

Module 1D: Parallel-Line Flat Pattern Development of A Sheet-Metal Folded Model Wrapping The 3D Space of an Oblique Hexagonal Prism

In this Module, we will create a parallel-line flat pattern development of a sheet-metal folded model wrapping the 3D space of an oblique hexagonal prism in Inventor. An oblique prism is one with all elements (or lateral edge lines) parallel to a central axis that is not upright (or at a 90° angle relative to horizontal), but inclined (or at any angle $0^\circ < \theta < 90^\circ$ relative to horizontal). The basic method to complete this task is:

- Creating the 3D solid model of an oblique prism with a hexagonal cross-section, to be used as a **Derived Part** in another Inventor **Sheet Metal (in).ipt** file;
- Importing the 3D solid model of the oblique prism with a hexagonal cross-section from the first file as a **Derived Part**, in a **new** Inventor **Sheet Metal (in).ipt** file, and creating **Face** features wrapping around the 3D geometry of the 3D solid model of the **Derived Part**.

In this *Module*, this basic method will be explored.

Section 1: Creating The 3D Solid Model of An Oblique Prism with A Hexagonal Cross-Section

Step 1: Creating the oblique prism with a hexagonal cross-section

Launch Autodesk Inventor and start a new **Sheet metal (in).ipt** file under the **English** tab. A new **Sketch1** is automatically created on the **XY Plane**; rename it *Construction Sketch* in the **Model** panel; select the **Project Geometry** tool and the **Center Point** feature from the **Model** panel to project it onto the sketch; select the **Line** tool and draw a vertical line, and a horizontal line, and an angled line (the “oblique axis line”), all starting from the projected **Center Point**, and another horizontal line starting at the top endpoint of the vertical line (click the mouse at the appearance of the green dot snap indicator), as shown in *Figure 1D-1A*; next, select the **General Dimension** tool to apply a 60° angular dimension between the angled line and the first horizontal line, and a

36-inch linear dimension on the vertical line (*Figure 1D-1B*); select the angled line and go to the **Command Bar** to change its **Style** to **Centerline**; next, select the **Extend** tool and the “oblique axis line” to extend it up to the height of the vertical line; click the **Return** button on the **Command Bar** to exit the sketch. Save the file as *Tut-Oblique Prism* inside a new folder to be created and named as *Tut-Oblique Prism.ipt* in the **Save As** window. Save often at the end of each step.

Next, go to the **View**→**Isometric** menu to switch to an isometric view (*Figure 1D-1C*). Select the **Work Plane** tool; move the mouse cursor closer to the top endpoint of the angled line; the outline indicator of a perpendicular work plane appears on the screen (*Figure 1D-1D*); click once to create the **Work Plane** (*Figure 1D-1E*); and rename it *Hex Prism Profile Work Plane* in the **Model** panel.

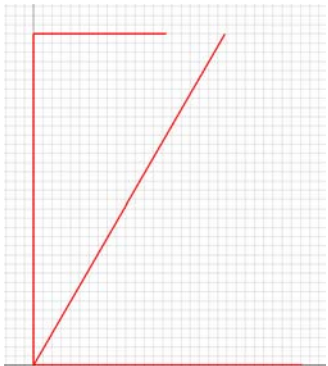


Figure 1D-1A: Starting the Construction Sketch.

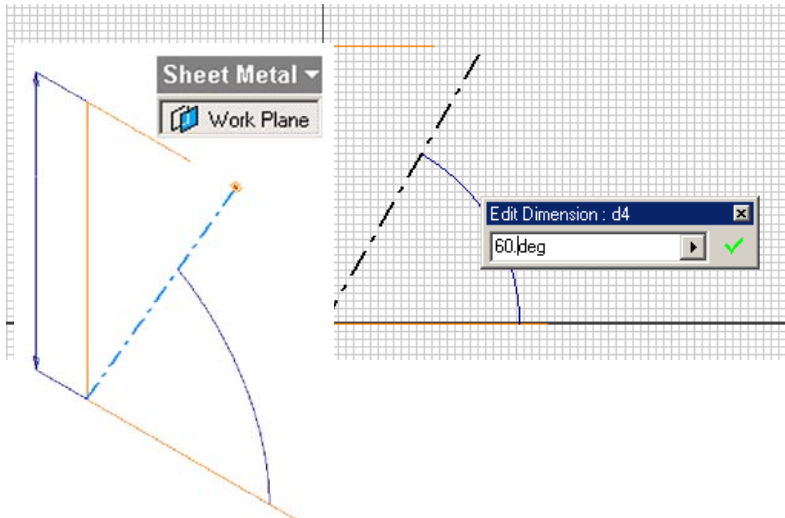


Figure 1C-1B: Applying linear and angular dimensions to the Construction Sketch.

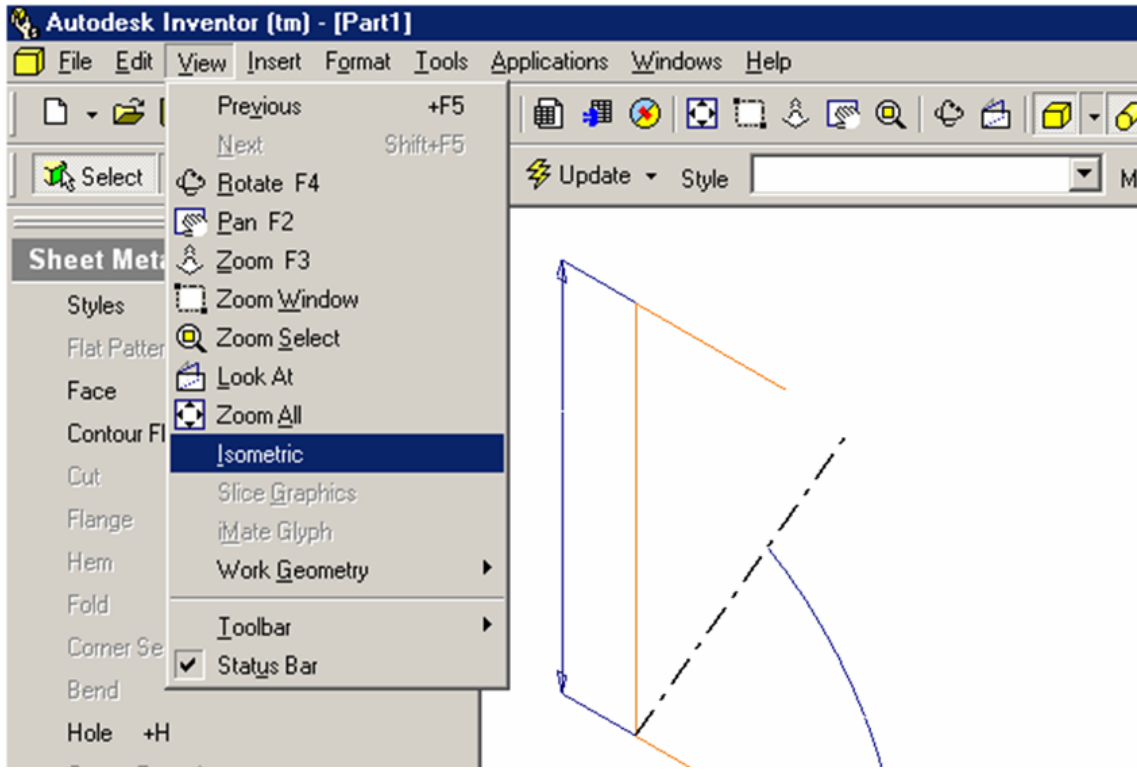


Figure 1D-1C: Switching to an isometric view.

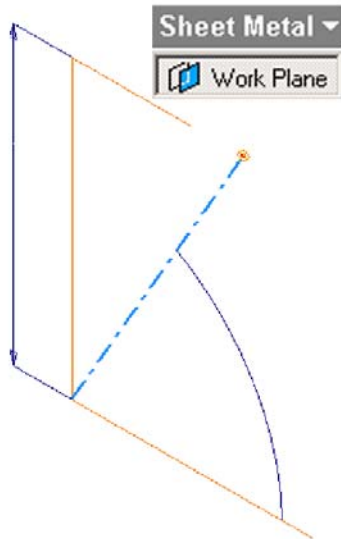


Figure 1D-1D: Selecting the Work Plane tool and moving the cursor closer to the top endpoint of the angled line.

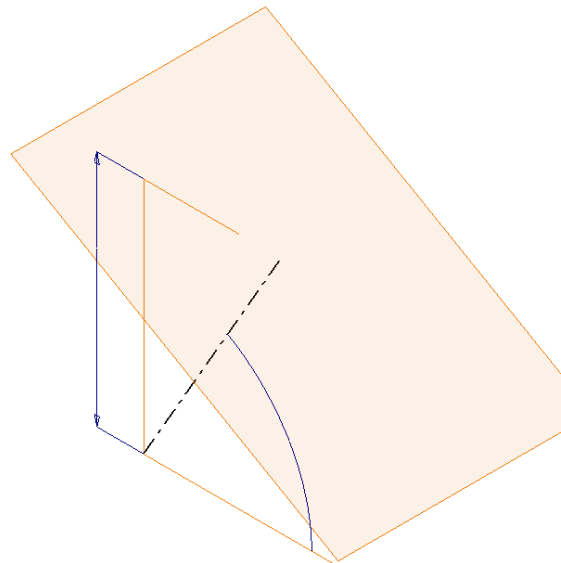


Figure 1D-1E: Creating the Hex Prism profile Work Plane.

Next, click-select the *Hex Prism Profile Work Plane* and click the **Sketch** button in the **Command Bar** to start a new sketch and rename it *Hex Prism Profile* in the **Model** panel; select the **Project Geometry** tool and click-select the “oblique axis line” to project it onto the new sketch (*Figure 1D-1F*); a point appears on the sketch, proving that the *Hex Prism Profile Work Plane* just created is indeed perpendicular to the “oblique axis line;” next, select the *Hex Prism Profile sketch* feature in the **Model** panel and click the **Look At** tool button on the **Command Bar** to switch to an orthographic view; next, select the **Line** tool to draw a horizontal line snapped to and starting at the point projected from the “oblique axis line;” and select the **General Dimension** tool to apply a 24-inch linear dimension (*Figure 1D-1G*); next, select the **Polygon** tool, click the **Circumscribed** option button and type 6 for hexagon in the text field, in the tool’s dialog window; move the cursor closer to the projected “oblique axis line” center point, click once at the appearance of the green dot endpoint snap indicator for the center point of the hexagon, and then move the cursor closer to the other endpoint of the horizontal line just drawn, and click once to create the circumscribed hexagon (*Figure 1D-1H*); click the **Done** button to exit the tool; and click the **Return** button on the **Command Bar** to exit the sketch feature.

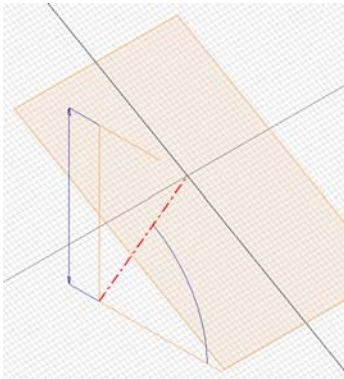


Figure 1D-1F: Projecting the “oblique axis line” onto the Hex Prism Profile sketch.

