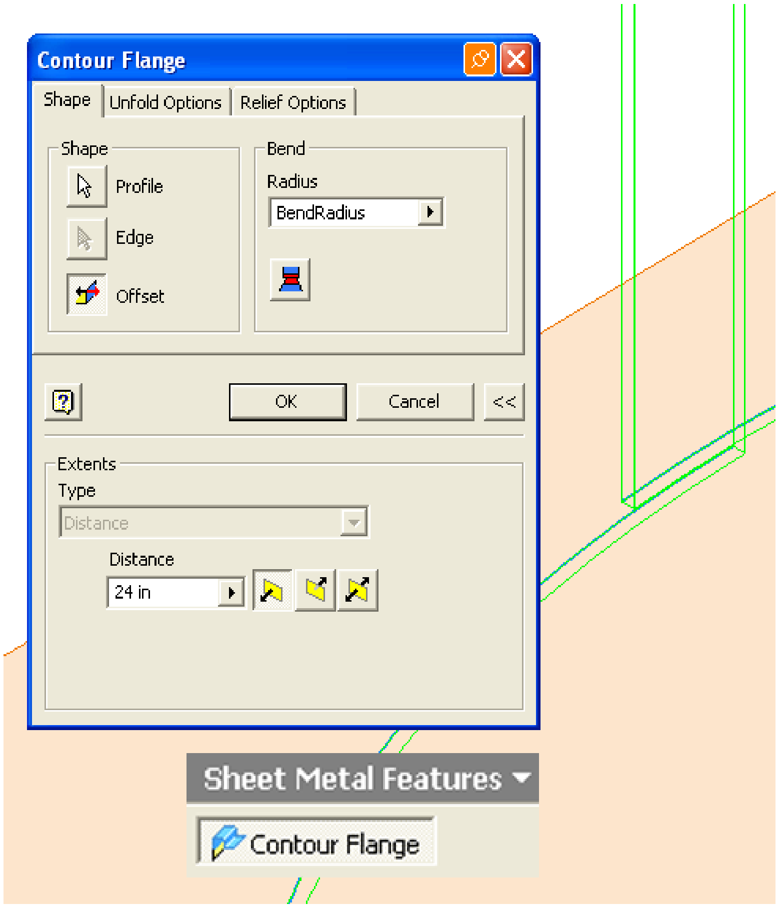


Figure 1G-4B: Clicking the Offset button in the tool's dialog window so that the Contour Flange's points inward.

Next, select the **Corner Chamfer** tool; click the **Corner** button in the tool's dialog window and then the front and rear end corners of the overlapping seam; red geometry outlines for the **Corner Chamfer** features appears on the screen (*Figure 1G-5A*); click the **OK** button to create the **Corner Chamfer** features (*Figure 1G-5B*), and rename them *Seam Corner Chamfers* in the **Model** panel.

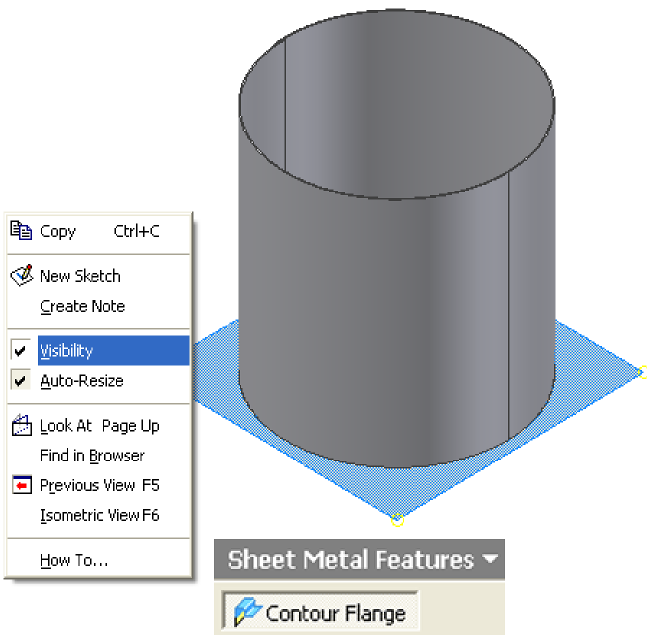


Figure 1G-4C: Un-checking the Visibility of the XZ Plane.

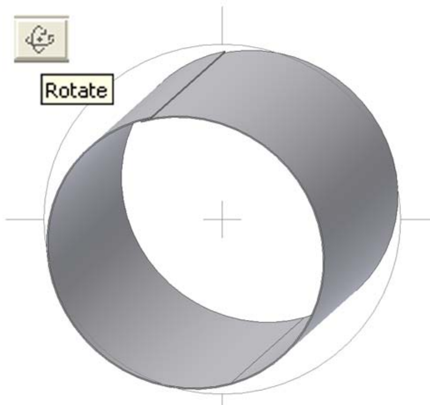


Figure 1G-4D: Rotating the model.

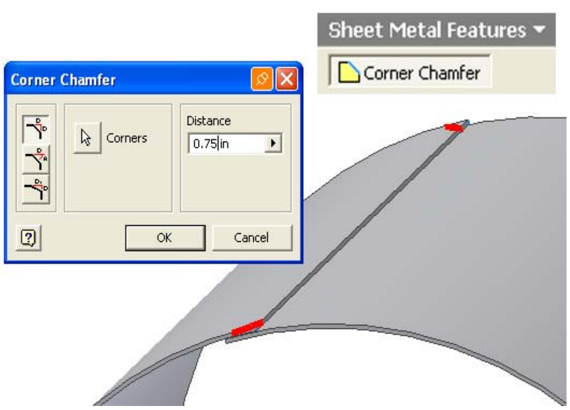


Figure 1G-5A: Adding Corner Chamfers.

