Module 1E:
Parallel-Line Flat Pattern Development of Sheet-Metal Folded Model Wrapping the 3D Space of An Oblique Circular Cylinder

In this *Module*, we will explore the topic of oblique cylinder by creating sheet metal parts in Autodesk Inventor wrapping the 3D space of an oblique cylinder with a circular cross section. An oblique cylinder is one with the central axis as well as all element lines being not straight up but forming an angle $0°<\theta<90°$ with the horizontal ground. There may be two cases:

- An oblique cylinder with a circular cross-section; in this case, the development of 2D **Flat Pattern** can be obtained through parallel-line development method (*Figure1E-1A*);

- An oblique cylinder with a circular base; in this case, the development of 2D **Flat Pattern** can be obtained through triangulation development method.

In this *Module*, we will study the methods of creating sheet metal parts in Autodesk Inventor wrapping the 3D space of an oblique cylinder with a circular cross section. There are two basic methods for creating the lateral piece:

- Creating the lateral piece of the oblique cylinder with a circular cross-section with the **Revolve** tool; apply a horizontal cut the both the top and bottom portion of the lateral piece;

- Creating the lateral piece of the oblique cylinder with a circular cross-section with the **Contour Flange** tool; apply a horizontal cut the both the top and bottom portion of the lateral piece.

In this *Module*, the **Revolve** method will be explored.

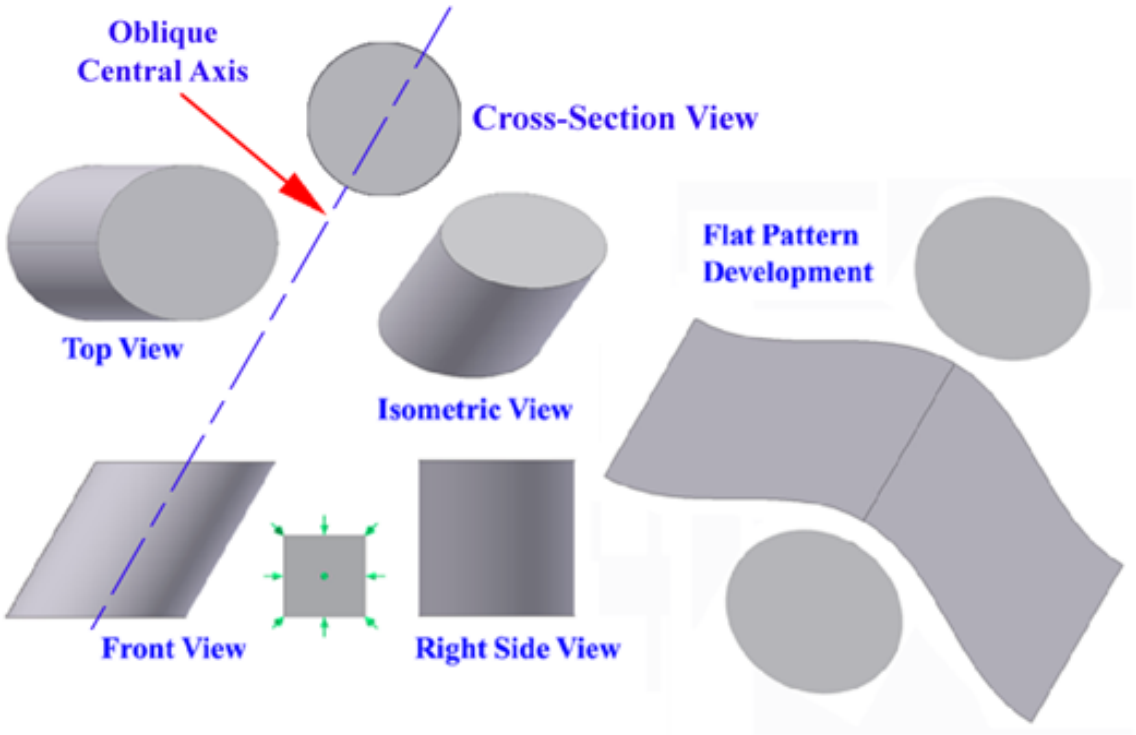


Figure1E-1A: The orthographic Views, isometric view and flat pattern of an oblique cylinder with a circular cross-section.

Step 1: Creating the lateral cylindrical piece with the Revolve tool

Launch Inventor, start a new **Sheet Metal (in).ipt** part file under the **English** tab (*Figure1E-1B*). An Inventor sheet-metal file opens. "*Sketch1*" is created by default in the **Model** panel on the **XY Plane**; rename the new sketch as *Oblique Cylinder Profile* in the **Model** panel; click the **Look At** tool button on the **Command Bar** and then the *Oblique Cylinder Profile* in the **Model** panel to switch to an orthographic normal view; click-select the **Project Geometry** tool button from the **Sketch** tool panel, and then the **Center Point** feature under the **Origin** folder in the **Model** panel; the **Center Point** is projected onto the sketch to provide a snap point; next, select the **Line** tool, move the mouse closer to the projected **Center Point**, and click once when the green dot snap point indicator appears, then move the mouse rightward horizontally (with the help of the **Horizontal** short bar indicator) and click one at any convenient location on the screen; right-click for the **Line** tool's shortcut menu and choose the **Restart** option to draw another line; move the mouse upward vertically (with the help of the **Perpendicular** ⊥ indicator) and click one at any convenient location on the screen; next, select the **General Dimension** tool and click the vertical line to apply a 36 in (inch) height dimension; next select the **Line** tool again, move the mouse cursor closer to the projected **Center Point** again and click

once when the green dot snap point indicator appears, then move the mouse upward and rightward at an angle and click one at a convenient location fairly above the height of the vertical line and click once to draw a slanted line (the "oblique central axis line"); right-click for the shortcut menu again and choose **Done** to exit the tool; next, click-select the "oblique central axis line" and go to the **Command Bar** to change its **Style** to **Centerline** from the drop-down menu; next, select the **General Dimension** again and click the horizontal line and the "oblique central axis line" to apply a 60 deg (60°) angular dimension between them (*Figure 1E-1C*); next, select the **Line** tool again, click once at a point on the left of and fairly above the "oblique central axis line," move the mouse cursor closer to the "oblique central axis line" to pick up a **Parallel** constraint indicator (which appears as //) and move the mouse back, leftward and downward to a location fairly below the "oblique central axis line," and with the help of the // **Parallel** constraint indicator, click once to create a line (the "cross-section outer wall line") parallel to the "oblique central axis line" (*Figure 1E-1D*) next, select the **General Dimension** tool, click the "cross-section inner wall line" and the "oblique central axis line" to apply a ∅36.0 diameter dimension (*Figure 1E-1E*); next, select the **Offset** tool, click the "cross-section outer wall line" once, move the mouse cursor rightward and click a second time to create an offset line (the "cross-section inner wall line," as shown in *Figure 1E-1F*); next, select the **General Dimension** tool, click the "cross-section inner wall line" and the "cross-section outer wall line" to apply an **Aligned Dimension** of 0.12 in (inch) for the thickness of the sheet-metal material (*Figure 1E-1G* and *Figure 1E-1H*); next, select the **Line** tool to add a short line on the top to close the cross-section and a short line on the base to close the cross-section, with the help of the **Zoom Window** tool and the green dot endpoint snap indicator (*Figure 1E0-1J* and *Figure 1E0-1K*); next, select the **General Dimension** tool to apply a 80.0-inch (or greater) aligned dimension to the "cross-section outer wall line" to ensure enough length, if necessary (*Figure 1E-1L*). The closed cross-section profile is completed; click the **Return** button on the **Command Bar** to exit the sketch. Save the file as *Tut-Oblique Cylinder.ipt* in side a folder to be created and name *Tut-Oblique Cylinder* in the **Save As** dialog window. Save often at the end of each feature's creation.