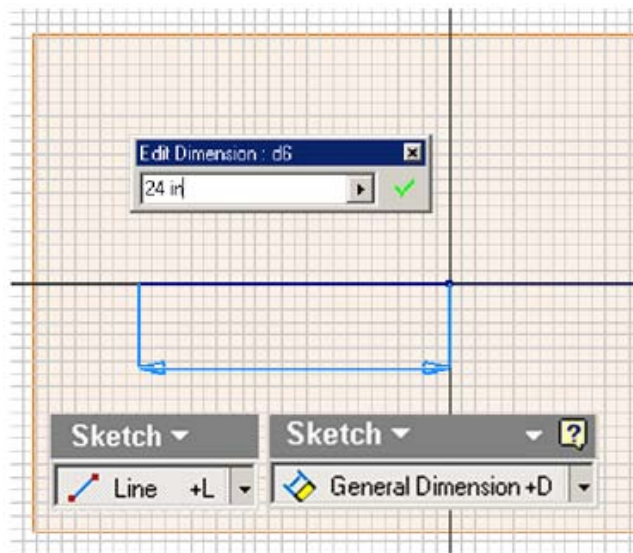


Figure 1D-1G: Drawing the “radius line” and applying a linear dimension.

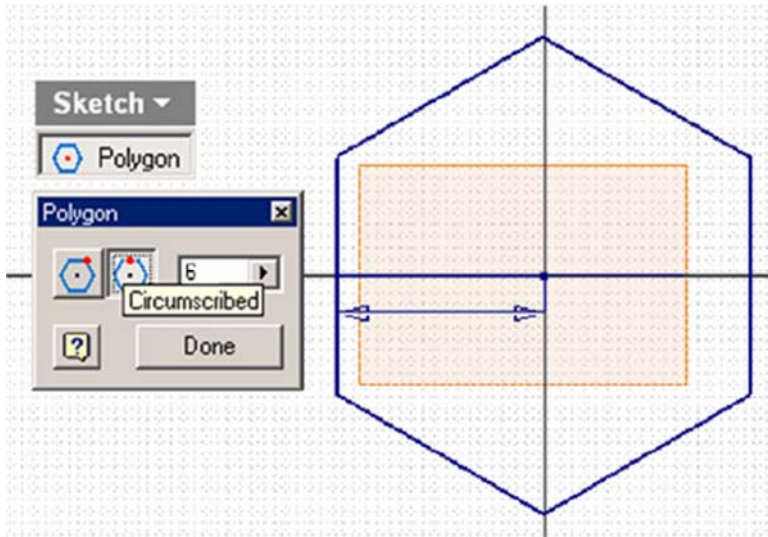


Figure 1D-1H: Creating the hexagon Circumscribed hexagon.

Next, select the **Extrude** tool; in the tool's dialog window, select the **Distance** option and the **Midplane** direction, and type 36 in (inches) in the text field; the outline geometry of the oblique prism appears on the screen (Figure 1D-1J); click the **OK** button to create the **Extrude** feature and rename it *Hex Prism* in the **Model** panel (See Figure 1D-1K).

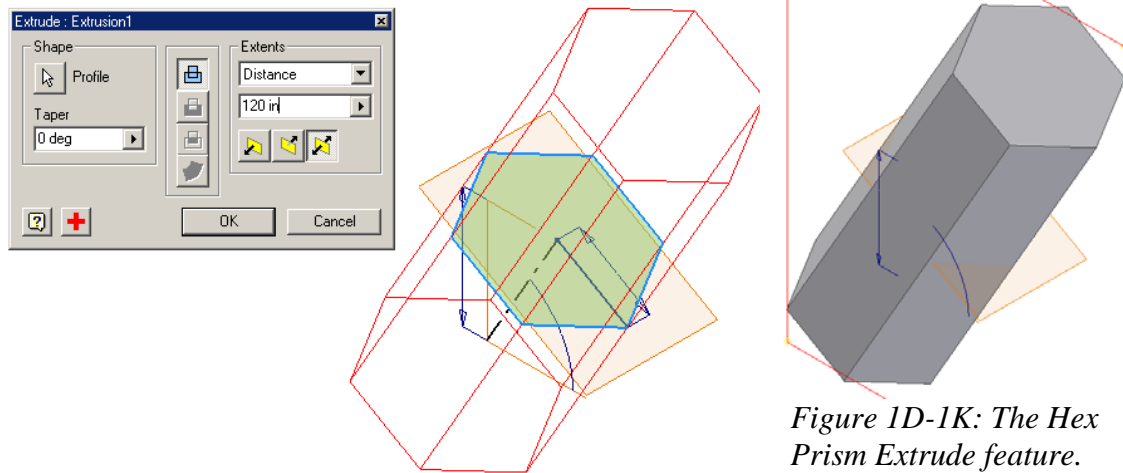


Figure 1D-1K: The Hex Prism Extrude feature.

Figure 1D-1J: Creating the Hex Prism with the Extrude tool.

Step 2: Truncating the prism

Select the **XY Plane** from the **model** panel; click the **Look At** and **Sketch** buttons on the **Command Bar** to switch to a normal view and to start a new sketch (*Figure 1D-2A*); and rename the new sketch *Horizontal Cut Profile* in the **Model** panel; select the **Project Geometry** tool and click-select the **Center Point** feature in the **Model** panel to project it onto the new sketch; next, select the **Line** tool and draw one horizontal line starting at and snapped to the projected **Center Point**, and another horizontal line above it; next, select the **General Dimension** tool to apply a 60 inch height dimension between the two lines (*Figure 1D-2B*); next, use the **Line** tool to draw additional horizontal and vertical lines (*Figure 1D-2C*); and use the **Extend** and **Trim** tools to complete two closed rectangular profiles which cover the upper and lower portions of the *Hex Prism Extrude* feature (*Figure 1D-2D*); click the **Return** button on the **Command Bar** to exit the sketch.

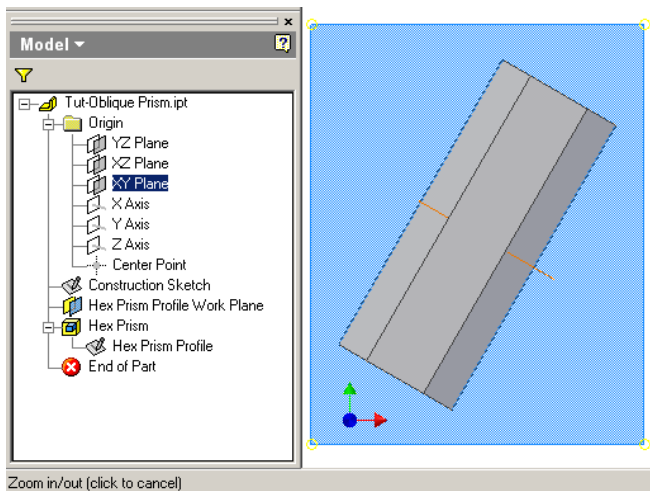


Figure 1D-2A: Selecting the XY Plane and switching to an orthographic normal view.

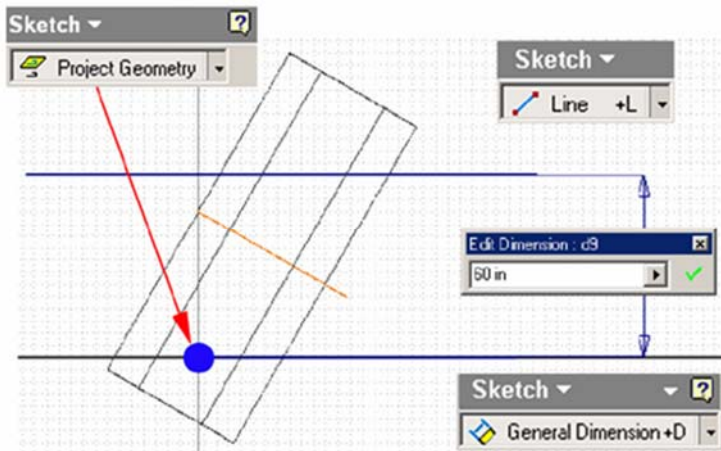


Figure 1D-2B: Projecting the Center Point, drawing two horizontal lines, and applying a height dimension.

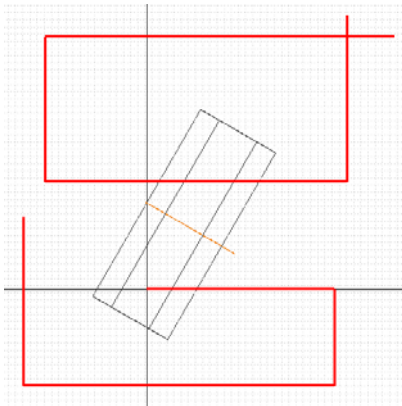


Figure 1D-2C: Drawing additional horizontal and vertical lines.

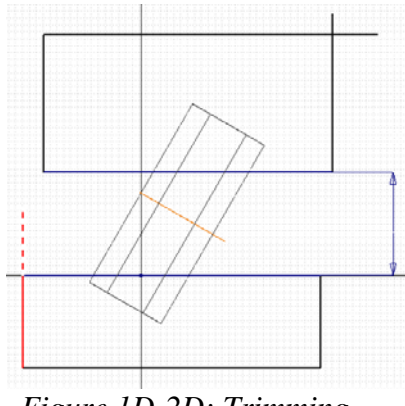


Figure 1D-2D: Trimming and extending.

Next, select the **Extrude** tool; in the tool's dialog window, choose the **Cut** option for extrusion type, **All** and **Midplane** in the **Extents** section; click the **OK** button to cut the oblique prism (*Figure 1D-2E*); and rename the **Extrude** feature *Horizontal Cut* in the **Model** panel. The 3D solid of the oblique prism is completed (*Figure 1D-2F*). Save and close the file.

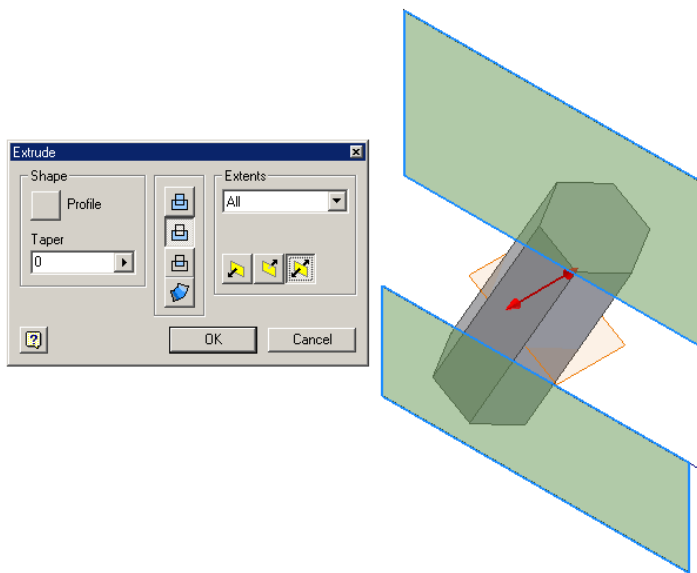


Figure 1D-2E: Cutting the oblique prism.

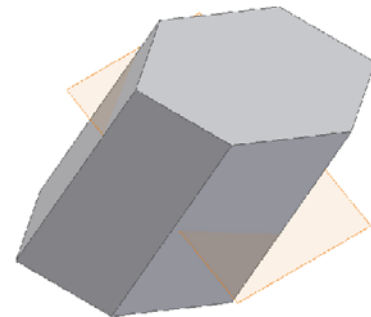
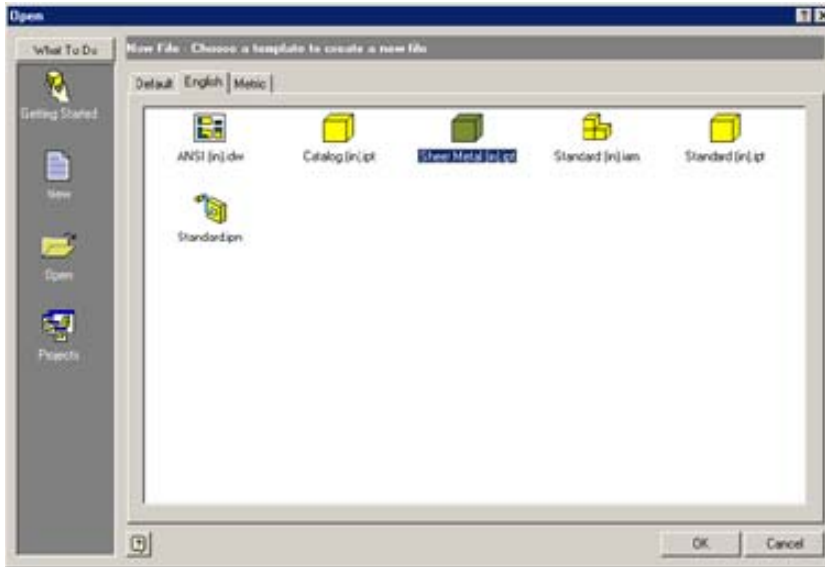


Figure 1D-2F: The completed 3D oblique prism.