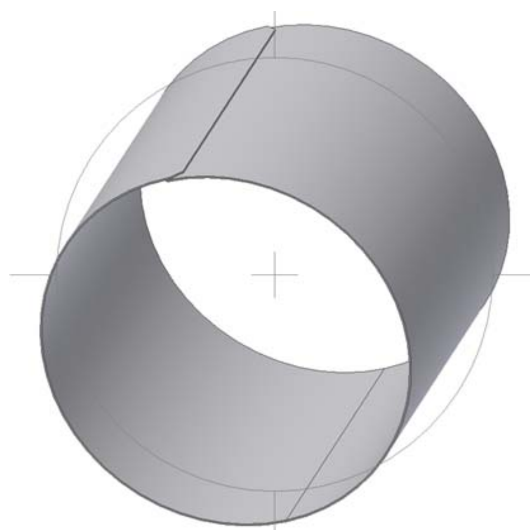


Figure 1G-5B: The Corner Chamfers.

Next, go to the **View→Isometric** pull-down menu to switch to an isometric view (*Figure 1G-5C*); next, click-select the outer surface of the folded-up model and click the **Flat Pattern** tool button in the **Sheet Metal Features** to create the **Flat Pattern** view (*Figure 1G-D*). The **Flat Pattern** view window opens; close the **Flat Pattern** window so as to return to the 3D model screen by clicking the lower X box at the upper-right corner of the screen (*Figure 1G-E*).

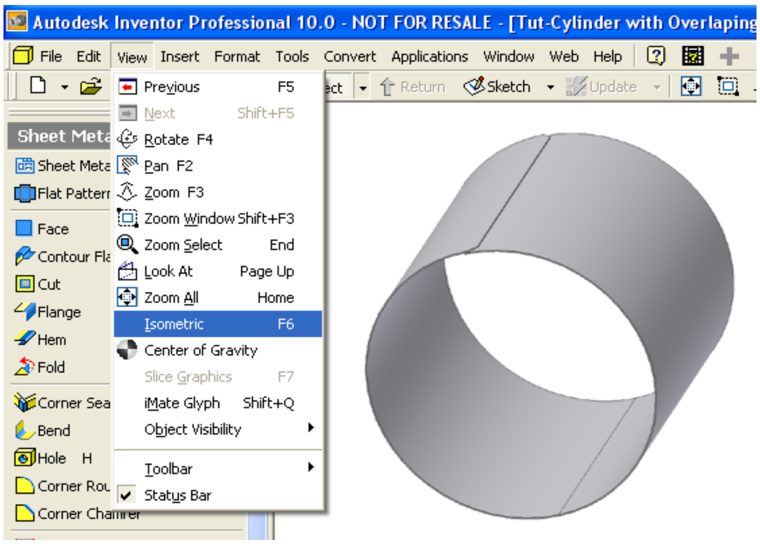


Figure 1G-5C: Switching to an isometric view.

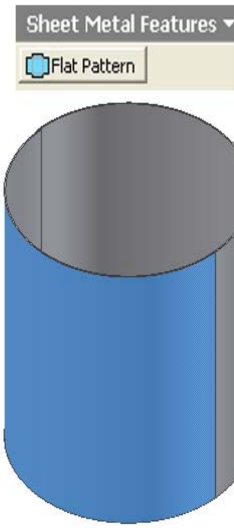


Figure 1G-D: Creating the Flat Pattern view.

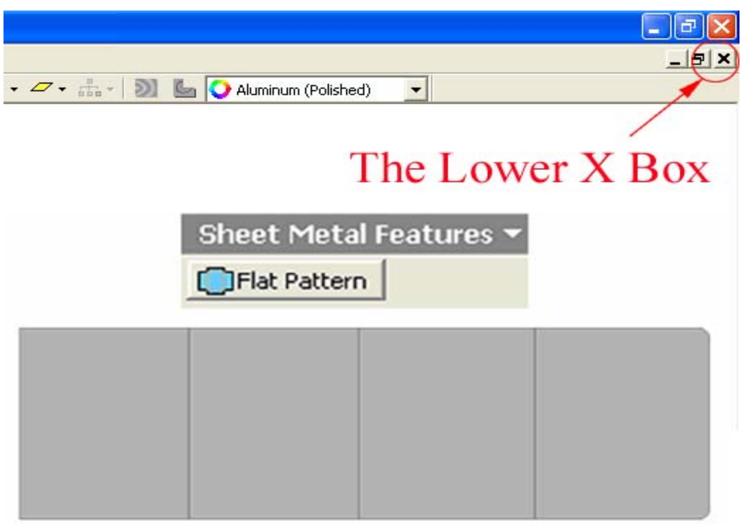


Figure 1G-4E: The Flat Pattern view window.

