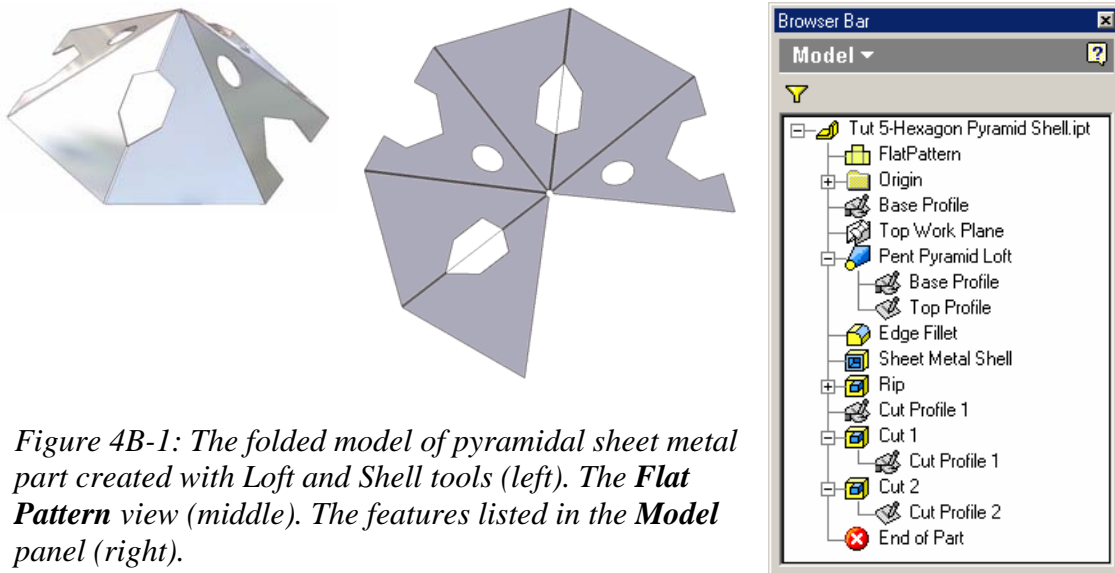


Module 4B: Creating Sheet Metal Parts Enclosing The 3D Space of Right and Oblique Pyramids With The Work Surface of Derived Parts

In *Module 4B*, we will learn how to create sheet metal parts enclosing right and oblique pyramids, using the **Work Surface of Derived Part** 3D solid models of right and oblique pyramid (regular or truncated), which have been created in *Module 4A*. These sheet metal parts shall be created with the **Face** tool, an Inventor tool from the **Sheet Metal** panel, which is specially created for sheet metal design, in another sheet metal file.

Like the development of a cone, the flat pattern of a right-axis pyramid (either regular, frustum or truncated), falls into the category of “radial-line” development, since all element lines “radiate” out of its central point called the vertex. On the other hand, the development of an oblique-axis pyramid is generally considered as a development by triangulation.



*Figure 4B-1: The folded model of pyramidal sheet metal part created with Loft and Shell tools (left). The **Flat Pattern** view (middle). The features listed in the **Model** panel (right).*

The creation of folded model and flat pattern of sheet metal parts wrapping the 3D space of pyramids can also be completed by using Inventor's generic 3D modeling tools such as Loft, Shell, as shown in *Figure 4B-1*. However, in this *Module*, the special **Sheet Metal** panel tool of **Face** with or without **Bend** option will be explored.

Section 1:

Creating A Sheet Metal Part Enclosing The 3D Space of A Regular Right-Axis Triangle-Based Pyramid

Step 1: Creating the 1st and 2nd Face features with projected sketch profiles

Launch Inventor, start a new **Sheet Metal (in).ipt** file under the **English** tab. Click the **Return** button to dismiss the default sketch; select the default *Sketch1* feature in the **Model** panel and delete it with the **Delete** key on the keyboard. Go to the **View**→**Isometric** menu for better visualization in 3D space. Click-select the **Derived Component** tool in the **Features** panel; the **Open** dialog window appears; select the *Tut-Right Triangular-Base Pyramid.ipt* file inside the *Tut-Pyramid* folder; and click the **Open** button (*Figure 4B-2A*); the **Derived Part** tool dialog window opens, and the 3D model of the *Tut-Right Triangular-Base Pyramid.ipt* file appears on the screen; click-select the **Body as Work Surface** option; the 3D body turns light orange; click the **OK** button (*Figure 4B-2B*); the **Derived Part Work Surface** model is created in the file as a reference; and its feature symbol with the name of the reference file is listed in the **Model** panel (*Figure 4B-6I*). Save the file as *Tut-Right Triangular-Base Pyramid Sheet Metal.ipt* in the same *Tut-Pyramid* folder. Save often throughout the entire project.