Section 2: Creating The Sheet-Metal Part Wrapping The 3D Space of An Oblique Hexagonal Prism

Step 1: Creating the Face profile sketches



*Figure 1D-3A:
Start a new
Autodesk Inventor
Sheet Metal(in).ipt
file under the
English tab.*

Start a new Autodesk Inventor **Sheet Metal (in).ipt** file under the **English** tab (*Figure 1D-3A*); a new sketch is automatically created on the **XY Plane**; click the **Return** button to exit; click-select the *Sketch1* feature from the **Model** panel; right-click for a shortcut menu and choose **Delete** to delete it. Next, select the **Features** tool panel and select the **Derived Component** tool; in the **Open** dialog window, navigate to the *Tut-Oblique Prism* folder and select the *Tut-Oblique Prism.ipt* file (*Figure 1D-3B*); click the **Open** button; in the **Derived Part** dialog window (*Figure 1D-3C*), select the **Body as Work Surface** option, and click the **OK** button. The orange-color surface model of the oblique prism is created on the screen as reference geometry (*Figure 1D-3D*). Notice that the *Hex Prism Profile Work Plane* is also visible; if desired, then select this **Work Plane** feature either on the screen or from the **Model** panel, right-click for the shortcut menu and uncheck **Visibility** to make it invisible (*Figure 1D-3E*).

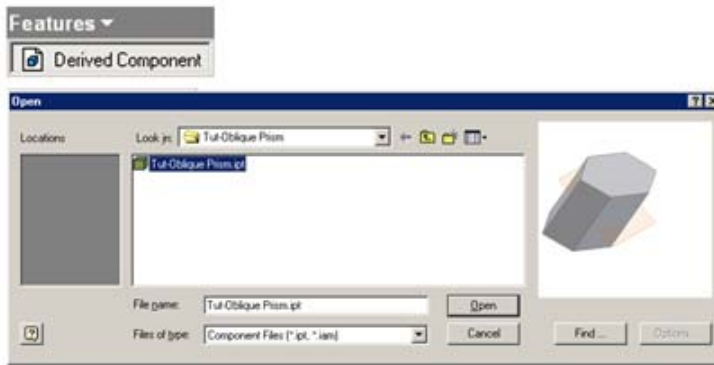


Figure 1D-3B: Selecting the Tut-Oblique Prism.ipt file in the Open window (left).

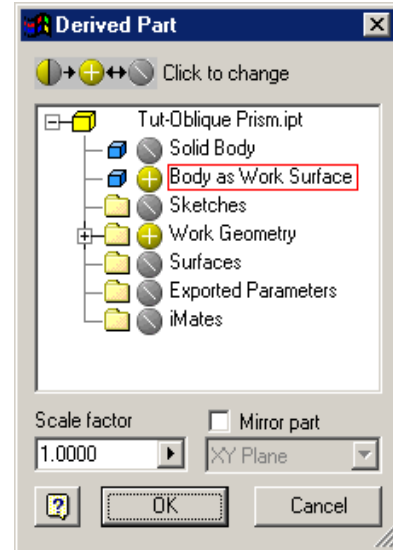


Figure 1D-3C: Selecting the Body as Work Surface option in the Derived Part window (right).

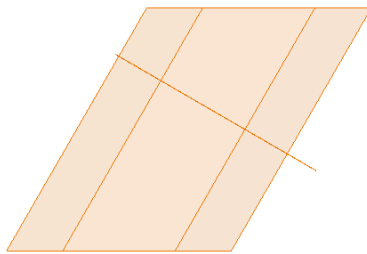


Figure 1D-3D: The Work Surface reference geometry of the 3D model of the oblique prism.

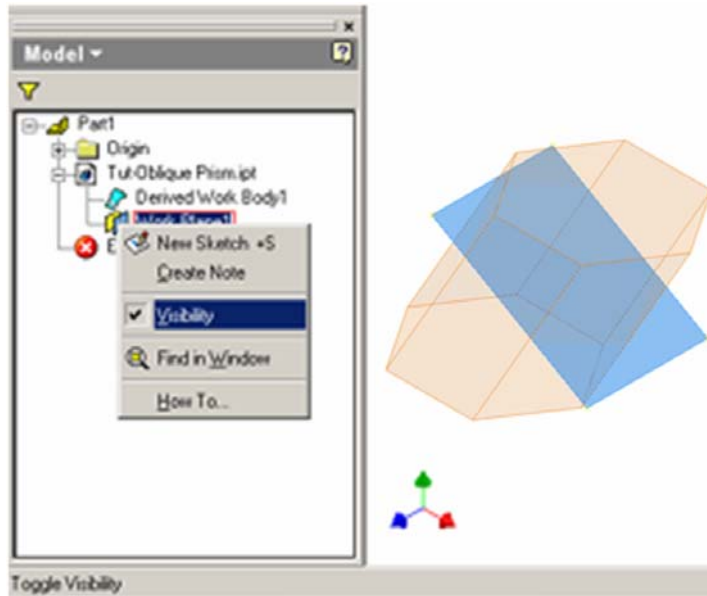


Figure 1D-3E: Turning off the Visibility of the Hex Prism Profile Work Plane.

Next, select the first five lateral surfaces of the **Derived Component** model to create closed quadrilateral profiles for the creation of sheet-metal **Face** features. Click-select the first surface on the model and click the **Sketch** button on the **Command Bar** to start a new sketch (Figure 1D-3F); select the **Project Geometry** tool and click-select the four edges of the surface to project them onto the new sketch (Figure 1D-3G); click the

Return button on the **Command Bar** to exit the sketch. Repeat the same procedures to create all five **Sketch** features corresponding to the first five lateral surfaces of the oblique prism, using the **Rotate** tool whenever needed; and rename the **Sketch** features in the **Model** panel as *Lateral Face 1 Sketch*, *Lateral Face 2 Sketch*, *Lateral Face 3 Sketch*, *Lateral Face 4 Sketch*, and *Lateral Face 5 Sketch* (Figure 1D-3H through Figure 1D-3N). Save the file as *Tut-Oblique Prism SM.ipt* inside the same *Tut-Oblique Prism* folder. Save often at the end of each step.

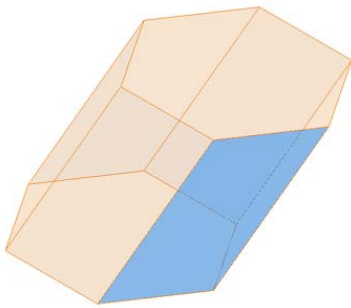


Figure 1D-3F:
Selecting the 1st lateral surface for creating the Lateral Face 1 Sketch.

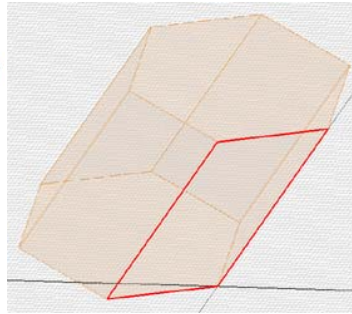


Figure 1D-3G:
Creating the Lateral Face 1 Sketch.

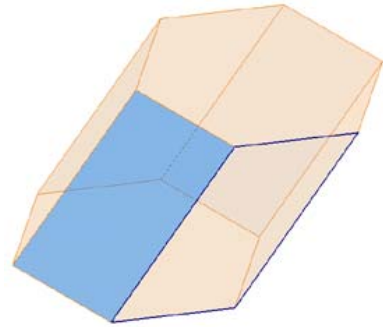


Figure 1D-3H: Selecting the 2nd lateral surface for creating the Lateral Face 2 Sketch.

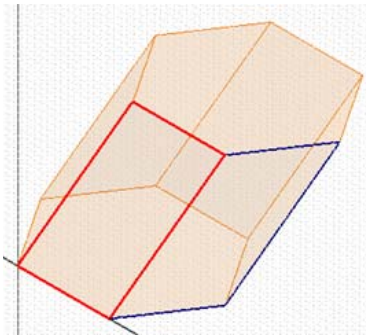


Figure 1D-3J:
Creating the Lateral Face 2 Sketch.

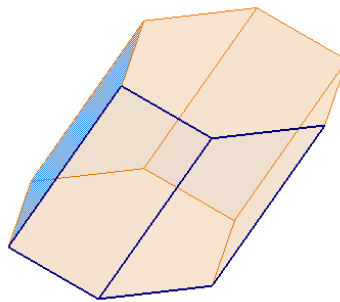


Figure 1D-3K: Selecting the 3rd lateral surface for creating the Lateral Face 2 Sketch.

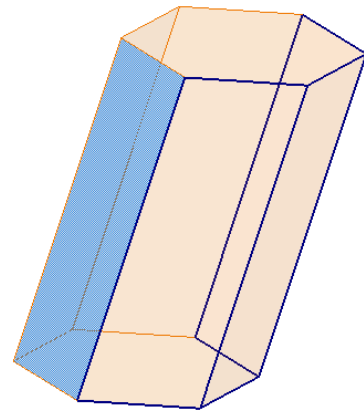


Figure 1D-3L: Creating the Lateral Face 3 Sketch, and selecting the 4th lateral surface for creating the Lateral Face 4 Sketch.

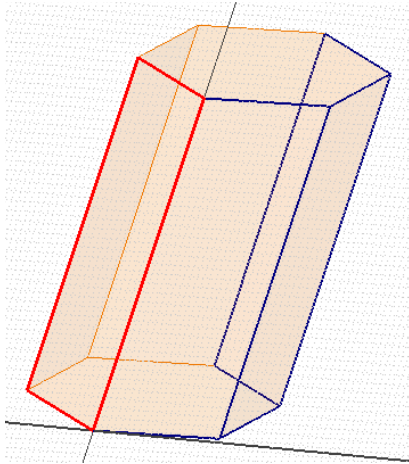


Figure 1D-3M: Creating the Lateral Face 4 Sketch.

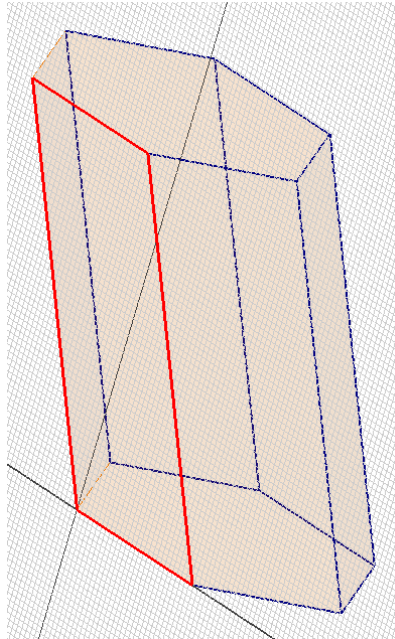


Figure 1D-3N: Creating the Lateral Face 5 Sketch.