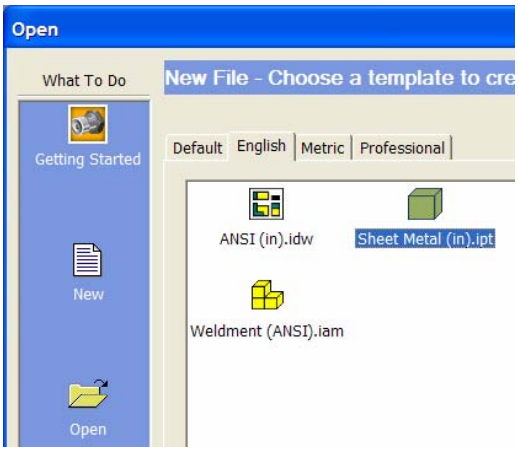


Figure 1E-1B: Starting a Sheet Metal (in) file in Autodesk Inventor.

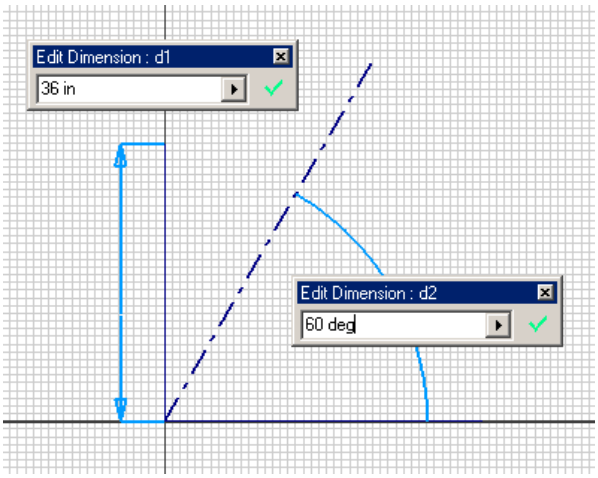


Figure 1E-1C: Projecting the Center Point, Creating the horizontal and vertical lines, applying a height dimension to the vertical line, creating the 60° "oblique central axis line" beyond the height of the vertical line.

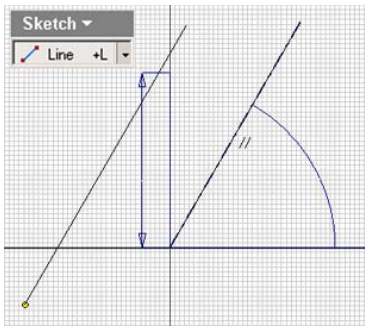


Figure 1E-1D: Creating the "cross-section outer wall line" parallel to the "oblique central axis line."

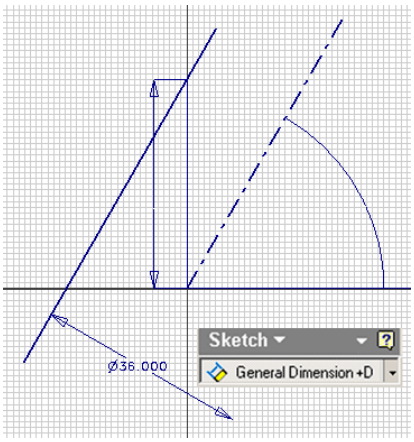


Figure 1E-1E: Applying a ∅36.0-inch diameter dimension.

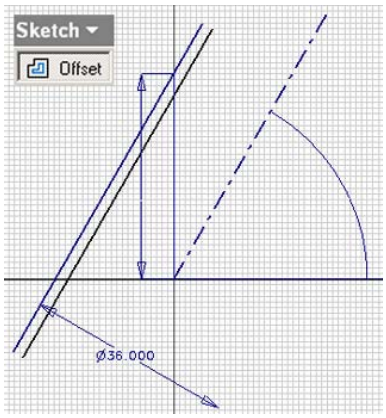


Figure 1E-1F: Creating the "cross-section inner wall line."

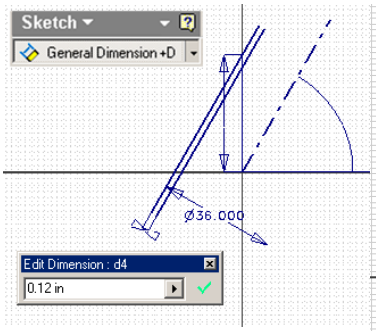


Figure 1E-1G: Applying a 0.12 in dimension for the thickness of sheet-metal material.

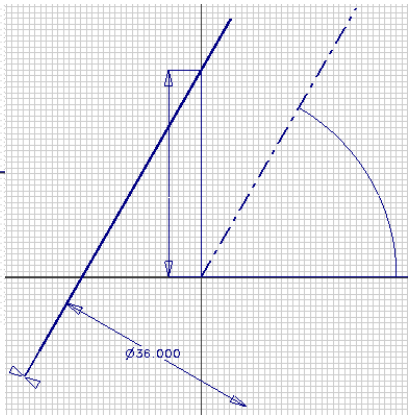


Figure 1E-1H: The thickness dimension applied.

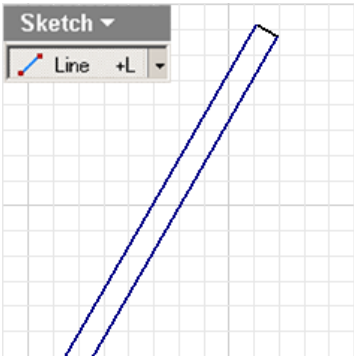


Figure 1E0-1J: Adding a short line on the top to close the cross-section.

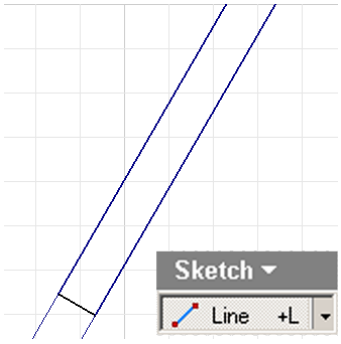


Figure 1E-1K: Adding a short line on the base to close the cross-section.