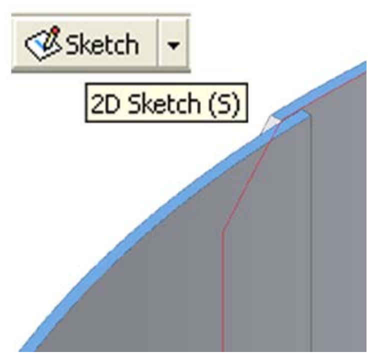


Figure 1G-5A: Selecting the top edge surface to start a new sketch (the Dove-tail Seams Profile).

Step 3: Creating the dove-tail seams on the top edge of the cylindrical sheet-metal lateral piece with a "work-around" method

We will now create dove-tail seams on the top edge of the cylindrical sheet-metal lateral piece with a "work-around" method. Since **Inventor** cannot create **Flange** features (the seams with the appropriate **Bend** features) on a curved surface (the surface of the lateral piece of the cylindrical sheet-metal part, we will use the **Extrude** tool to create "standing-up" seams without the appropriate **Bend** features, as a "work-around" method. Although the dive-tail seams so created are not bent as they should be, bends can be made in the fabrication process.

First, select the **Zoom Window** tool to zoom in any convenient portion of the top surface of the lateral piece so that the surface can be click-selected; click-select the top surface and then the **Sketch** tool button to start a new **Sketch** feature, as shown in *Figure 1G-5A*, and rename it *Dove Tail Seam Profile* in the **Model** panel; next, click the **Look At** button on the **Inventor Standard** tool bar and then the *Dove Tail Seam Profile* **Sketch** feature to switch to an orthographic normal view (*Figure 1G-5B*). Notice that the edges of the top edge surface are automatically projected onto the new **Sketch** feature when started, by Inventor's default settings; window-select the projected outlines, and press the **Delete** key on the keyboard to delete them. Next, select the **Project Geometry** tool and select the **Center Point** feature from the **Model** panel to project it onto the *Dove Tail Seam Profile* **Sketch** feature. Next, select the **Line** tool and create a vertical line and an inclined line, both starting from the projected **Center Point** (*Figure 1G-5C*).

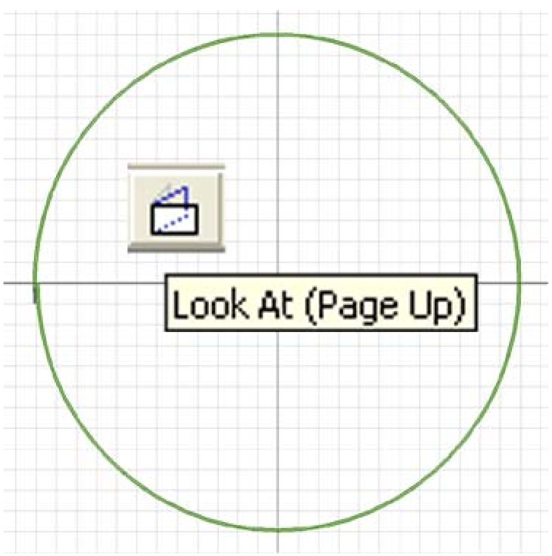


Figure 1G-5B: Switching to an orthographic normal view.

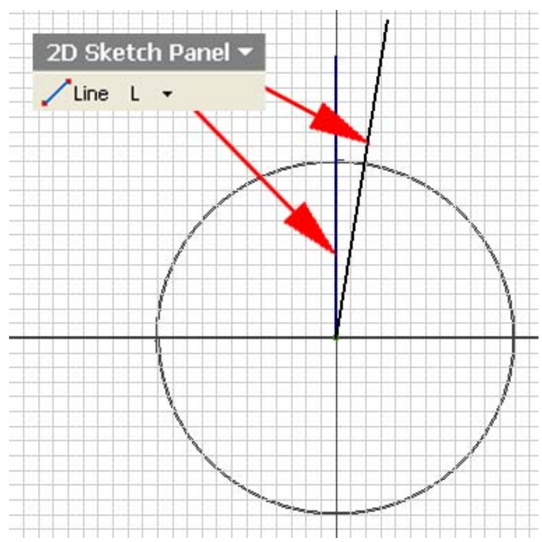


Figure 1G-5C: Creating a vertical line and an inclined line.

Next, graphically determine the approximate inner angle of a slice of the circle enclosed by the outer circular edge of the cylindrical sheet-metal part, for the cord length (the dove-tail width) to be approximately 1.00 inch. Select the **Center Point Circle** tool to create a circle centered at the approximate point of intersection between the vertical line and the outer circular edge of the cylindrical sheet-metal part, and use the **General Dimension** tool to apply a 2.00-inch diameter dimension to the circle (the “dove-tail seam width construction circle,” corresponding to twice the intended 1.00-inch width for the dove-tail seams; *Figure 1G-5D*); next, applying a angular dimension to the vertical and inclined lines; the default measured value indicate that the angle is 9.08° and its two points of intersection with the outer edge of the cylindrical sheet-metal part forms a cord with a length twice as large as the radius of the “dove-tail seam width construction circle;” therefore, type 5 deg in the **General Dimension** tool’s text field and click the green checkmark (*Figure 1G-5E*); the inclined line now appears to be tangent to the “dove-tail seam width construction circle” (*Figure 1G-5F*). Therefore, the inner angle is approximately 5°. The alternative mathematical method can be used to determine the inner angle of the slice of the circle enclosed by the outer circular edge of the cylindrical sheet-metal part, for the dove-tail width to be an intended value:

$$Circumferance_{Circle} = Diameter_{Circle} \times \pi = 24.0 \times 3.14 \approx 75.4in$$

$$Number_{Slice} = \frac{Circumferance_{Circle}}{Width_{\substack{Dove-tail \\ Seam}}} = \frac{75.4in}{1.00in} \approx 75$$

$$Angle_{inner} = \frac{Angle_{Circle}}{Number_{Slice}} = \frac{360°}{75} = 4.8° \approx 5.0°$$