Next, select the 6th surface and click the **Sketch** button on the **Command Bar** to start the profile sketch for the last lateral surface of the oblique prism (*Figure 1D-3P*); use the **project Geometry** tool to project the four edge lines onto the sketch; and rename it *Lateral Face 6 Sketch* in the **Model** panel (*Figure 1D-3Q*); use the **Rotate** tool to rotate the model to a better position (*Figure 1D-3R*); next, use the **Line** tool to draw a vertical line by the edge where the 1st and the 6th lateral surfaces meet, on the side of the 6th surface, and extending beyond the top and bottom horizontal edges, as shown in *Figure 1D-3S*.

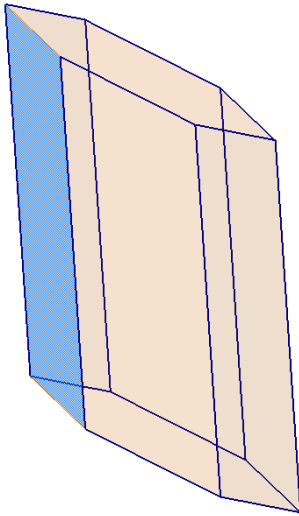


Figure 1D-3P: Selecting the 6th surface for the Lateral Face 6 Sketch.

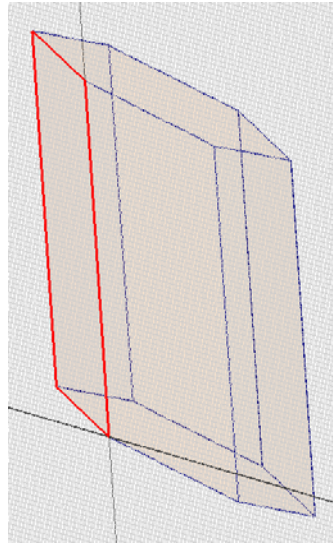


Figure 1D-3Q: Projecting the four edge lines onto the new sketch.



Figure 1D-3R: Rotating the 3D model to a better position for visualization.

Next, select the **General Dimension** tool to apply a 0.01-inch linear dimension between the vertical line just drawn and the projected edge line between the 1st and 6th lateral surfaces (*Figure 1D-3S*), this creates a tiny gap (*Figure 1D-3T*) between the two **Face** features to be created later, which allows Inventor to generate the **Flat Pattern** features after the completion of the **Face** features for all lateral surfaces of the oblique prism and the addition of a lateral edge seam; next, window-select all lines in the sketch projected or drawn so far and go to the **Style** drop-down menu on the **Command Bar** to change the lines' **Style** to **Normal** (*Figure 1D-3V*), this allows lines to be trimmed); next, use the **Zoom Window** tool to zoom in the top-left corner of the profile; select the **Trim** tool to delete the projected edge line between the 1st and the 6th lateral surfaces (*Figure 1D-3W*), and to cut off the unneeded segments of the vertical and horizontal lines (*Figure 1D-3X*). Repeat the same procedures to cut off the unneeded segments of the vertical and horizontal lines at the lower-left corner of the profile (*Figure 1D-3Y* and *Figure 1D-3Z*). Click the **Return** button in the **Command Bar** to exit the sketch.

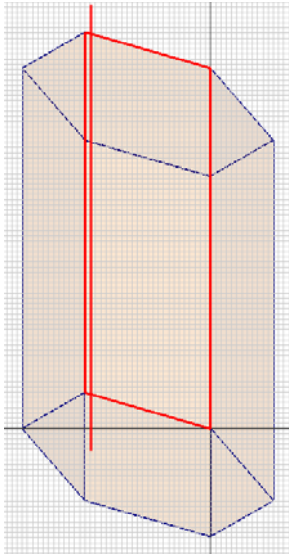


Figure 1D-3S:
Drawing the
vertical line for
the “gap.”

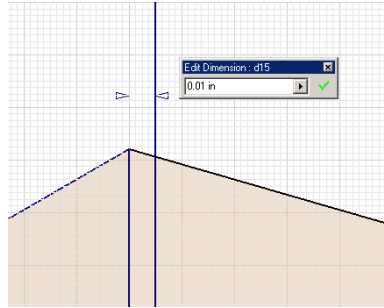


Figure 1D-3T: Applying a
0.01-inch dimension for
the “gap.”

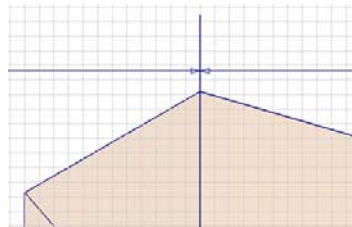


Figure 1D-3U: The tiny
“gap” needed to allow Flat
Pattern feature creation.

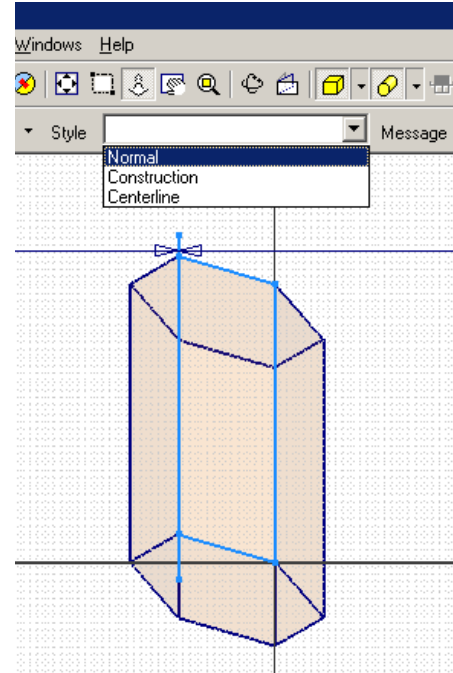


Figure 1D-3V: Window-
selecting all lines to change
their style to Normal.

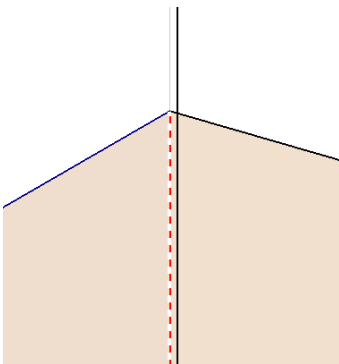


Figure 1D-3W:
Trimming off the
projected edge line
between the 1st and the
6th lateral surfaces.

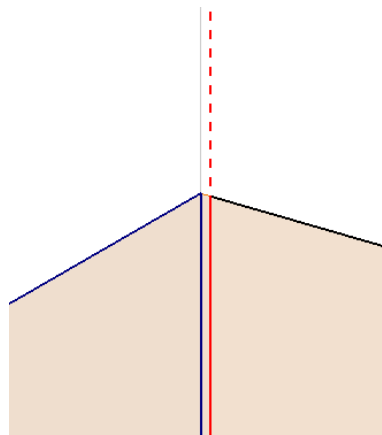


Figure 1D-3X: Cutting off
the unneeded segments of
the vertical and horizontal
lines.

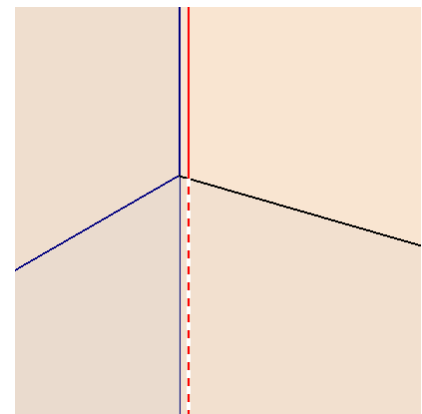


Figure 1D-3Y: Cutting off
the unneeded segments of
the vertical line at the
lower-left corner of the
profile.

Step 2: Creating the sheet-metal Face features for the lateral surfaces of the oblique prism

Next, select the **Face** tool to create the lateral sheet-metal panels (*Figure 1D-4A*); make sure that the green arrow points outward from the 3D **Work Surface** model of the oblique prism; if not, then click the **Offset** button in the tool's dialog window to change the direction; use the **Rotate** tool to help accessing all face profiles. Rename the **Face** features *Lateral Face 1*, *Lateral Face 2*, *Lateral Face 3*, *Lateral Face 4*, *Lateral Face 5*, and *Lateral Face 6* in the **Model** panel.

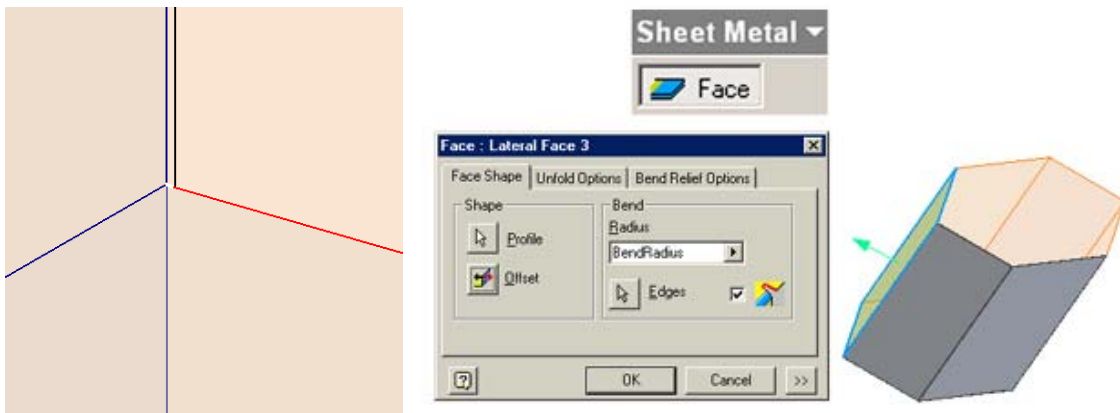


Figure 1D-3Z: Cutting off the unneeded segments of the horizontal line at the lower-left corner of the profile.

Figure 1D-4A: Creating the Lateral Face 1 through Lateral Face 6 features, with the Face extrusion direction arrow pointing outward from the 3D Work Surface model.

Step 3: Creating the lateral seam connecting the 1st and the 6th sheet-metal Face features

Use the **Rotate** tool to rotate the 3D model to a convenient position where the inside surfaces of the *Lateral Face 1* and *Lateral Face 6* can be seen (*Figure 1D-4B*); use the **Zoom Window** tool to zoom in the small area close to the corner so that the tiny gap can be seen (*Figure 1D-4C*). Next, select the **Flange** tool from the **Sheet Metal** tool panel; in the tool's dialog window, type 1 in (inch) in the **Distance** text field, click the arrow button in the **Shape** section and click-select the inner edge of the *Lateral Face 6* lateral panel, outline geometry of the **Flange** feature appears on the screen (*Figure 1D-4D*); highlight the default 90.0 (degrees) text in the **Angle** text field and press the **Delete** key on the keyboard to delete it; right-click for the **Angle** option's shortcut menu and

choose the **Measure** option (*Figure 1D-4E*); click the inner surface of *Lateral Face 1* and *Lateral Face 6* to measure the dihedral angle; the value of the angle automatically appears in the text field and the outline geometry of the **Flange** feature updates to the new angle (*Figure 1D-4F*); use the **Rotate** tool to rotate the model and check if the **Flange** feature is parallel to the *Lateral Face 1* surface panel; if not, then type 180- in front of the angle value in the **Angle** text field to correct it by changing the side from which the angle is calculated (*Figure 1D-4G*).