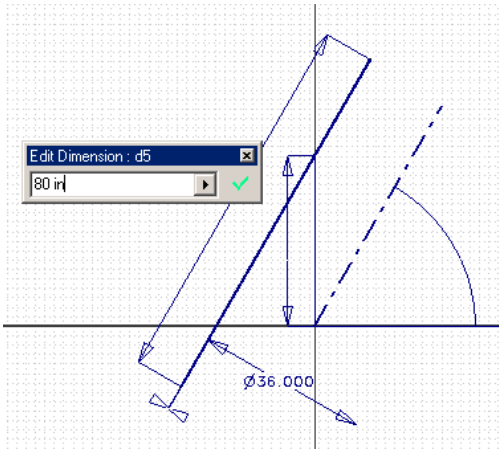


Figure 1E-1L: Applying a 80.0-inch aligned dimension to the "cross-section outer wall line" to ensure enough length.

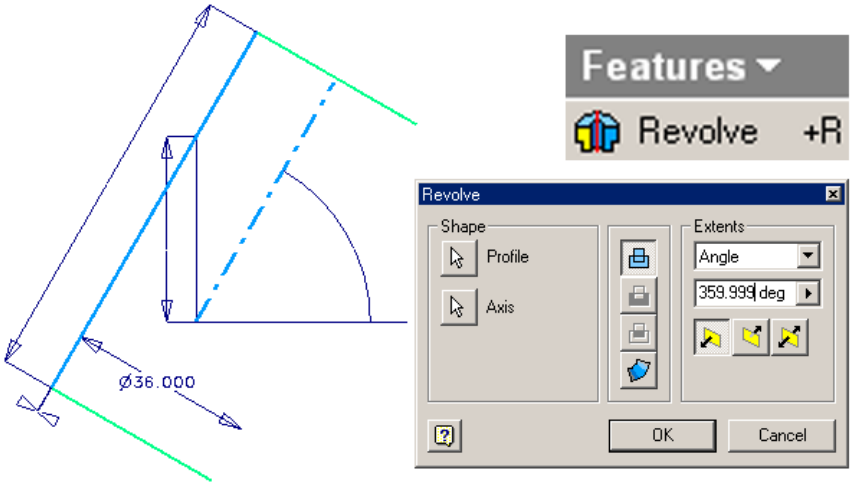


Figure 1E-1-1M: Revolving the cross-section profile by 359.999 degrees.

Figure 1E-1N: The basic lateral piece.

Next, select the **Revolve** tool, the closed cross-section profile and the "oblique central axis line" with **Centerline** as **Style** are automatically selected as **Profile** and **Axis**; in the tool's dialog window, select the **Angle** option from the drop-down menu and type 359.999 deg (359.999°) in the text field, in the **Extents** section (*Figure 1E-1-1M*); click the **OK** button to create the **Revolve** feature (*Figure 1E-1N*), and rename it *Oblique Cylinder* in the **Model** panel.

Step 2: cutting the top and base portion of the cylindrical piece

Select the **XY Plane** and click the **Look At** button and **Sketch** button on the **Command Bar** to switch to an orthographic normal view and start a new sketch (*Figure 1E-2A*); rename it *Truncating Profile* in the **Model** panel; select the **Project Geometry** tool and the **Center Point** from the **Model** panel to project it onto the sketch (*Figure 1E-2B*); select the **Line** tool to draw three horizontal lines: one starting and snapped to the projected **Center Point** and ending at its left side (the "ground edge line"), one starting and snapped to the projected **Center Point** and ending at its right side, and one above the first two lines (the "upper edge line" as shown in *Figure 1E-2C*), all with the help of the **Horizontal** constraint indicator (- sign); next, draw additional lines to complete two closed cutoff profiles that cover the upper and lower portions of the oblique cylindrical piece; next, select the **General Dimension** tool to apply a 36 in (inch) height dimension between the "ground edge line" and the "upper edge line" (*Figure 1E-2D*); the *Truncating Profile* is completed; click the **Return** button on the **Command Bar** to exit the sketch.

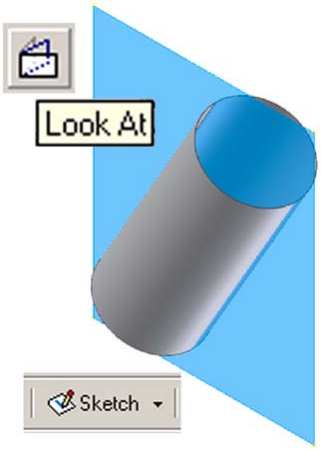


Figure 1E-2A: Selecting the XY Plane for Truncating Profile sketch.

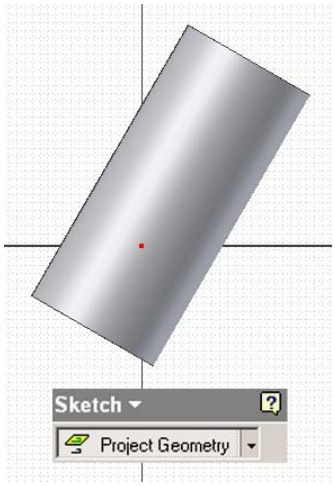


Figure 1E-2B: Projecting the Center Point.

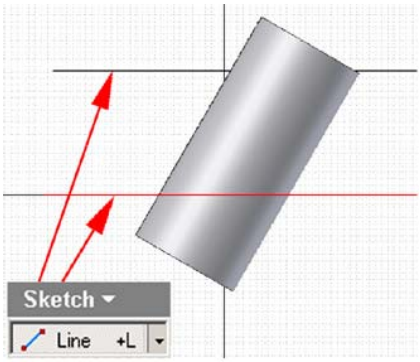


Figure 1E-2C: Drawing horizontal lines.

Next, select the **Extrude** tool; the two closed cutoff profiles are automatically selected; in the tool's dialog window, select the **Cut** option, **All** for **Extents**, and **Midplane** for direction; click the **OK** button to finish (*Figure 1E-2E*); the oblique cylindrical piece is completed (*Figure 1E-2F*); rename the **Extrude** feature *Truncating Cut* in the **Model** panel; next, click-select the surface of the oblique cylindrical piece, and select the **Flat Pattern** tool, the **Flat Pattern** window opens (*Figure 1E-2G*). All features of the part are listed tin the **Model** panel (*Figure 1E-2H*). Save and close the file.