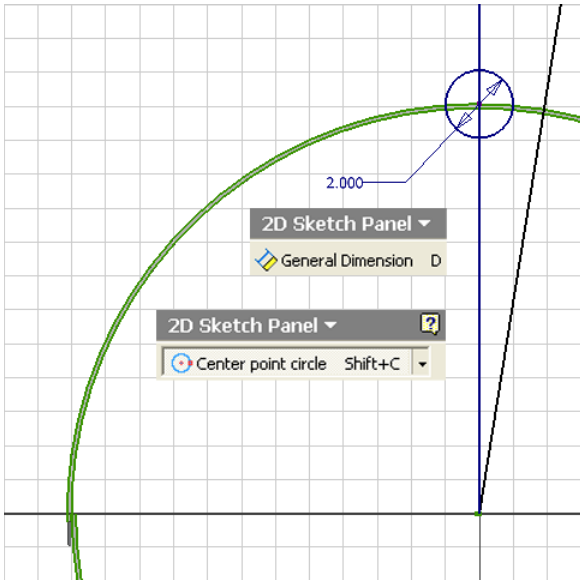


Figure 1G-5D: Creating the 2.00-inch diameter "dove-tail seam width construction circle" centered at the approximate point of intersection between the vertical line and the outer circular edge of the cylindrical sheet-metal part.

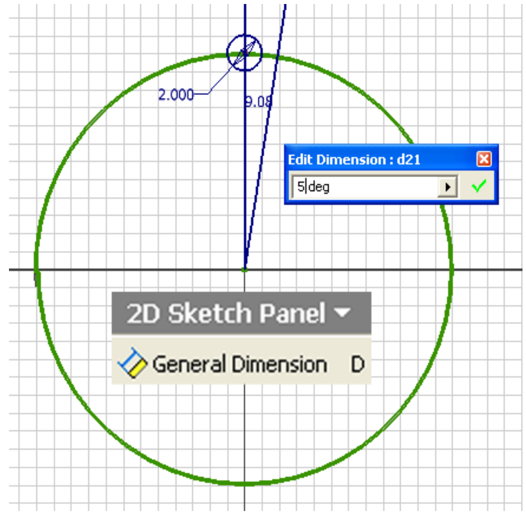


Figure 1G-5E: Applying an angular dimension.

Next, select the **Project Geometry** tool and click-select the inner and outer edges of the sheet-metal part to project them onto the sketch. Next, select the inclined line and the "dove-tail seam width construction circle" and press the **Delete** key on the keyboard to delete them (*Figure 1G-5F*). Next, select the **Trim** tool to trim off the excessive segment of the vertical line (*Figure 1G-5G*). Next, select the **Circular Pattern** tool; click the **Geometry** button in the tool's dialog window, and click-select the vertical line (*Figure 1G-5H*); click the arrow button on the left of the **Axis** button in the tool's dialog window, and click-select the center point of the circle (lower endpoint of the vertical line) (*Figure 1G-5J*); the **Circular Pattern** geometry appears on the screen (*Figure 1G-5K*); type 360/5 for number of Instances of the Circular Pattern in the tool's dialog window (*Figure 1G-5L*); click the **OK** button to create the Circular Pattern 2D sketch feature (*Figure 1G-5M*).

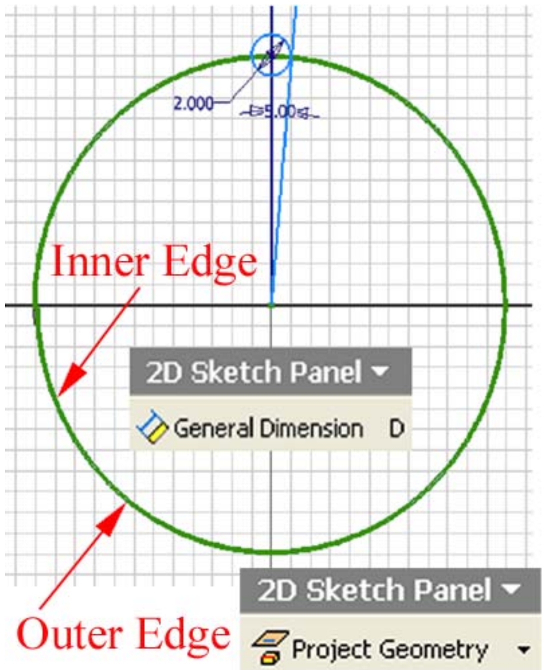


Figure 1G-5F: The inclined line appearing to be tangent to the "dove-tail seam width construction circle;" selecting and deleting the inclined line and the "dove-tail seam width construction circle;" and projecting the inner and outer edges of the sheet-metal part onto the sketch.