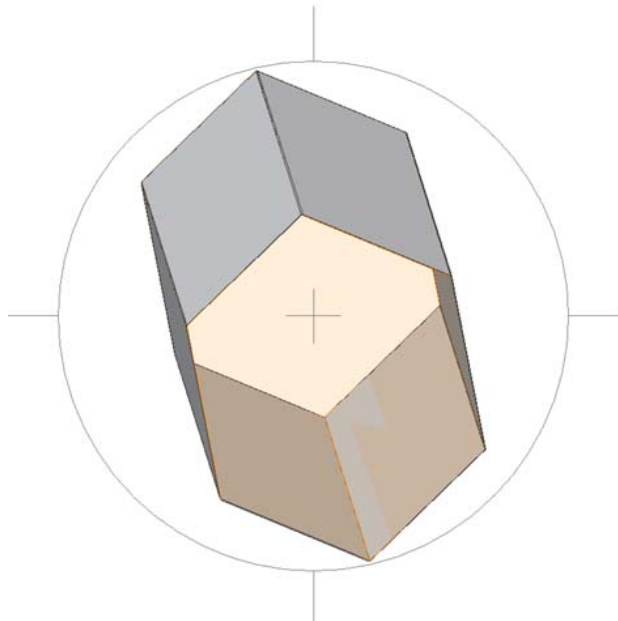


Figure 1D-4B: Rotating the model to a convenient position to view the inside surfaces of the Lateral Face 1 and Lateral Face 6.

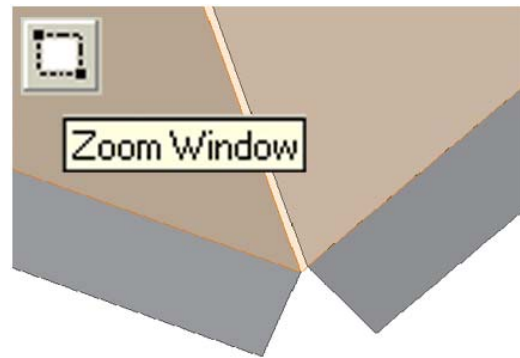


Figure 1D-4C: Zooming in the small area of the tiny gap.

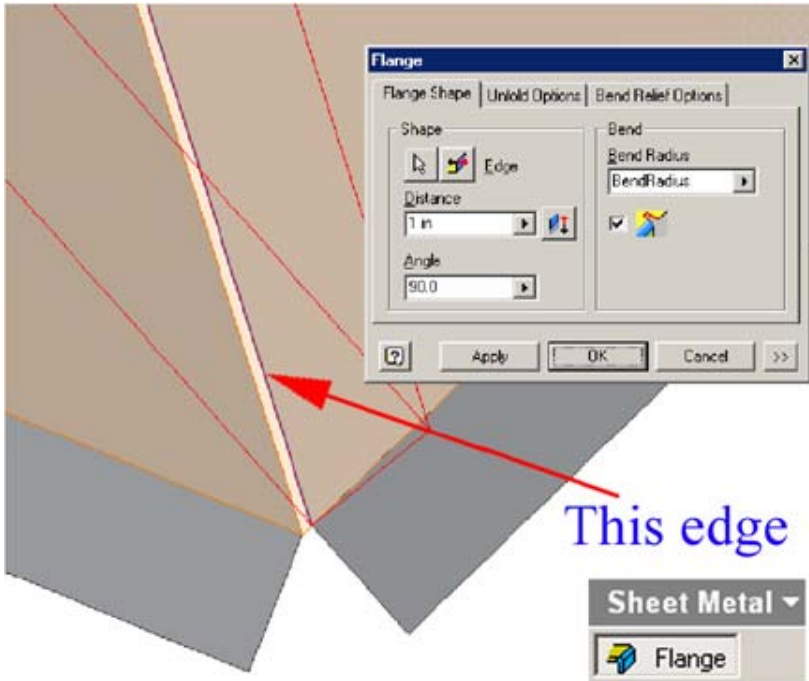


Figure 1D-4D:
Selecting the inner edge of the Lateral Face 6 lateral panel.

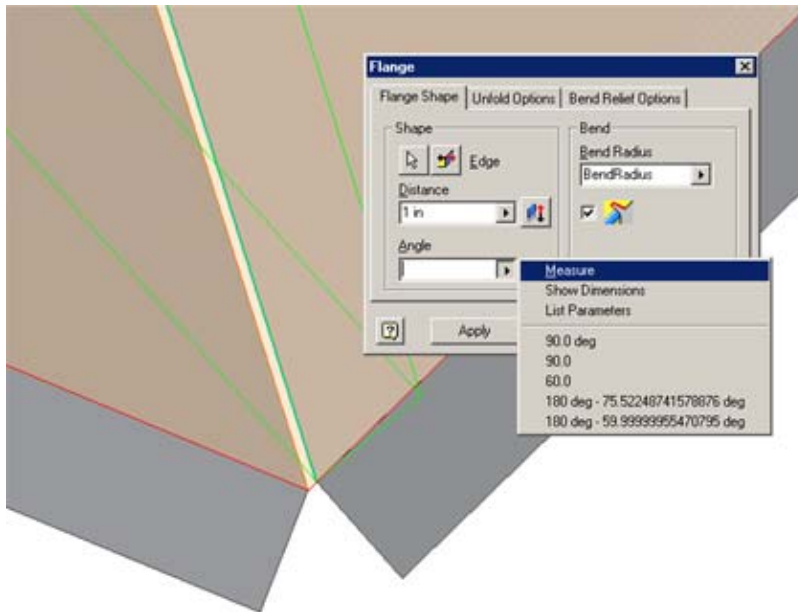


Figure 1D-4E:
Selecting the Measure tool.

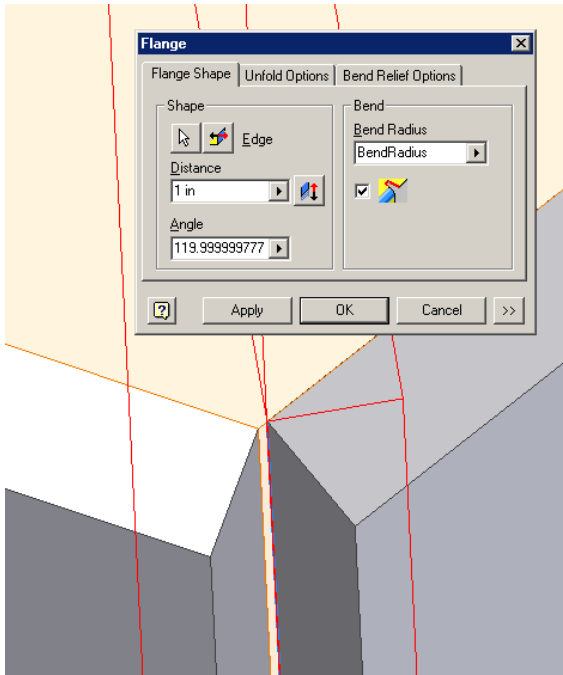


Figure 1D-4F: The measured angle.

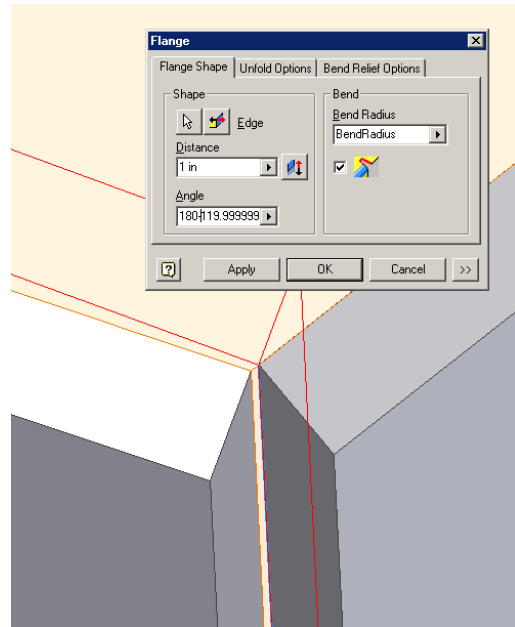


Figure 1D-4G: Correcting the angle measurement by typing 180- to change the side from which the angle is calculated.

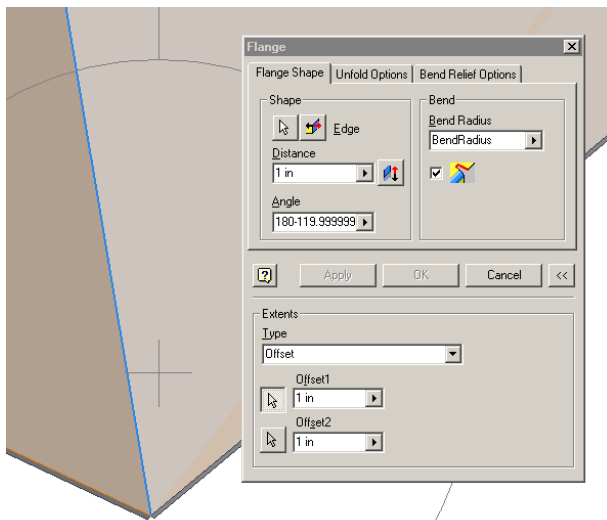


Figure 1D-4H: Expanding the tool's dialog window and selecting the Offset Type.

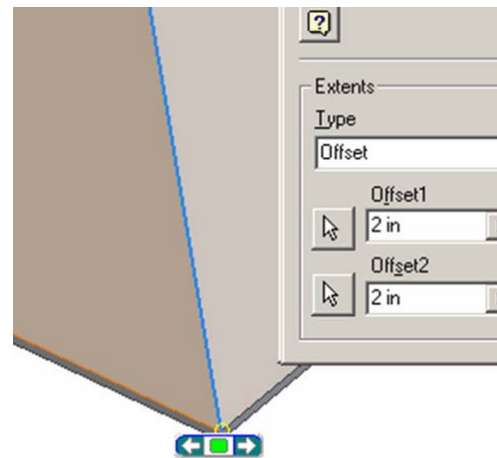


Figure 1D-4J: Changing the Offset1 and Offset2 values and selecting the corner points.

Next, click the >> button on the lower-right corner of the default dialog window to expand it and select the **Offset** option from the Type pull-down menu (*Figure 1D-4H*); type 2 in (inches) in the **Offset1** and **Offset2** text field for the **Offset** distance from the end corner points to be selected; and click-select the two end corner points of the inner edge line (*Figure 1D-4J*); next, use the **Rotate** tool again to check if the **Flange** will be extruded inwardly by verifying the direction of the green arrow (*Figure 1D-4K*), click the **Edge** button to make correction if necessary; click the **OK** button to create the **Flange** and rename it *Lateral Flange* in the **Model** panel.

Next, add chamfers to both end corners of the *Lateral Flange*. Select the **Corner Chamfer** tool from the **Sheet Metal** tool bar; by default, the **One Distance** option is selected in the tool's dialog window; type 0.875 in (inch) in the **Distance** text field; click-select the short edges on the cross-section at both end corners of the *Lateral Flange*; red outline geometry of the **Corner Chamfer** feature appear on the screen (*Figure 1D-4L*); click the **OK** button to create the **Corner Chamfers**; rename it *Lateral Flange Corner Chamfer* in the **Model** panel.