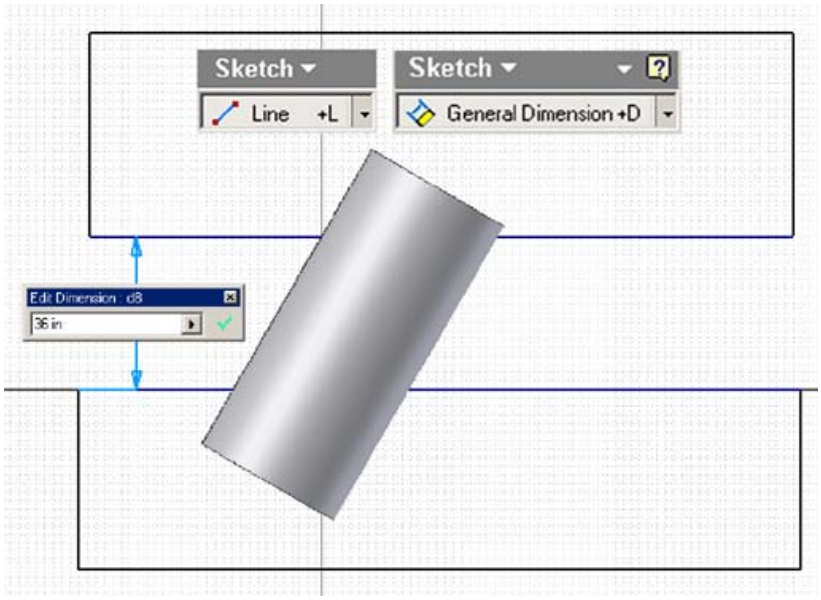
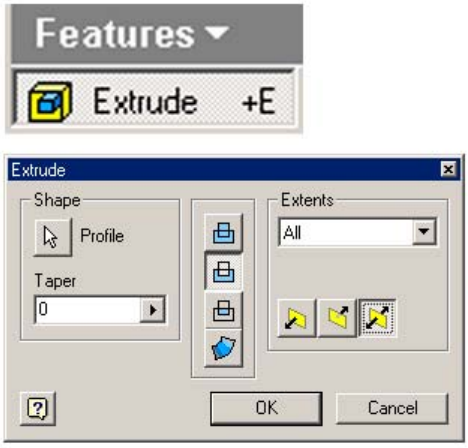


Figure 1E-2D: Completing the Truncating Profile sketch.



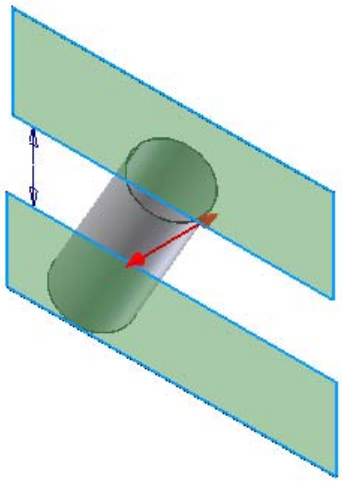


Figure 1E-2E: Using the Extrude tool with Cut option to truncate the oblique cylindrical piece.

Figure 1E-2F: The completed oblique cylindrical piece.

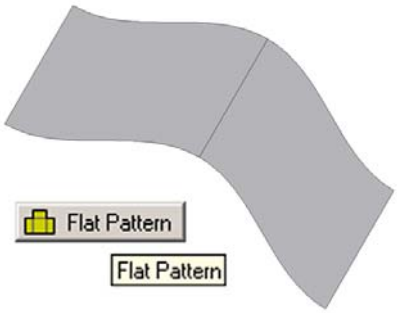


Figure 1E-2G: The Flat Pattern view.

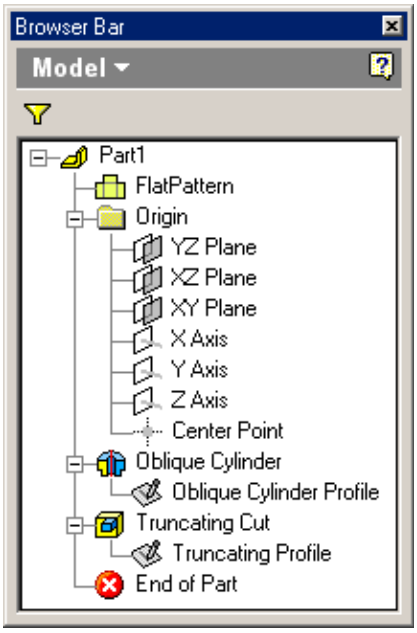


Figure 1E-2H: All features of the part are listed tin the Model panel.

Step 3: Creating the top piece

Go to the **File→New** menu to start another Inventor **Sheet Metal (in).ipt** part file; the *Sketch1* feature is automatically created; click the **Return** button to exit the sketch; right-click the *Sketch1* feature in the **Model** panel for the shortcut menu and choose the **Delete** option to delete it (*Figure 1E-3A*).

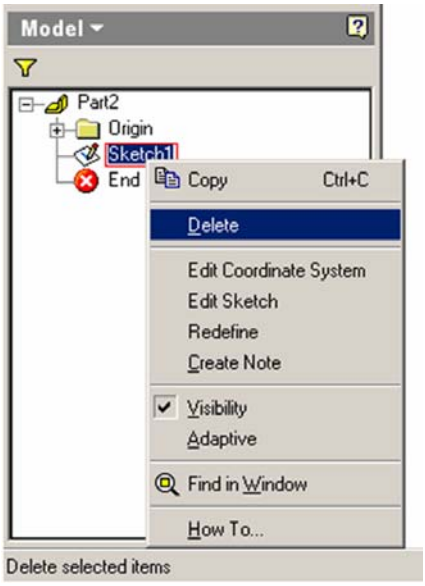


Figure 1E-3A: Deleting the default Sketch1 feature.

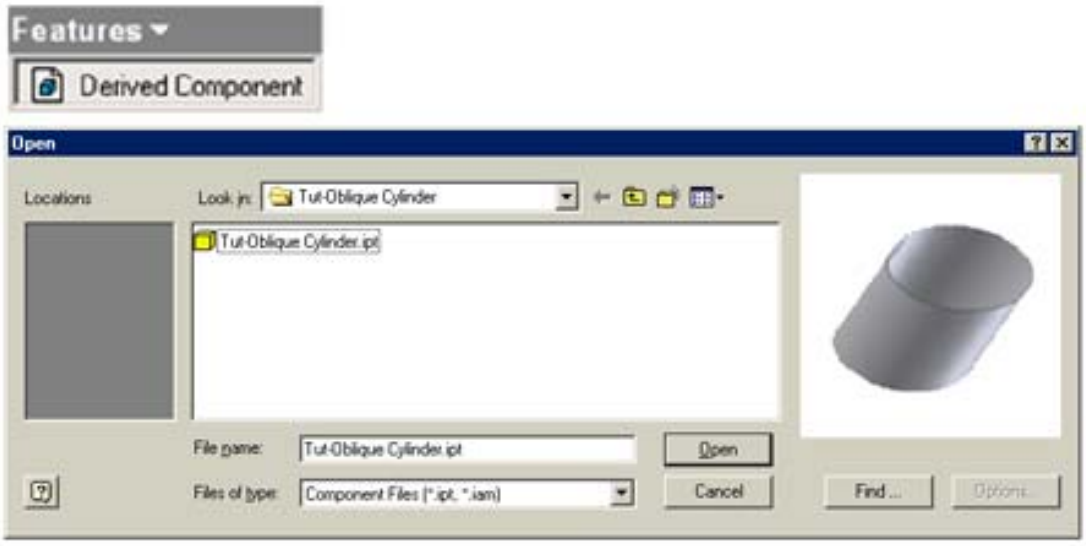


Figure 1E-3B: Opening the Tut-Oblique Cylinder.ipt file as a Derived Part.