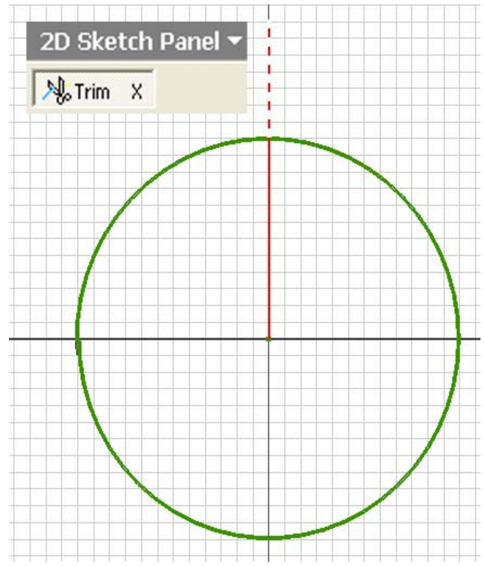


Figure 1G-5G: Trimming the excessive segment of the vertical line.

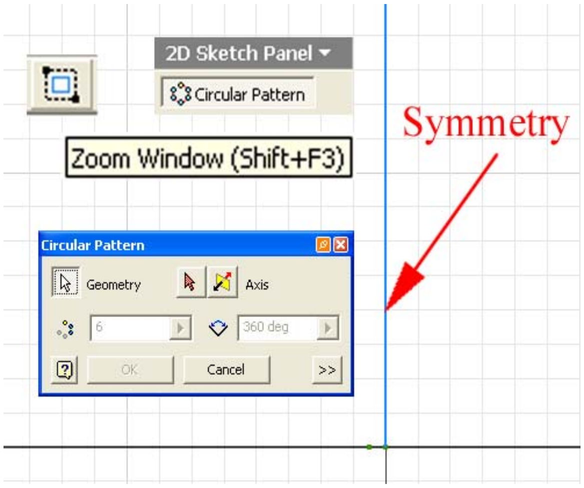


Figure 1G-5H: Selecting the vertical line for Geometry.

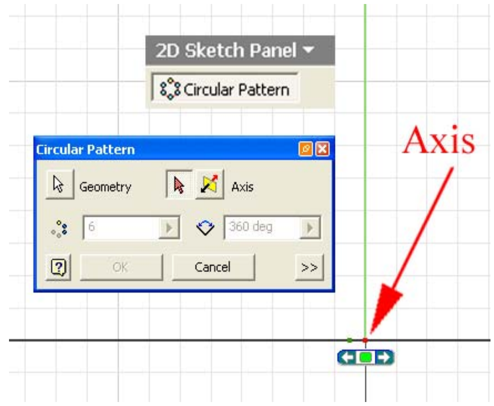


Figure 1G-5J: Selecting the center point of the circle (lower endpoint of the vertical line) for Axis.

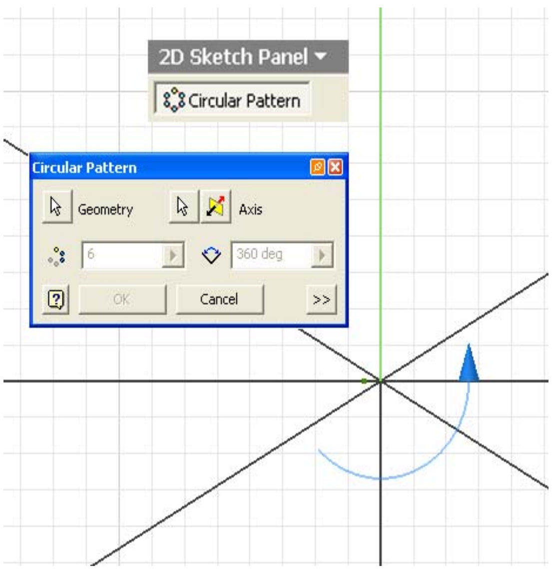


Figure 1G-5K: Circular Pattern geometry appearing on the screen.

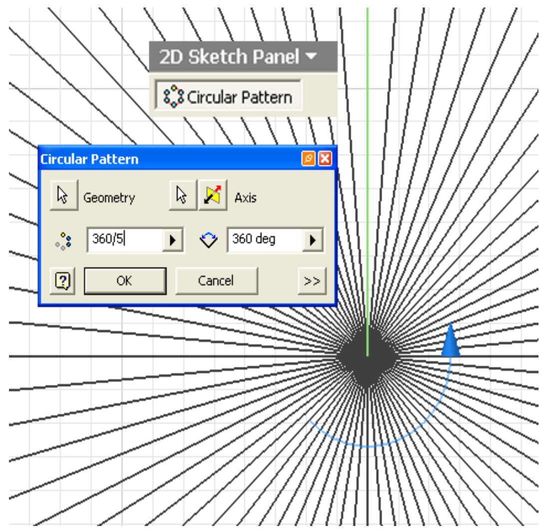


Figure 1G-5L: Typing 360/5 for number of Instances of the Circular Pattern.

Next, use the **Zoom Window** tool to zoom in a reasonably sized area of the top edge surface; select the **Trim** tool to trim off pairs of arc segments for the gaps between the dove-tail seams (*Figure 1G-5N*); hold the middle button of the mouse and drag to pan to next areas when needed; when finished, right-click for the shortcut menu and choose the **Done** option to exit the tool (*Figure 1G-5P*).