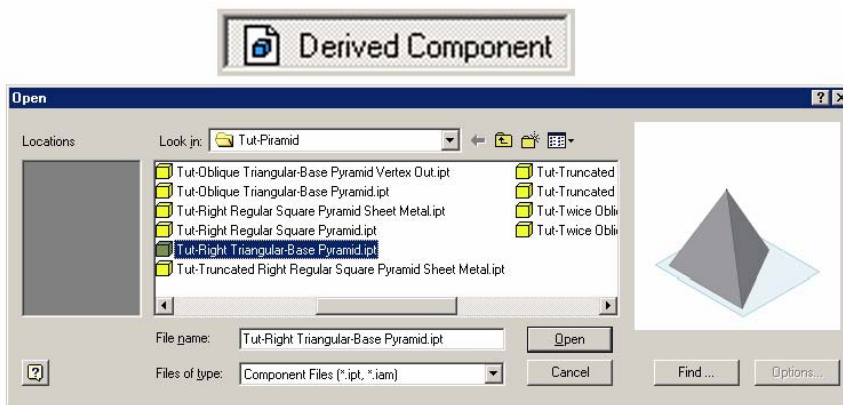


Figure 4B-2A:
The Derived Component tool and dialog window.

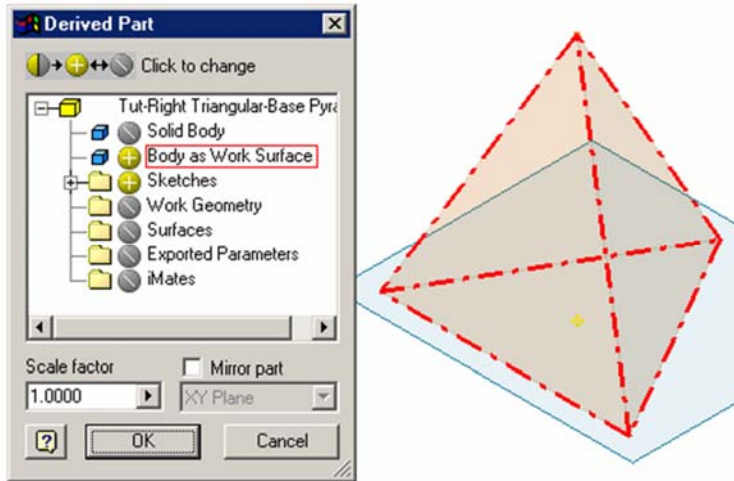


Figure 4B-2B: The **Derived Part** dialog window.

Next, click-select the first lateral surface of the pyramid (Figure 4B-2C); click the **Sketch** button to start a new sketch; rename the new sketch “*Surface 1 Profile*” in the **Model** panel; use the **Project Geometry** tool to project all edge lines of the first lateral surface onto the *Surface 1 Profile* sketch (Figure 4B-2D); go to the **Command Bar** to change the **Style** of all projected lines to **Normal** (Figure 4B-2E); in addition, to prevent problems of “open loop” when edge lines are projected onto a sketch, use the **Circle Center Point** tool to draw a large circle, use the **Extend** tool to extend all projected edge lines to the circle (Figure 4B-2F), and use the **Trim** tool to trim off all extended segments (Figure 4B-2G), and delete the circle with the **Delete** key on the keyboard (Figure 4B-2H); this method of preventing “open loop” problem will be called the “**Circle-Extend-Trim**” method in later *Modules*; next, click the **Return** button to exit the *Surface 1 Profile* sketch. Repeat the same procedures to create the sketch for the second lateral surface (Figure 4B-2I), and rename it “*Surface 2 Profile*” in the **Model** panel.

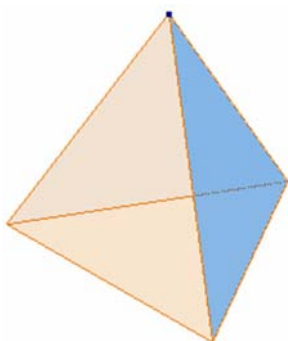


Figure 4B-2C: Selecting the first lateral surface.



Figure 4B-2D: Projecting the edge lines onto the “*Surface 1 Profile*” sketch.