Next, select the **Derived Part** tool from the **Features** tool pane; the **Open** dialog window appears (*Figure 1E-3B*); select the *Tut-Oblique Cylinder.ipt* file from the *Tut-Oblique Cylinder* folder; click the **Open** button; next, in the **Derived Part** dialog window that opens, select the **Body as Work Surface** option, and click the **OK** button (*Figure 1E-3C*) to import the reference geometry of the 3D model from the *Tut-Oblique Cylinder.ipt* file into the new file. Save the file as *Tut-Oblique Cylinder Top.ipt* in the same *Tut-Oblique Cylinder* folder. Save often after the completion of each feature.

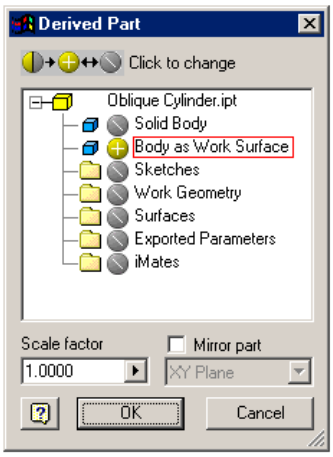


Figure 1E-3C: Opening the Derived Part with the Body as Work Surface option.

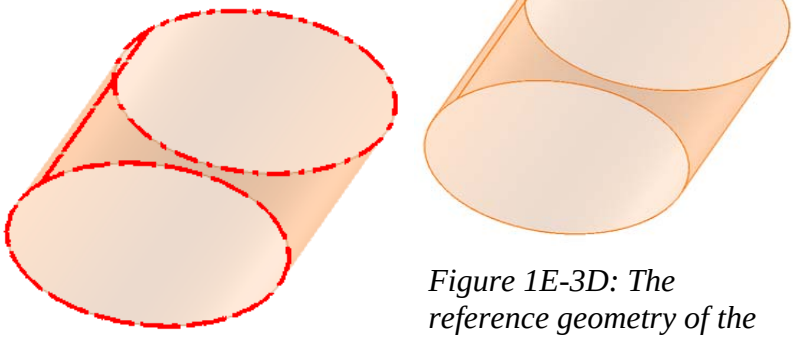


Figure 1E-3D: The reference geometry of the 3D model of the Tut-Oblique Cylinder.ipt file imported into the new Tut-Oblique Cylinder Top.ipt file.

Next, use the **Zoom Window** tool to zoom in a small portion of the top edge so that the truncated surface can be selected (*Figure 1E-3E*); click-select the narrow truncated surface and click the **Sketch** button to start a new sketch; rename it *Top Face Profile* in the **Model** panel; select the **Project Geometry** tool and click the outer edge of the **Work Surface** of cylindrical piece to project it onto the sketch (*Figure 1E-3F*). Remember that when the cylindrical piece is created by the **Revolve** tool, its cross-section profile is revolved by 359.999° and therefore, there is a gap to allow Inventor to generate the **Flat Pattern** view; however, in order to create the top face sheet metal feature, a closed profile is needed; thus, the gap in the projected outer edge line must be filled in. Use the **Zoom Window** tool to zoom in the tiny area of the gap (*Figure 1E-3G*); use the **Line** tool to add a short line segment so as to fill in the gap and turn the open profile into a close profile (*Figure 1E-3H*); click the **Return** button to exit the sketch. Save the file.

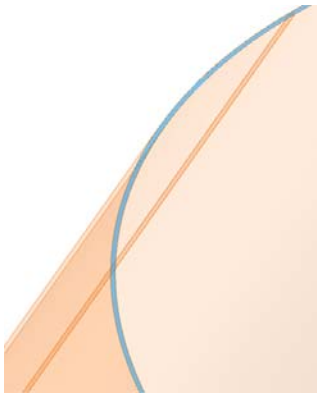


Figure 1E-3E: Zooming in a small portion of the truncated surface and click-selecting it to start the Top Face Profile sketch.

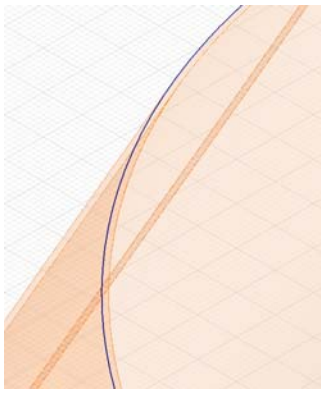


Figure 1E-3F: Projecting the outer edge of the Work Surface of cylindrical piece onto the sketch.

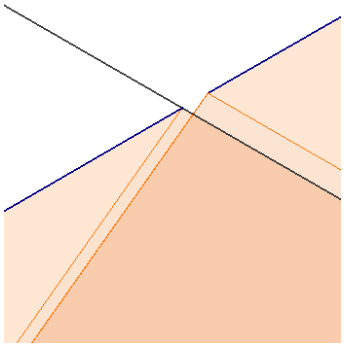


Figure 1E-3G: Zooming in the tiny area of the gap.

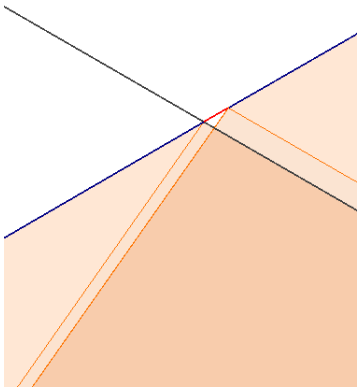


Figure 1E-3H: Drawing a short line segment to fill in the gap.

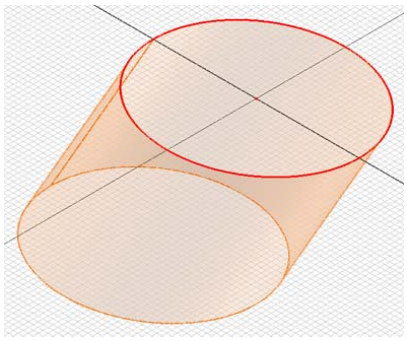


Figure 1E-3J: The closed profile for the top piece.

Next, select the **Face** tool to create the top piece; make sure that the green **Offset** direction arrow points upwards; otherwise, click the **Offset** button in the tool's dialog window to change the direction (*Figure 1E-4A*); click the **OK** button to create the **Face** feature; and rename it *Top Face* in the **Model** panel. Select the top surface of the *Top Face* (*Figure 1E-4B*) to create the Flat Pattern view (*Figure 1E-4C*). All features of the part are listed in the **Model** panel (*Figure 1E-4D*). Save and close the file.

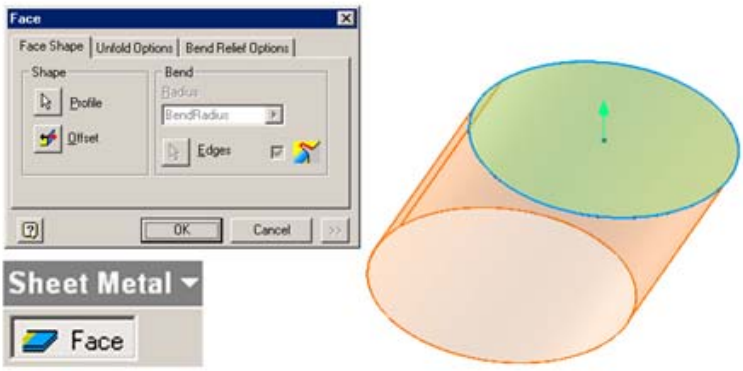


Figure 1E-4A: Creating the Top Face feature.