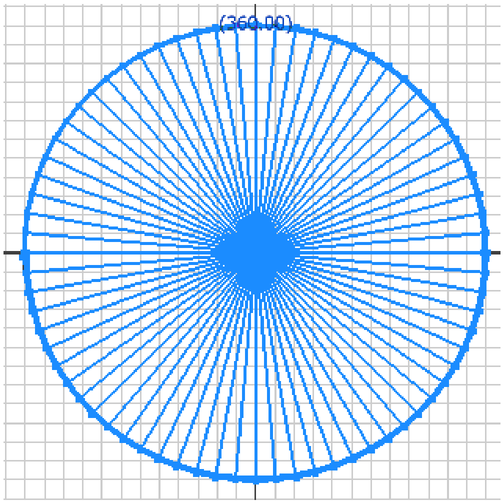


Figure 1G-5M: The Circular Pattern 2D sketch feature.

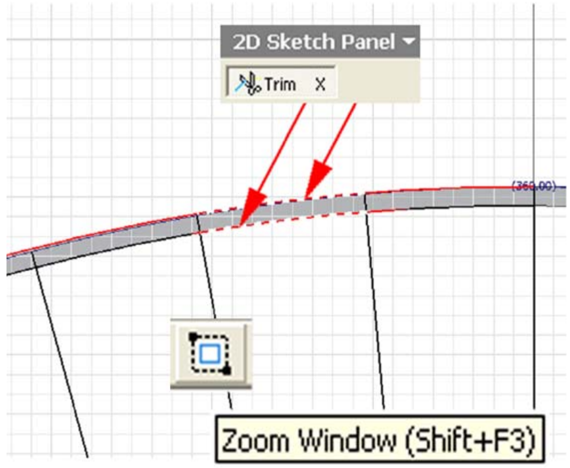


Figure 1G-5N: Trimming off pairs of arc segments for the gaps between the dove-tail seams.

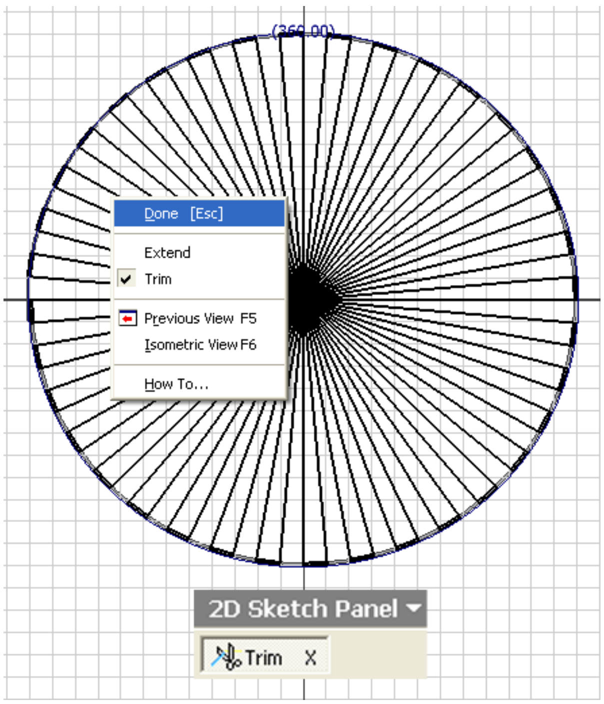


Figure 1G-5P: Right-clicking for the shortcut menu and choose the Done option to exit the Trim tool after completing all gaps between the dove-tail seams.

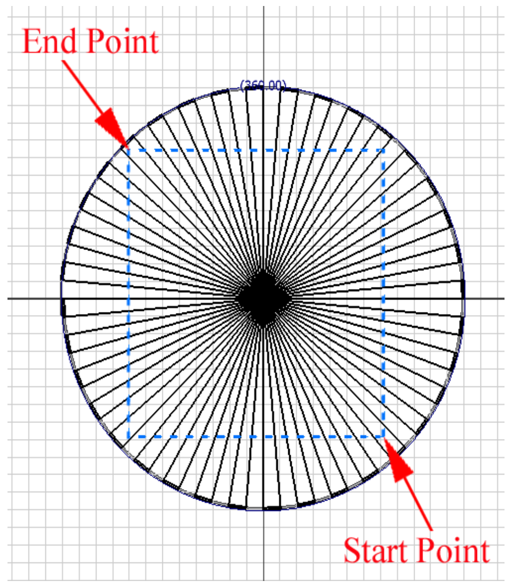


Figure 1G-5Q: Window-selecting all radiating Circular Pattern lines starting from a lower right corner and ending at an upper left corner.

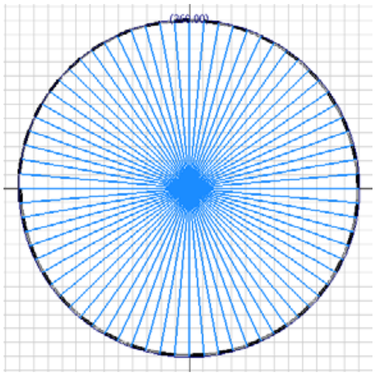
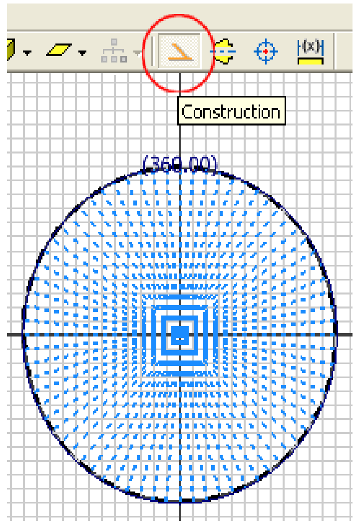


Figure 1G-5R: All radiating Circular Pattern lines selected.