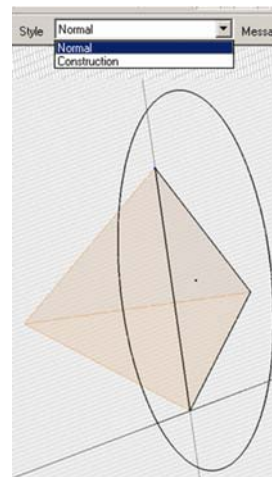


Figure 4B-2E: Change the **Style** of all projected edge lines to **Normal** and draw a circle.

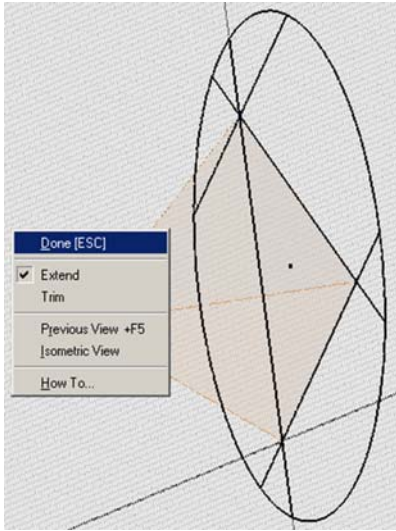


Figure 4B-2F: Extending all edge lines to the circle.

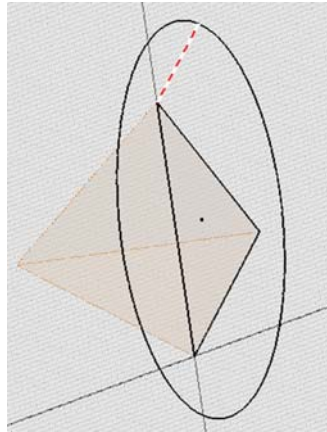


Figure 4B-2G: Trimming off all extended edge line segments.

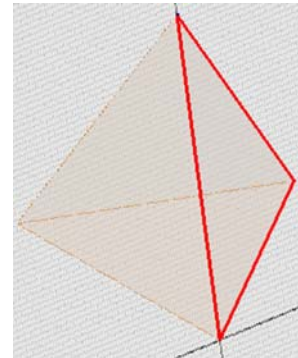


Figure 4B-2H: The completed sketch.

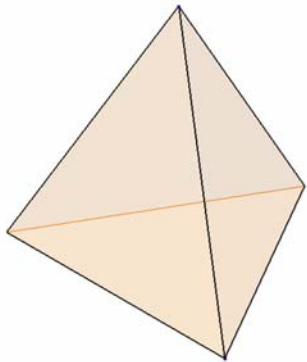


Figure 4B-2I: The “Surface 2 Profile” sketch for the second lateral surface.

Next, create the first and second lateral surface panels. First, click-select the **Style** tool from the **Sheet Metal** panel to set the thickness parameter for the sheet metal part; the **Sheet Metal Style** dialog window opens; accept the default **Thickness** value of 0.120 in and click the **Done** button to close the window (Figure 4B-3A). Next, use the **Face** tool with the two sketch outlines as Profiles to create the first and then the second lateral surface panels (Figure 4B-3B). Rename the **Face** features “Surface 1” and “Surface 2” in the **Model** panel. next, use the **Rotate** tool to rotate the 3D model to the position as shown in Figure 4B-3C. Use the **Zoom Window** tool to zoom in the top corner of the intersecting edge between the two lateral panels (Figure 4B-3D); it is obvious that Inventor automatically creates a **Bend** feature between the two lateral panels at their intersecting edge.

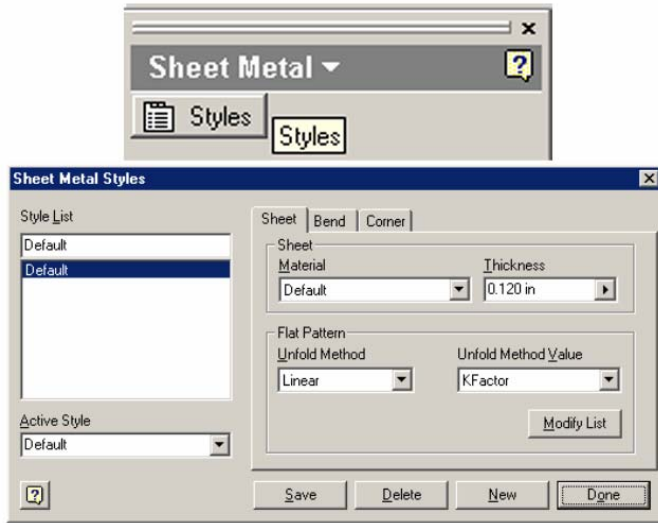


Figure 4B-3A: The Styles tool and the Sheet Metal Styles dialog window.