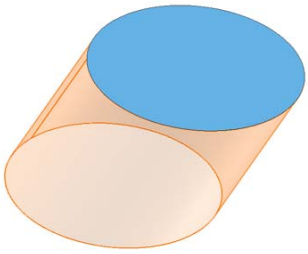


Figure 1E-4B: Selecting the top surface of the Top Face to create the Flat Pattern view.

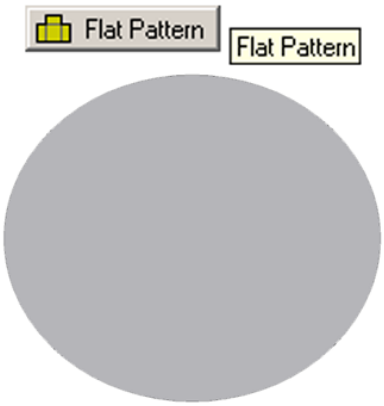


Figure 1E-4C: The Flat Pattern view window.

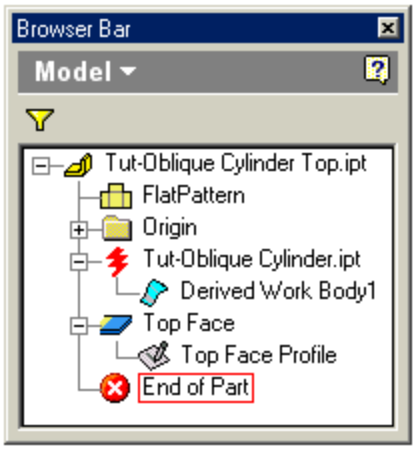


Figure 1E-4D: All features of the part are listed in the Model panel.

Step 4: Assembling the lateral, top and base pieces

Go to the **File→New** menu to start an Inventor **Standard (in).iam** assembly file (*Figure 1E-5A*); after the assembly file opens, select the **Place Component** tool; in the **Open** dialog window, select the *Tut-Oblique Cylinder Top.ipt* file, and click the **OK** button (*Figure 1E-5B*); the 3D model of the top piece appears on screen; click once to create an occurrence of the part; right-click for the shortcut menu and choose the **Done** option (*Figure 1E-5C*). Next, select the **Place Component** tool again; in the **Open** dialog window, select the *Tut-Oblique Cylinder.ipt* file, and click the **OK** button (*Figure 1E-5D*); the 3D model of the lateral cylindrical piece appears on screen; click once to create an occurrence of the part; right-click for the shortcut menu and choose the **Done** option to

finish (*Figure 1E-5E*). Save the file as *Tut-Oblique Cylinder Top Base & Lateral.iam* in the same *Tut-Oblique Cylinder.ipt* folder.

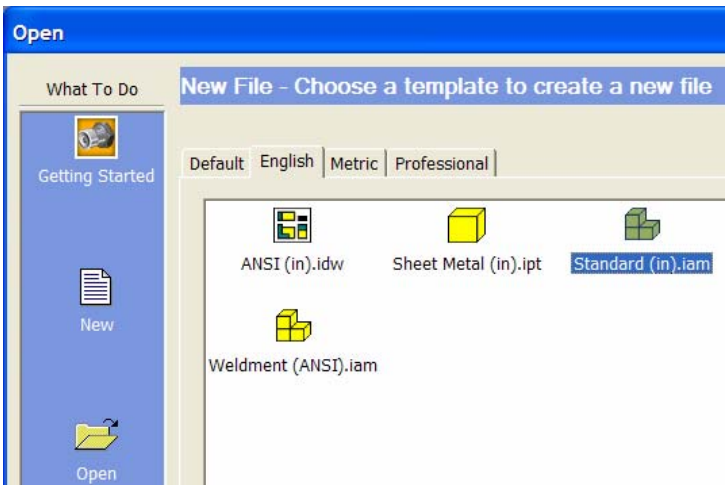


Figure 1E-5A: Starting an Inventor Standard (in).iam assembly file.

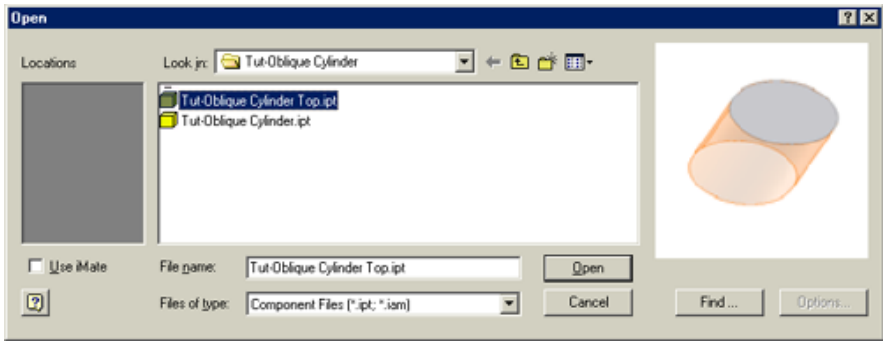


Figure 1E-5B: Selecting the Tut-Oblique Cylinder Top.ipt file in the Open dialog window.

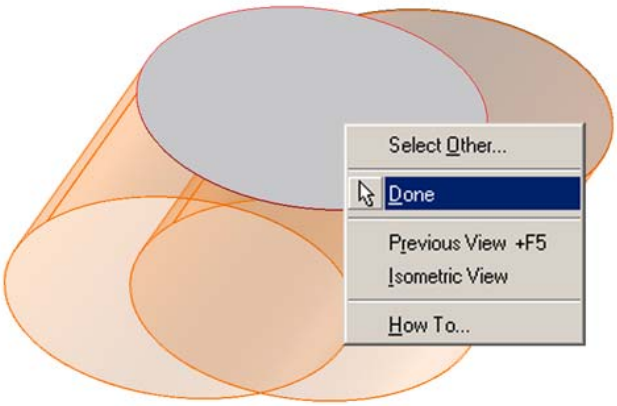


Figure 1E-5C: Creating an occurrence of the top piece.

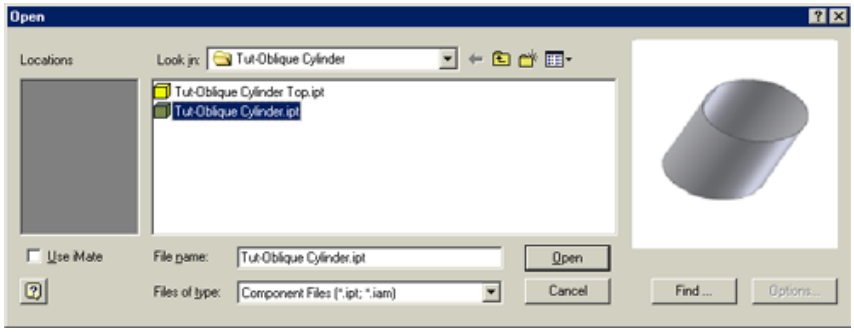


Figure 1E-5D: Selecting the Tut-Oblique Cylinder.ipt file in the Open dialog window.