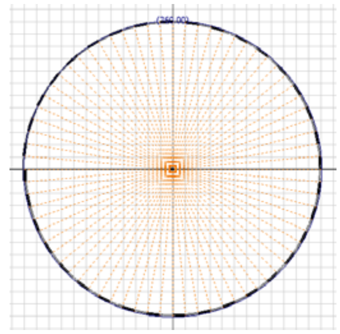


Figure 1G-5S: Changing the line Style of radiating Circular Pattern lines to Construction.

Next, window-selecting all radiating Circular Pattern lines starting from a lower right corner and ending at an upper left corner (*Figure 1G-5Q* and *Figure 1G-5R*); click the **Construction** button on the **Inventor Standard** tool bar to change their line **Style** to **Construction** (*Figure 1G-5S*). Next, select the **Line** tool to add short edges to the dove-tail seams' closed profile, click at the appearance of green dot endpoint snap indicators to make sure that the profiles are closed (*Figure 1G-5T*). Click the **Return** button to exit the sketch. Go to the **View→Isometric** menu to switch to an isometric view.

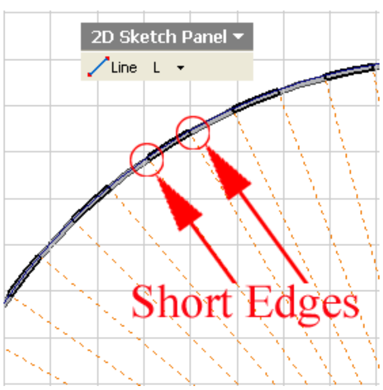


Figure 1G-5T: Using the Line tool to add short edges to the dove-tail seams' closed profiles.

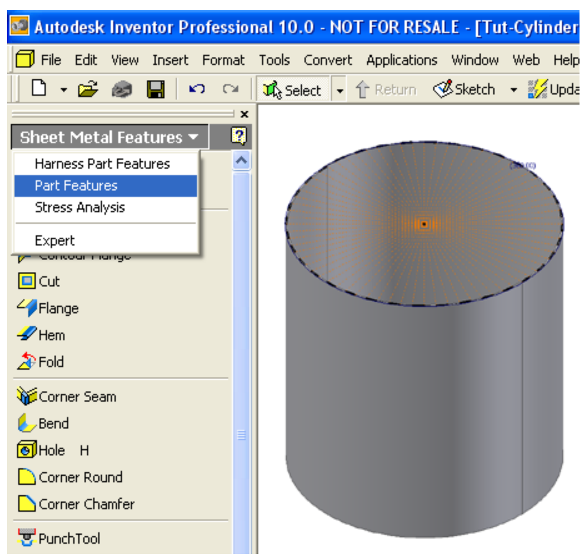


Figure 1G-6A: Switching to an isometric view, and to the Part Features tool panel.

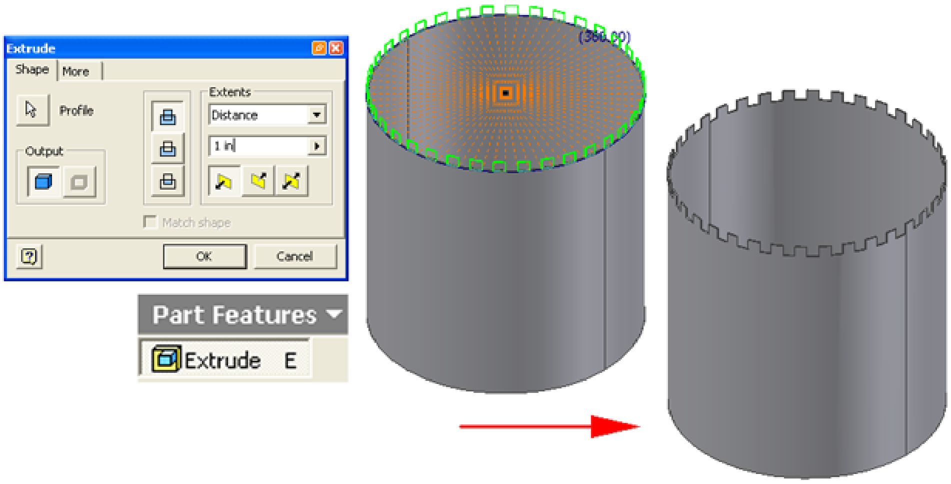


Figure 1G-6B: Creating the dove-tail seams.

Next, switch to the **Part Features** tool panel from the **Sheet Metal Features** tool panel by clicking the downwardly-pointing triangle next to the name and choosing the **Part Features** option from the pull-down menu **(***Figure 1G-6A*). Next, select the **Extrude** tool; click the **Profile** arrow button in the tool's dialog window and then click-select the closed profiles for dove-tail seams, with the help of the **Zoom Window** tool if needed; select **Distance** for **Extents** and type *1 in* (inch) in the text field, click the leftmost **Direction** button so that the dove-tail seams; green geometry outlines point upward; click the **OK** button to create the dove-tail seams (*Figure 1G-6B*); and rename the **Extrude** feature *Dove Tail Seams* in the **Model** panel.