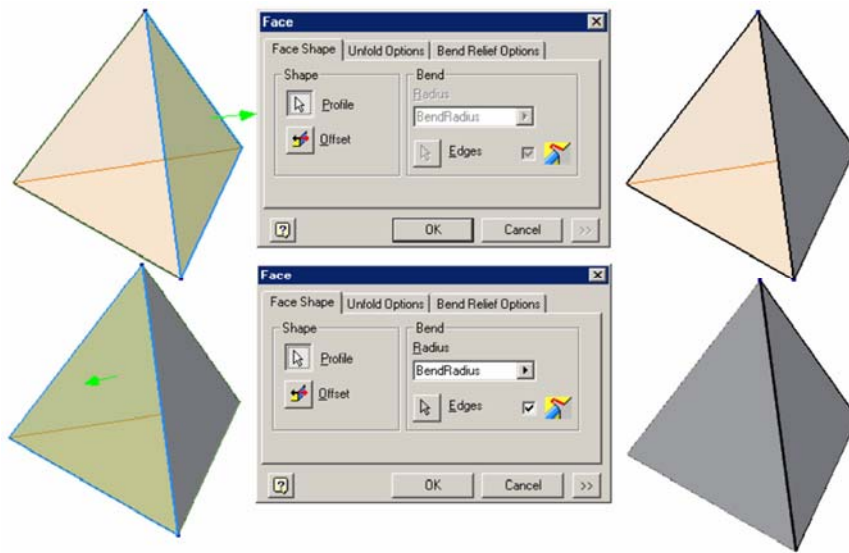


Figure 4B-3B: Creating the 1st (top) and 2nd (bottom) lateral surface panels.

Step 2: Creating the 3rd Face feature from a projected sketch profiles with modifications for an opening or gap in the sheet metal part

Use the **Rotate** tool to rotate the model again to a position where the remaining third lateral surface of the **Derived Part** can be viewed conveniently (Figure 4B-3E); click-select this surface and click the **Sketch** button for a new sketch (Figure 4B-3F); rename the new sketch “Surface 3 Profile” in the **Model** panel.



Figure 4B-3C:
Rotating the model.

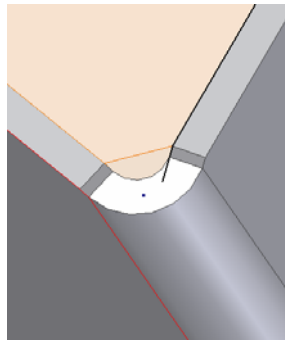


Figure 4B-3D:
Zoom in the corner.

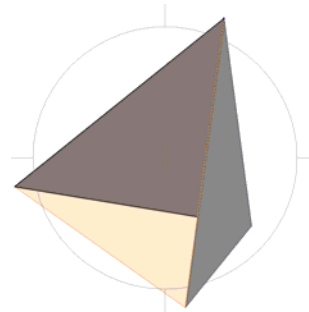


Figure 4B-3E:
Rotating the model again.

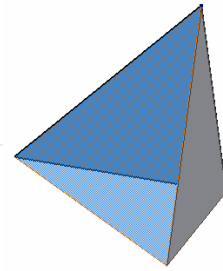


Figure 4B-3F:
Selecting the third surface.

To create flat pattern out of sheet metal parts in Inventor, a gap between the first and the last lateral panels is needed; therefore, some modifications to the projected profile need to be done. In the *Surface 3 Profile* sketch, use the **Project Geometry** tool to project the edge line between the first and the last lateral surfaces (Figure 4B-3G); use the **Offset** tool to offset this projected line, and use the **General Dimension** tool to apply a 0.001 gap dimension between the projected and the offset lines (Figure 4B-3H); use the **Zoom Window** tool to zoom in the corner and use the **Delete** key on the keyboard to delete the original projected edge line as shown in Figure 4B-3I; next, with the help of the **Zoom Window** tool (Figure 4B-3K), project the remaining edge lines onto the sketch and change the **Style** of these projected lines from **Reference** to **Normal**; next, repeat the “**Circle-Extend-Trim**” method or procedures of drawing a large circle, extending projected lines to the circle (Figure 4B-3L) and trimming (Figure 4B-3M), and finally deleting the circle with the **Delete** key on the keyboard, so as to prevent “open loop” problem and to complete the profile as a closed loop (Figure 4B-3N).