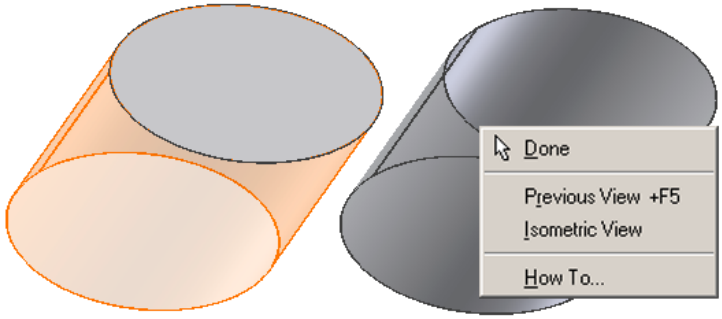


Figure 1E-5E: Creating an occurrence of the lateral oblique cylindrical piece.

Next, Assemble the top piece and the cylindrical lateral piece. The shortcut method to accomplish this is to apply **Flush Mate Constraint** to pairs of **YZ Plane**, **XZ Plane**, and **XY Plane** features between each of the **Place Component** features, and between each of the **Place Component** features and the assembly file's own **Plane** features. Select the **Place Constraint** tool; in the tool's dialog window, select **Mate** for **Type**, and **Flush** for **Solution**; in the **Model** panel, click the + button on the left of the **Origin** folder of the *Tut-Oblique Cylinder Top Base _Lateral.iam*, *Tut-Oblique Cylinder .ipt*, and *Tut-Oblique Cylinder Top.ipt* components, then click-select a pair of similar **Plane** features between two components; after selecting two similar **Plane** features, the program gives a loud sound; click the **Apply** button to apply a **Flush Mate** constraint; the **Flush** constraint features appears in the **Model** panel. Repeat the same process to apply the same **Constraint** to all pairs of same **plane** features (*Figure 1E-5F*). All features are listed in the **Model** panel (*Figure 1E-5G*). Save the file.

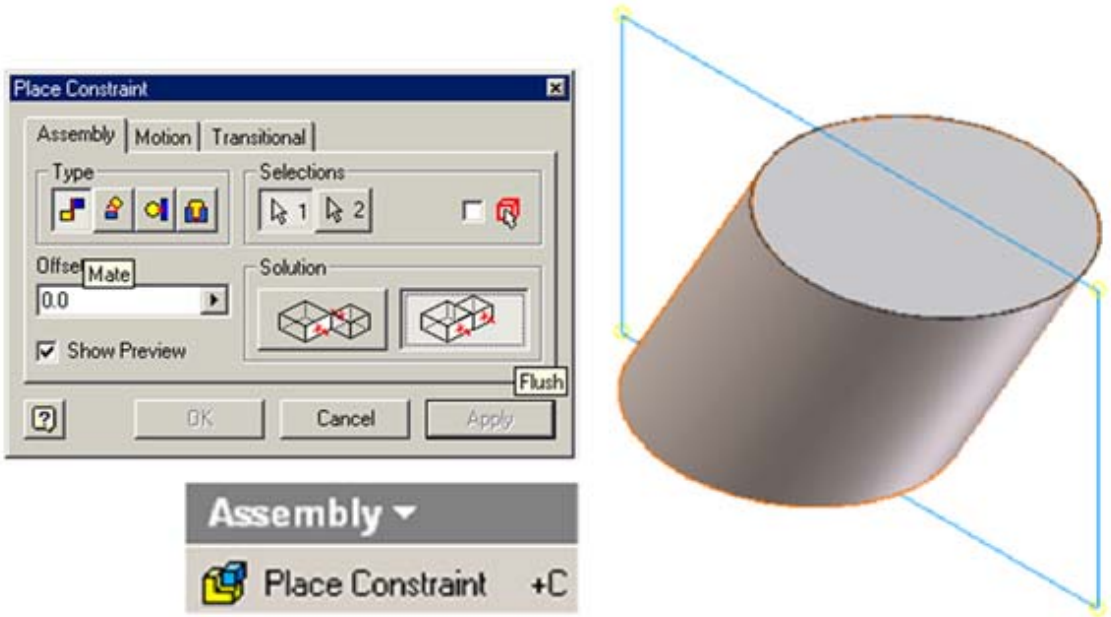


Figure 1E-5F: Repeating the same process to apply the same Constraint to all pairs of same plane features

Notice that the assembly has a strong orange tint; and this is so because the **Work Surface** model of the **Derived Part** in the *Tut-Oblique Cylinder Top.ipt* file is visible. Open this file, and in the **Model** panel, click the + button on the left of the *Tut-Oblique Cylinder.ipt* feature to open it; Select the *Derived Work Body1* feature, right-click for the shortcut menu and uncheck **Visibility** (*Figure 1E-5H*); the orange **Work Surface** model disappears from the screen (*Figure 1E-5J*) and its listing is grayed out in the **Model** panel. Save and close the file.

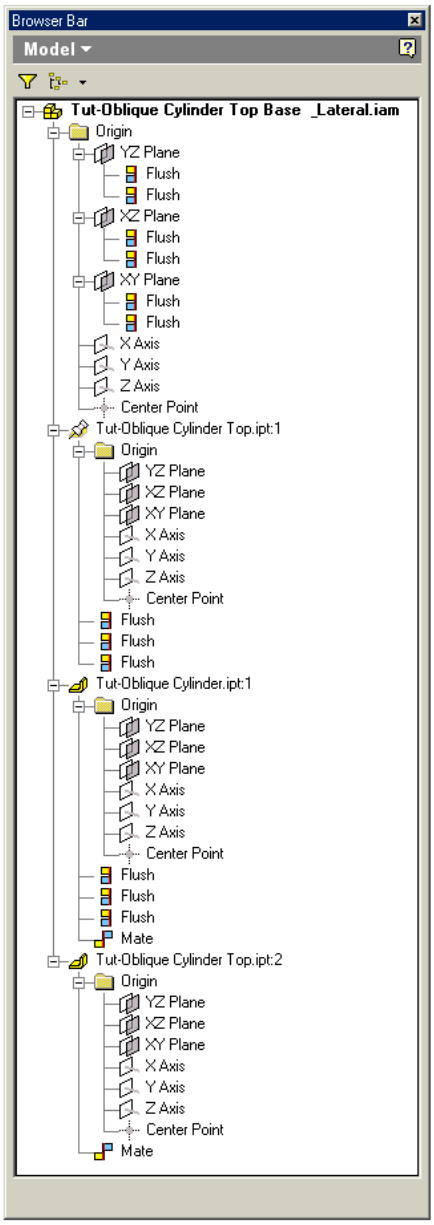


Figure 1E-5G: All features are listed in the Model panel.