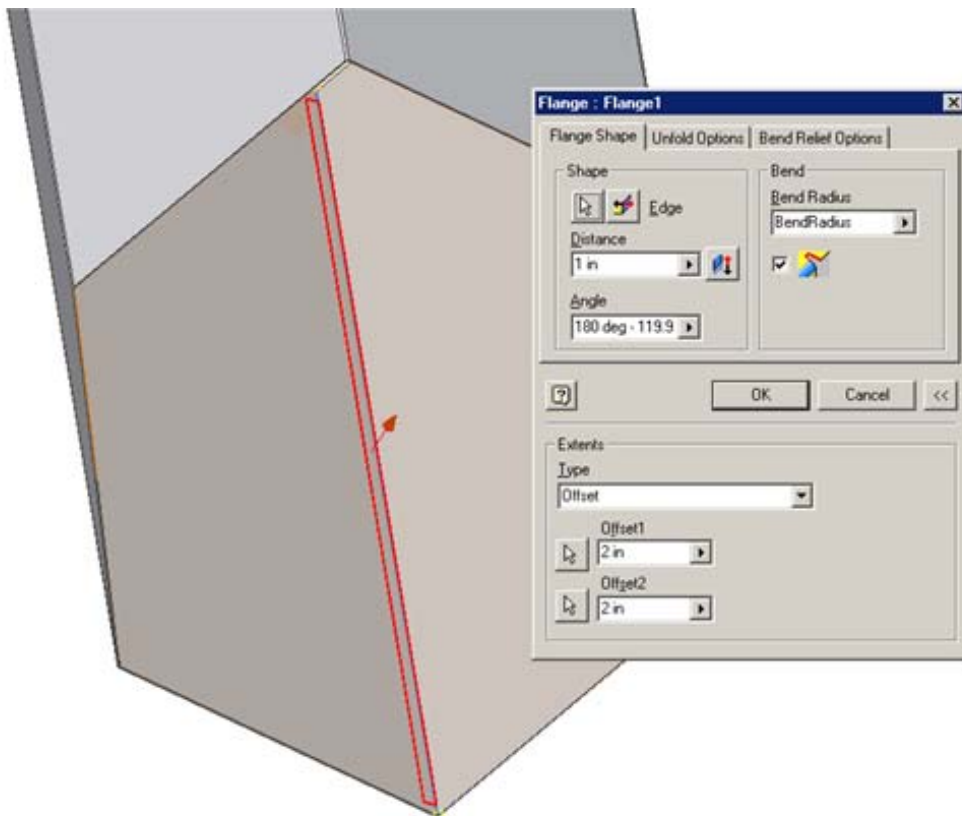


Figure 1D-4K: Checking if the Flange will be extruded inwardly.

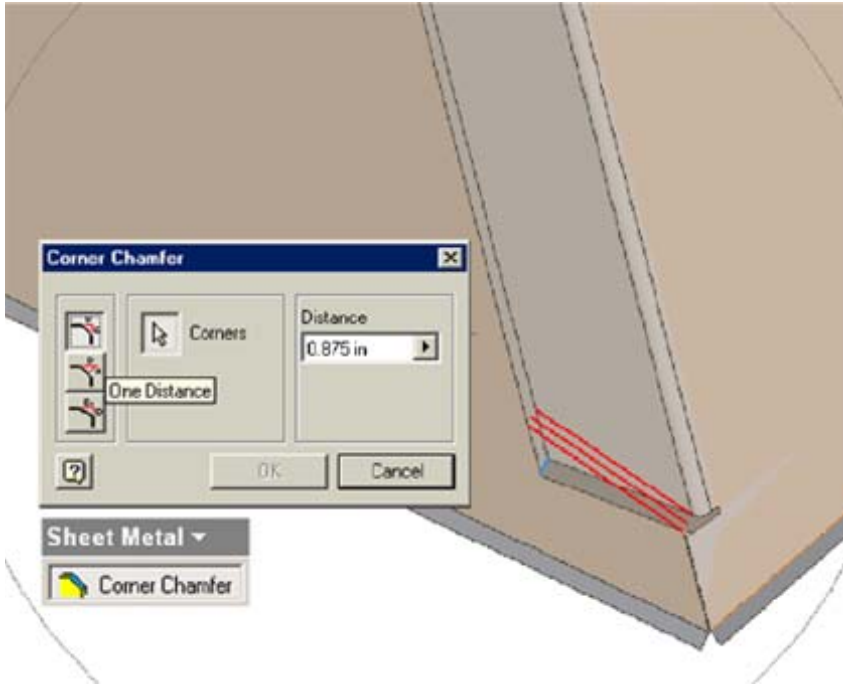


Figure 1D-4L: Creating the Lateral Flange Corner Chamfer feature.

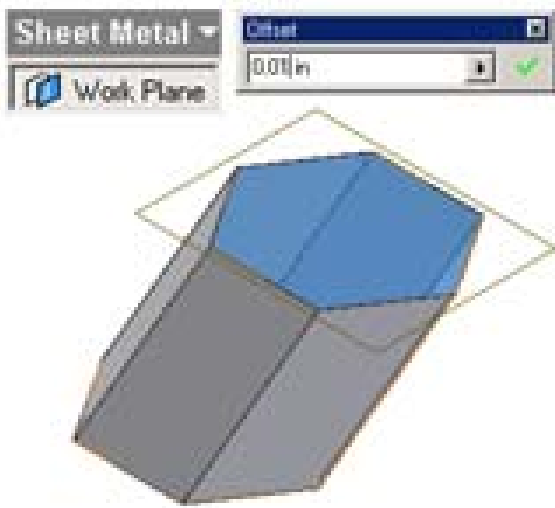


Figure 1D-4M: Selecting the top surface of the Work Surface model for creating the Top Work Plane, 0.01-inch gap above the top edges of the lateral panels.

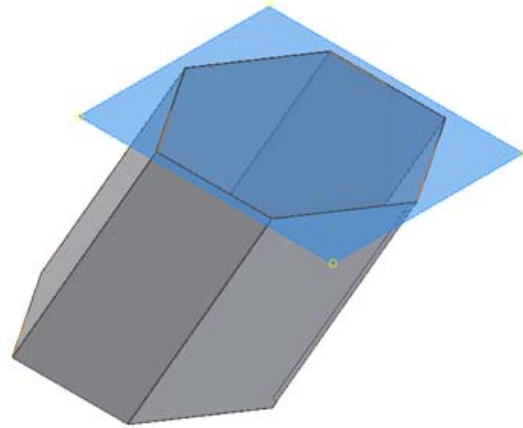


Figure 1D-4N: Selecting the Top Work Plane for the Top Face Sketch.

Step 4: Adding the top and base piece

Next, create a **Work Plane** feature for drawing the profile for the top piece. Select the **Work Plane** tool; click-select the top surface of the **Work Surface** model of the **Derived Part**, drag the mouse cursor upward; the **Offset** value in the pop-up text field increases from 0 to a certain number; type 0.01 (inch) for a tiny gap between the **Work Plane** where the top panel **Face** features will start and the top edges of the lateral pieces, so as to allow Inventor to generate the **Flat Pattern**; click the green check mark to create the new **Work Plane** feature (*Figure 1D-4M*), and rename it *Top Work Plane* in the **Model** panel.

Next, select the *Top Work Plane*; move the mouse cursor to the **Command Bar**; click the **Look At** button to switch to an orthographic normal view; click the **Sketch** button to start a new sketch (*Figure 1D-4N* and *Figure 1D-4P*), and rename it *Top Face Sketch* in the **Model** panel; use the **Zoom Window** tool to zoom in each corner of the top edges of the lateral panels; select the **Project Geometry** tool and click-select all straight inner edge lines to project them onto the sketch, then select all projected lines and go to the **Command Bar** to change their **Style** to **Normal**; next, use the **Fillet** tool to add 0.125-inch fillets to all corners where the projected straight lines meet (*1D-4Q*); the profile for the top panel is completed (*1D-4R*); click the **OK** button to exit the sketch.

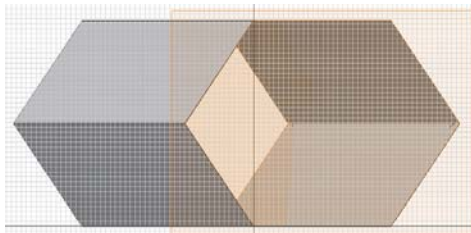
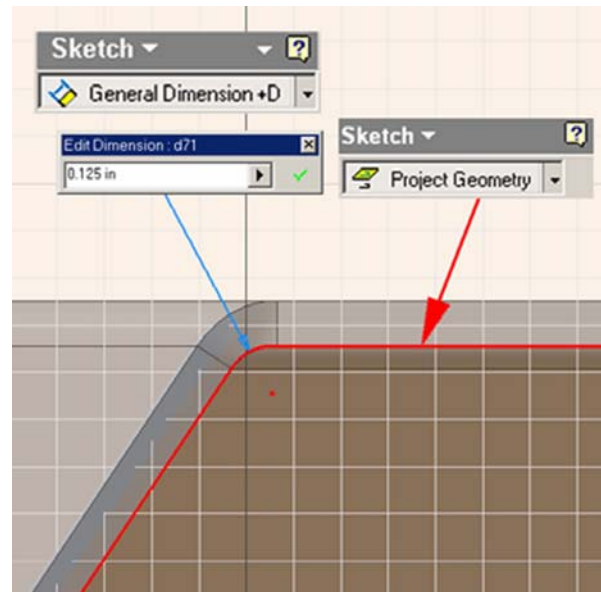


Figure 1D-4P: Switching to a normal view and starting the Top face Sketch on the Top Work plane.

Figure 1D-4Q: Projecting the straight inner edges of the lateral panels and applying 0.125-inch Fillets to all corners.



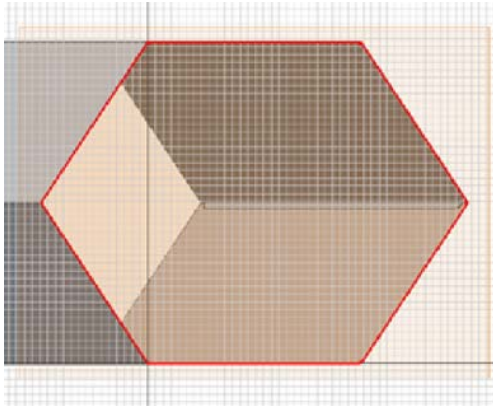


Figure 1D-4R: The top panel profile.

Next, select the **Face** tool; make sure that the green direction arrow points upwards or click the **Offset** button in the tool's dialog window to change the extrusion direction (Figure 1D-4S); click the **OK** button to create the top panel, and rename it *Top Face* in the **Model** panel. To connect the *Top Face* with one of the lateral panels (the *Lateral Face 1* in this case), select the **Bend** tool from the **Sheet Metal** tool bar; click-select the top edge of the *Top Face* and the outer top edge of the *Lateral face 1*, the green outline geometry appears; click the **OK** button to create the **Bend** feature (Figure 1D-4T), and rename it *Top Face Bend* in the **Model** panel.

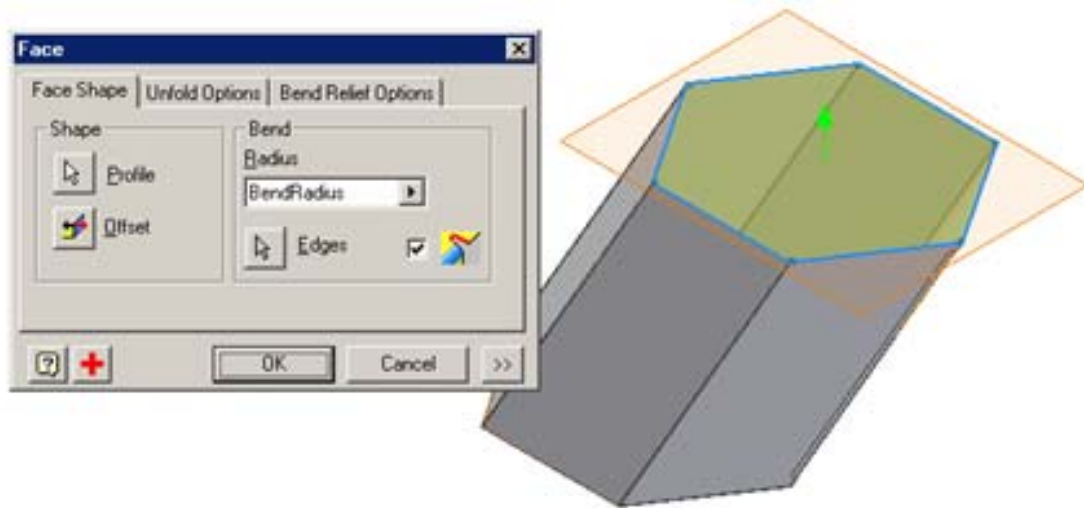


Figure 1D-4S: Creating the Top Face.