Next, switch to the **Sheet Metal Features** tool panel; click-select the frontal outer surface of the lateral piece and click the **Flat Pattern** tool button to create the **Flat Pattern** view (*Figure 1G-6C*). If desirable, apple a colorful material rendering by selecting a material type from the pull-down menu on the **Inventor Standard** tool bar in the **Flat Pattern** window; the folded-up 3D model window automatically updates **(***Figure 1G-6D***).** Save the file. The features of the sheet-metal part are listed in the **Model** panel (*Figure 1G-6F*).

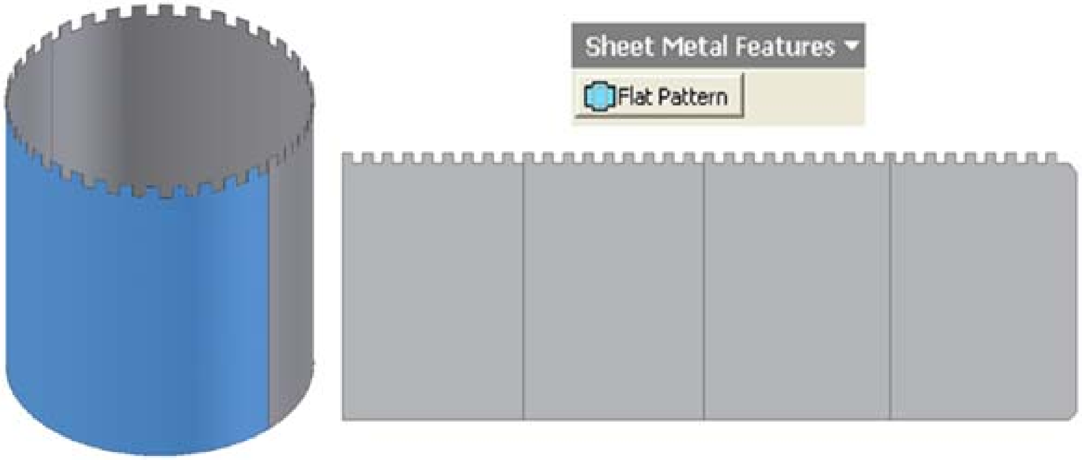


Figure 1G-6C: Creating the Flat Pattern view.

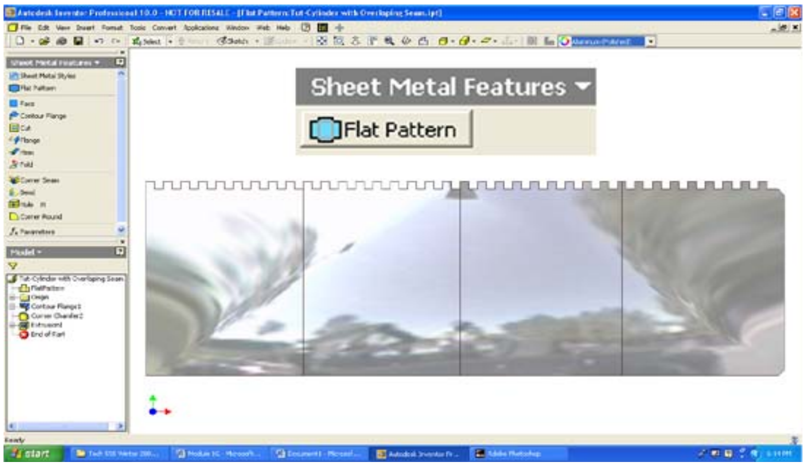


Figure 1G-6D: Applying an Aluminum (Polished) material rendering in the Flat Pattern window.

Figure 1G-6E: The folde-up3D model window automatically updating to the assigned material rendering.

Congratulations! In this Module, you have learned how to create a cylindrical sheet-metal part with an overlapping seam on its lateral edge and dove-tail seams on its top edge, and its **Flat Pattern** view.

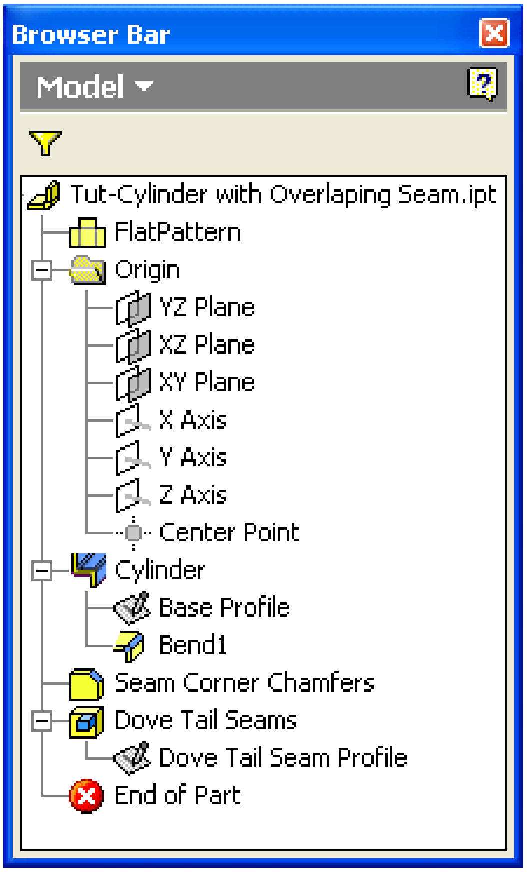


Figure 1G-6F: The Model panel feature list.