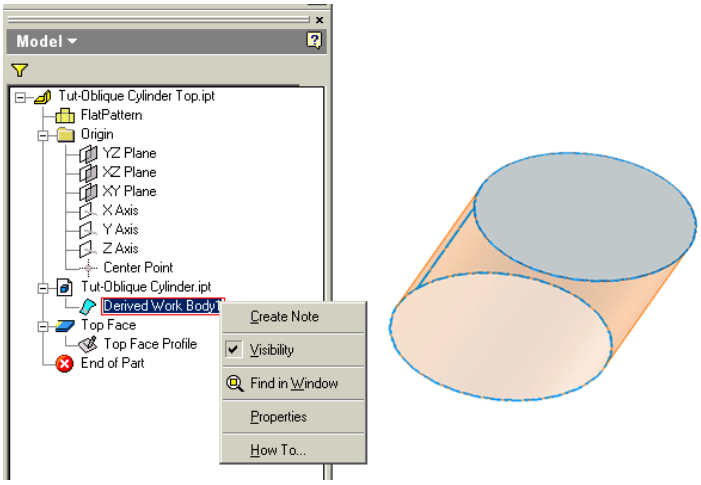


Figure 1E-5H: Removing Visibiity of the Derived Part.

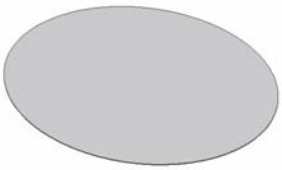


Figure 1E-5J: The Derived Part's Work Surface geometry no longer visible.

Figure 1E-5K: Placing another occurrence of the top piece, to be used as the base piece.

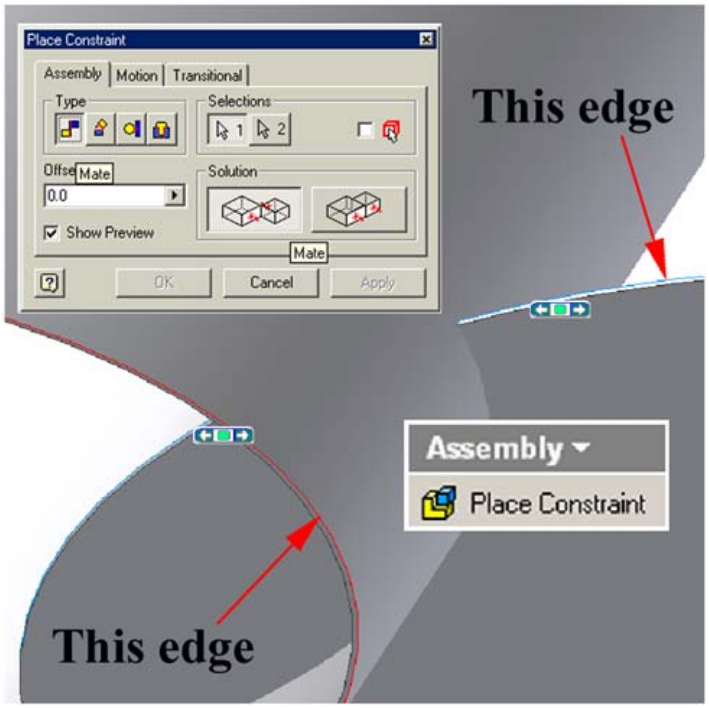


Figure 1E-5L: Applying a Mate Type Mate Solution constraint between the cylindrical lateral piece and the top piece used here as the base piece.

Next, go back to the *Tut-Oblique Cylinder Top Base & Lateral.iam* assembly file; select the **Place Component** tool again to pace another copy of the top piece from the *Tut-Oblique Cylinder Top.ipt* file in the assembly file (*Figure 1E-5K*); since the top piece and the base piece are the same, the copy of the top piece can be used as the base piece.

Next, select the **Place Constraint** tool; in the tool's dialog window, select **Mate** for **Type** and **Mate** for **Solution**, as shown in *Figure 1E-5L*, click-select the outer bottom edge of the cylindrical piece and the top edge of the top piece (used as the base piece here), click the **OK** button to apply the constraint. The two edges from both pieces immediately match. The assembly file is complete. Save the file.

If desired, then open the *Tut-Oblique Cylinder.ipt* file, go to the **Color** drop-down list to change the material rendering the cylindrical part. The *Tut-Oblique Cylinder Top Base & Lateral.iam* file immediately updates (*Figure 1E-5N*).

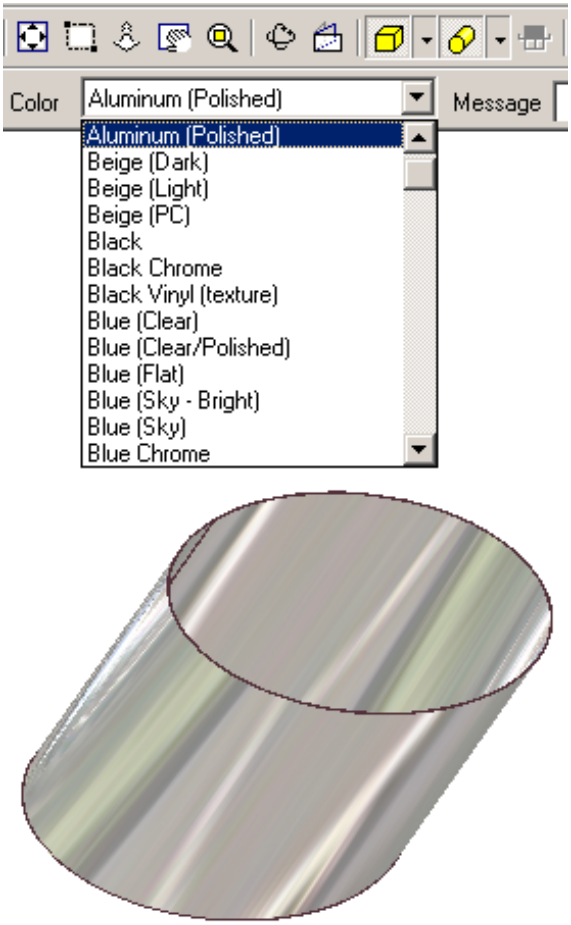


Figure 1E-5M: Applying a material style.

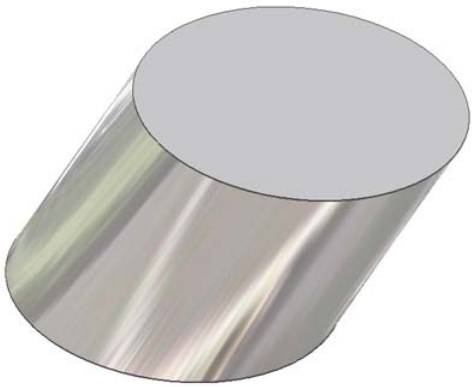


Figure 1E-5N: The resulting look of the assembly file.

Congratulations! In the *Module 1E*, you have learned how to create sheet-metal parts wrapping the 3D space of an oblique cylinder with a circular cross-section.