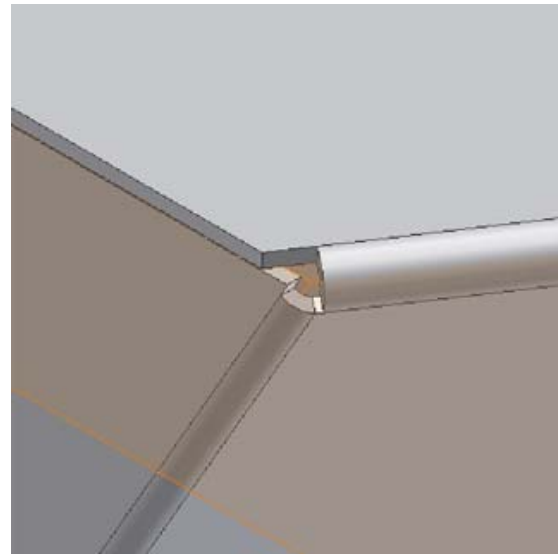
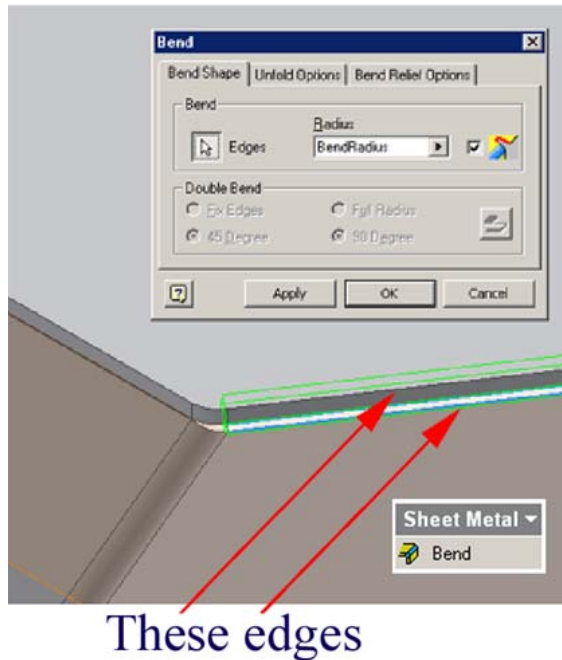


Figure 1D-4T: Selecting the top edge of the top panel and the outer top edge of the Lateral Face 1 panel to add the Bend feature (left). The Bend feature (right).

Next, use the same tools to create the *Base Face*. First use the **Rotate** tool to rotate the model to a convenient position where the base area can be clearly seen (*Figure 1D-4U*). Next, use the **Work Plane** tool to create a new **Work Plane** feature at a 0.1-inch **Offset** distance from the base surface of the **Derived Part**'s **Work Surface** model, and rename it *Base Work Plane* in the **Model** panel (*Figure 1D-4V*). Next, select the *Base Work Plane* to create the profile sketch for the base panel, and rename it *Base Face Sketch* in the **Model** panel (*Figure 1D-4W* and *Figure 1D-4X*). Next, use the **Face** tool to create the base panel (make sure that the green **Offset** direction arrow point downwards) and rename it *Base Face* in the **Model** panel (*Figure 1D-4Y*). Next, use the **Bend** tool to create a **Bend** connection between the *Lateral Face 1* and the *Base Face*, selecting the bottom edge of the *Base Face* and the outer bottom edge of the *Lateral Face 1* as **Edges** (*Figure 1D-4Z*), and rename it *Base Face Bend* in the **Model** panel.

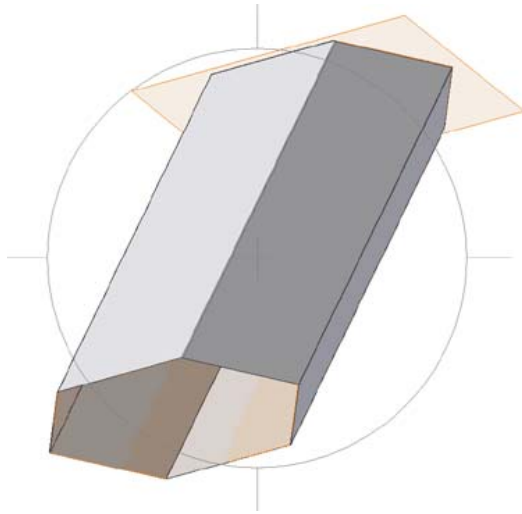


Figure 1D-4U: Rotating the model.

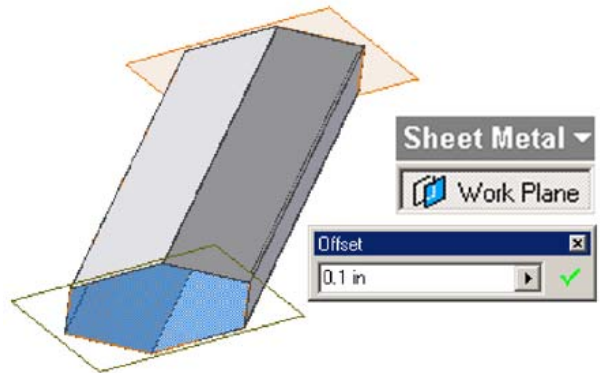


Figure 1D-4V: Creating the Base Work Plane at a 0.1-inch distance from the base surface of the Derived Part's Work Surface model.

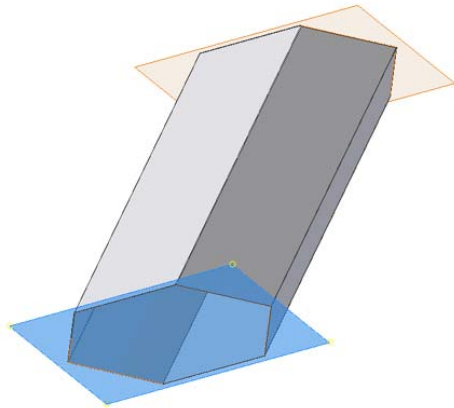


Figure 1D-4W: Selecting the Base Work Plane for the Base Face Sketch.

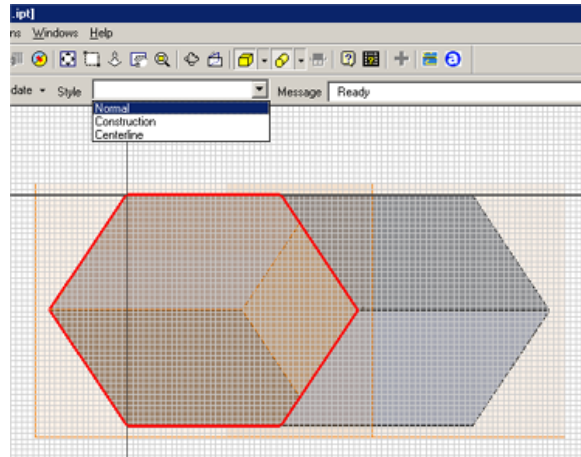


Figure 1D-4X: The Base Face Sketch.

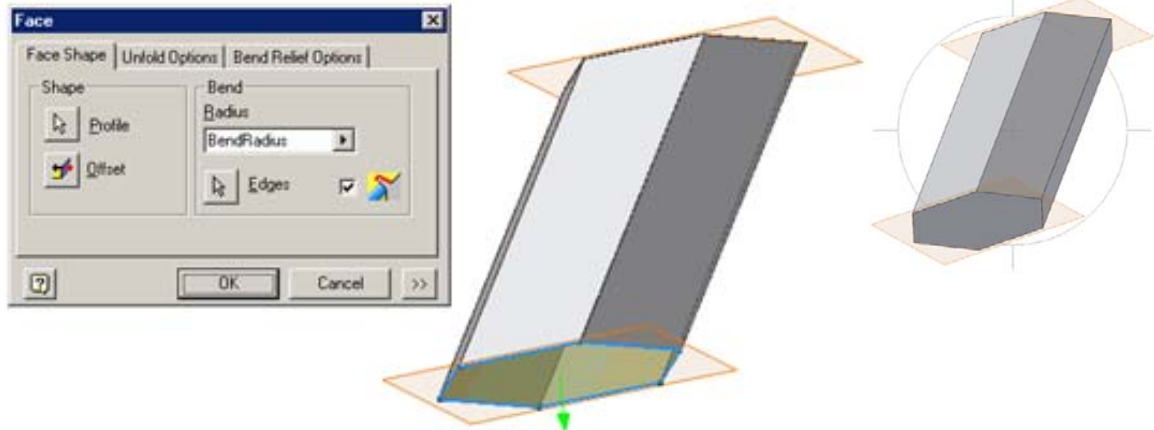


Figure 1D-4Y: Creating the Base Face.

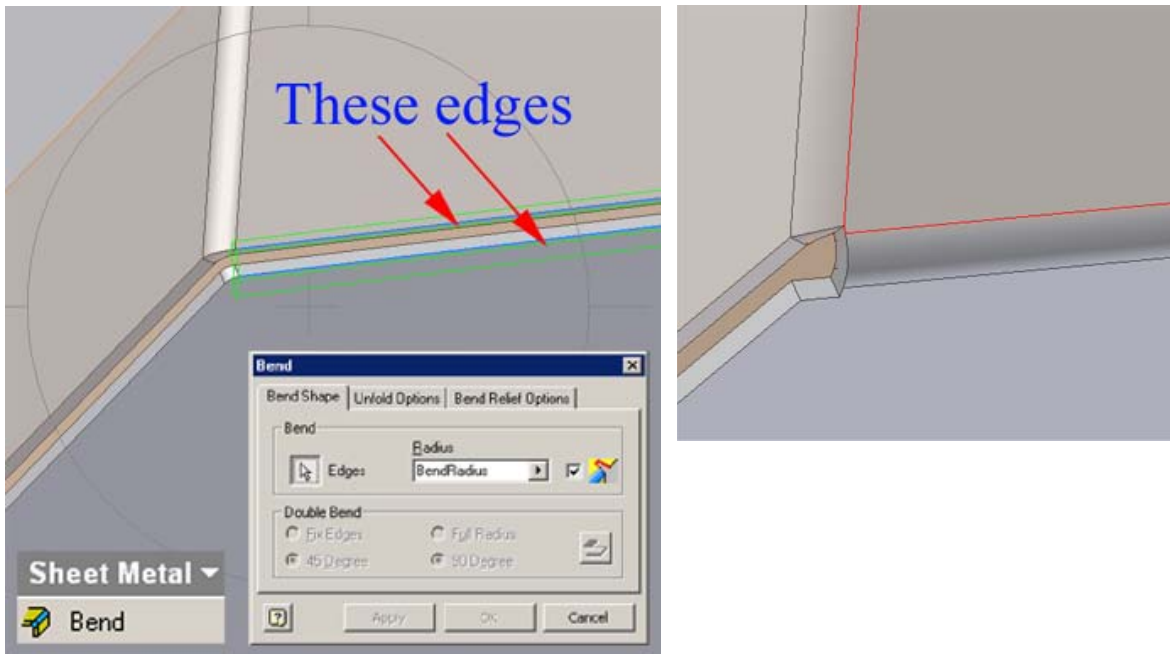


Figure 1D-4Z: Creating the Base Face Bend.

Step 5: Adding seams to the upper and lower edges of the lateral panels

The last step to complete this project is to add seams to the upper and lower edges of the lateral panels. We will create 10 pieces of seams with the **Flange** tool, or 5 pieces for the upper inner edges and 5 pieces for the lower inner edges of the *Lateral Face 2*, *Lateral Face 3*, *Lateral Face 4*, *Lateral Face 5* and *Lateral Face 6*, with the **Flange's** **Distance** set to 1.0 in (inch), both **Offset1** and **Offset2** set to 1.0 in (inch); for all of the

above **Flange** features, make sure that the green **Flip Offset** arrow points downward for the **Flange** seams on the upper inner edges of the lateral panels, and points upward for the **Flange** seams on the upper inner edges of the *Lateral Faces* (Figure 1D-5A and Figure 1D-5B), and that all **Flange** seams are extended inwardly towards the interior of the **Derived Part**. For the **Angle** of all **Flange** seams, delete the default Angle value, right-click inside the **Angle** text field for the shortcut menu and choose the **Measure** option; and then click-select the two surfaces (one being the outer surface of the lateral panel, the other being the outer surface of either the top panel or the base panel) to obtain the dihedral **Angles** of the **Flange** seams; use the **Rotate** tool to verify if the outline geometry of the **Flange** seams are parallel to either the *Top Face* or the *Base Face*; type 180- in front of the measured angle values if necessary (Figure 1D-5C). Rename the **Flange** seam features created on the upper inner edges of the *Lateral Face 2, 3, 4, 5* and *6* as *Lateral Top Flange 2, 3, 4, 5* and *6* (the number of **Flange** features matching the number of related **Face** features). Similarly, rename the **Flange** seam features created on the lower inner edges of the *Lateral Face 2, 3, 4, 5* and *6* as *Lateral Base Flange 2, 3, 4, 5* and *6*.

Next, click-select the outer surface of the *Lateral Face 2* feature, and click the **Flat Pattern** tool button on the **Sheet Metal** tool panel; the **Flat Pattern** window opens (Figure 1D-5D). All features of the model are listed in the **Model** panel (Figure 1D-5E).

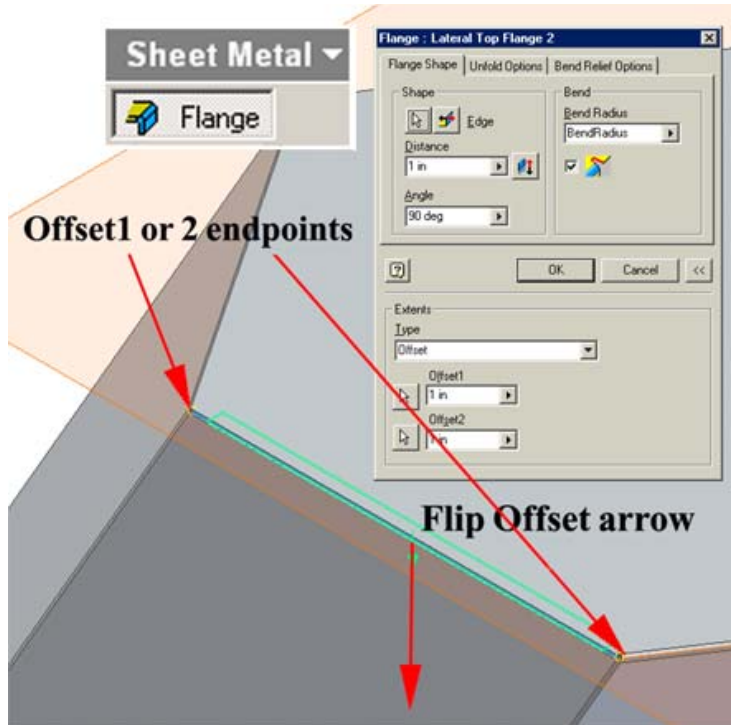


Figure 1D-5A: The Flip Offset green arrow pointing downwards for all Flange seams created on the upper inner edges of the lateral panels.

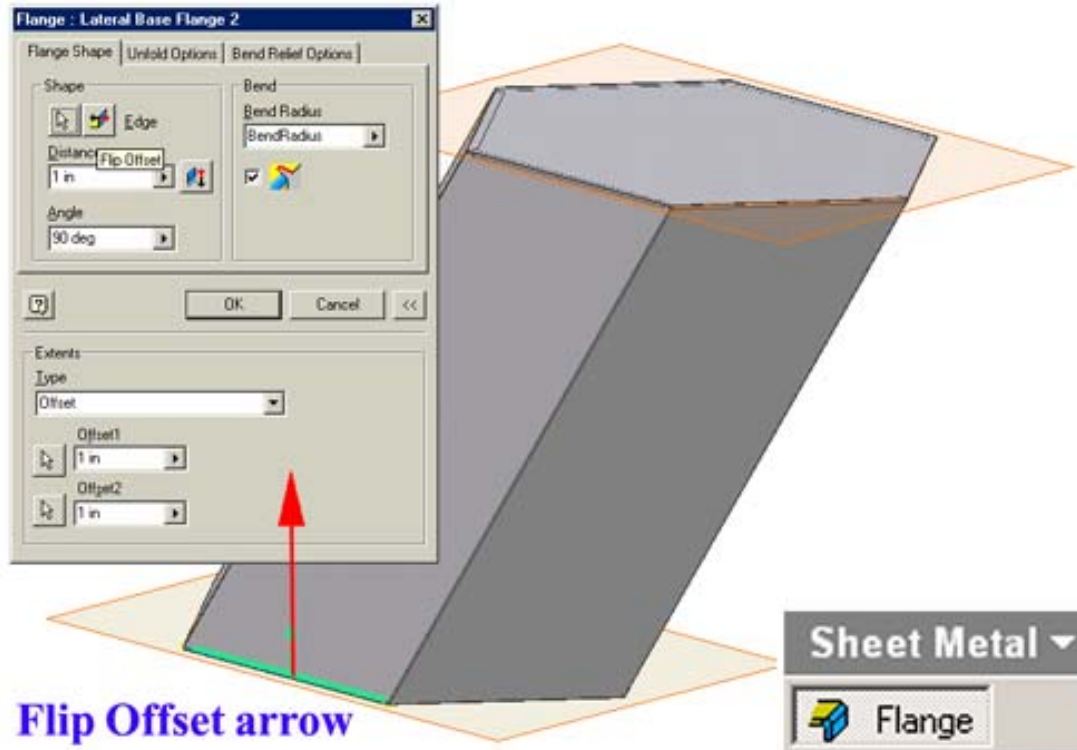


Figure 1D-5B: The Flip Offset green arrow pointing upwards for all Flange seams created on the lower inner edges of the lateral panels.

Congratulations!

In this Module, you have learned how to create a sheet metal part wrapping the space of an oblique prism with a hexagonal cross-section.

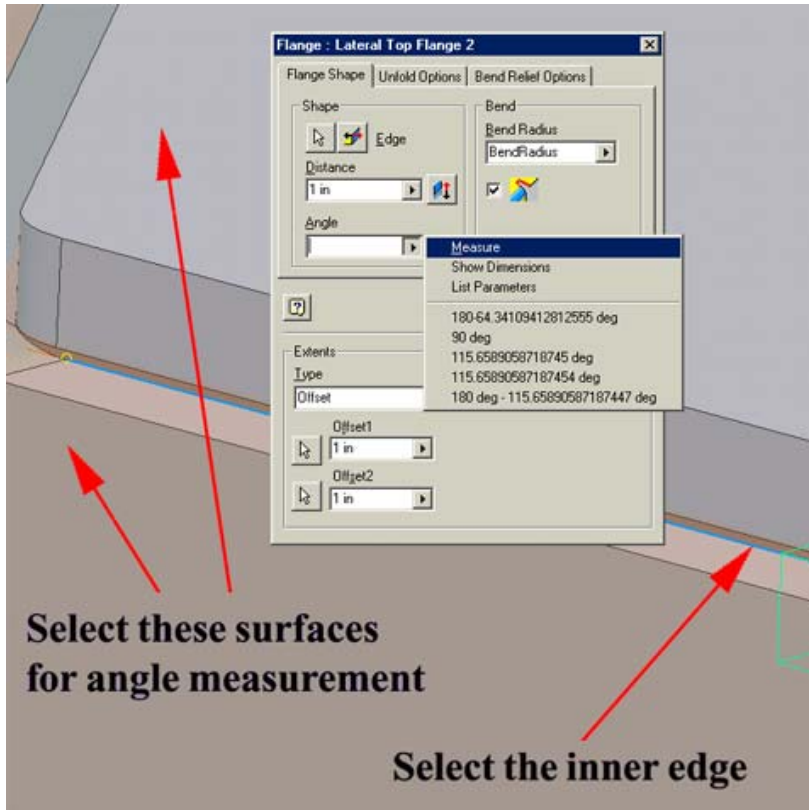


Figure 1D-5C: Deleting the default Angle value and using the Measure tool to obtain the Angle of the Flange seams.

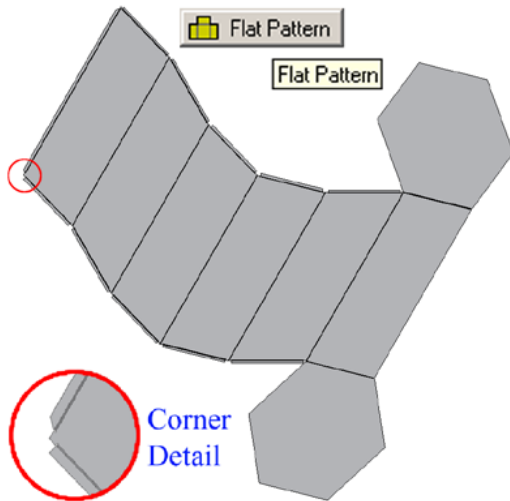


Figure 1D-5D: The Flat Pattern.

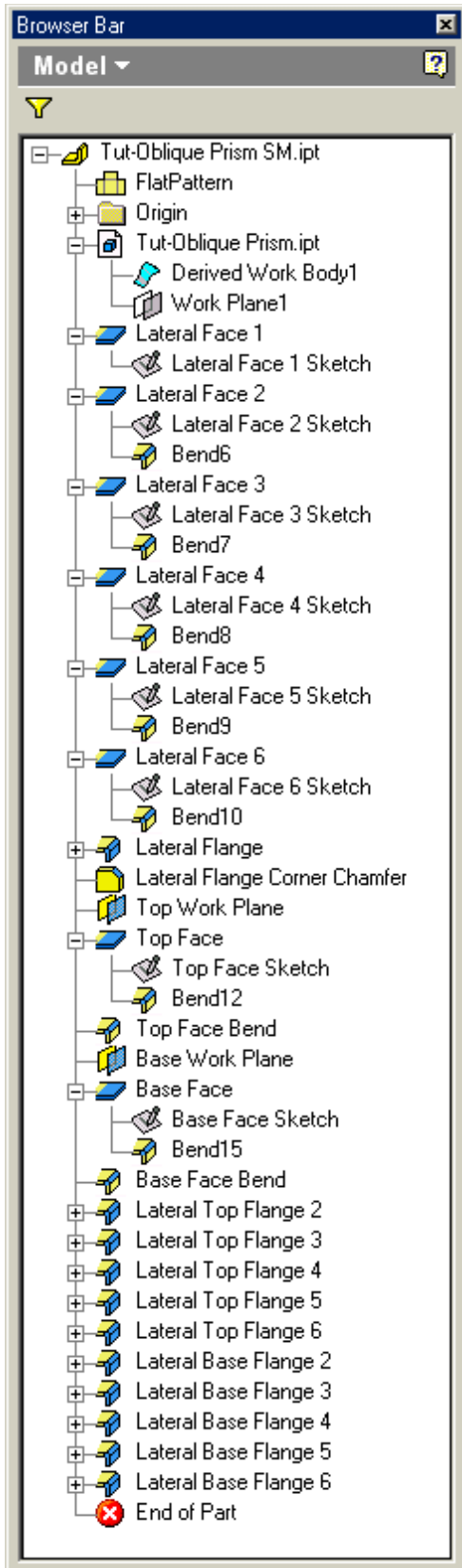


Figure 1D-5E: All features of the part listed in the Model panel.

Now, please proceed to the next Module.