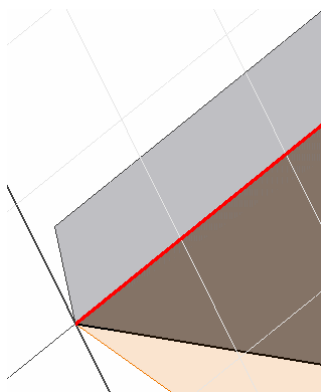


Figure 4B-3G: Projecting the edge between the first and third surfaces.

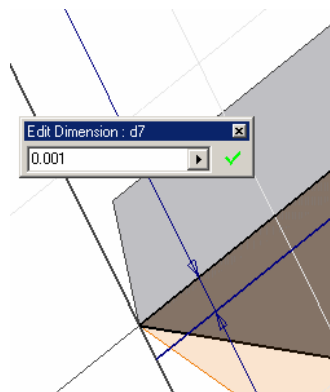


Figure 4B-3H: Offsetting a parallel line and apply a 0.001 dimension.

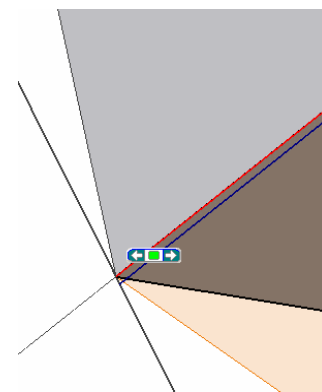


Figure 4B-3I: Zooming in and delete the original projected edge line.

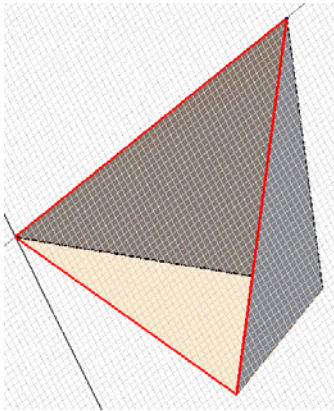


Figure 4B-3J: Projecting all remaining edge lines.

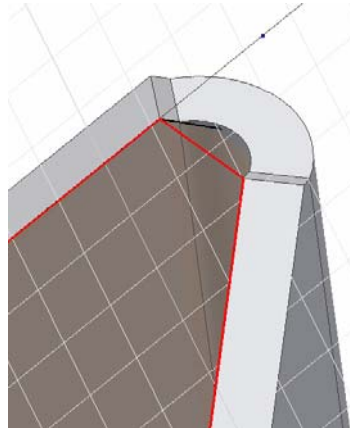


Figure 4B-3K: Using the **Zoom Window** tool.

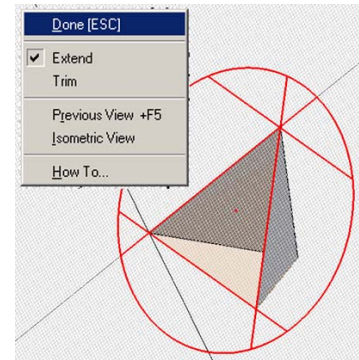


Figure 4B-3L: Drawing a large circle and extending the projected lines to it.

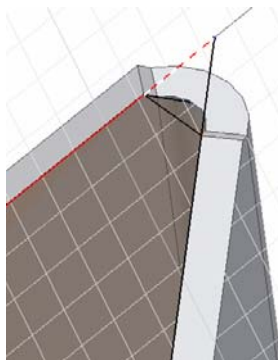


Figure 4B-3M: Trimming excessive line segments and deleting the circle.

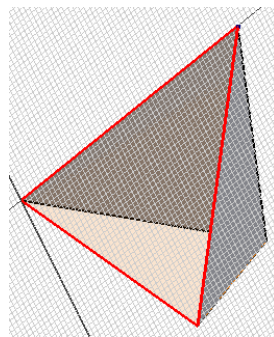


Figure 4B-3N: The complete closed loop profile for the third lateral panel.

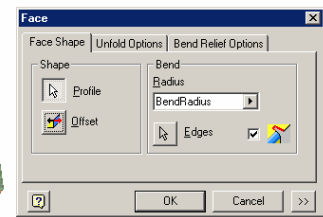
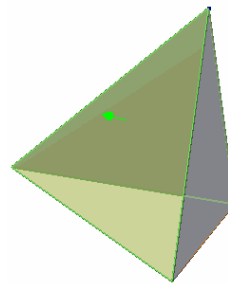


Figure 4B-3O: Using the **Face** tool to complete the basic sheet metal part enclosing a triangle-base right pyramid.

Next, use the **Face** tool to complete the basic sheet metal part enclosing the 3D space of a triangle-base right pyramid (Figure 4B-3O), and rename the new **Face** feature “Surface 3” in the **Model** panel. Use the **Rotate** and **Zoom Window** tools to view how Inventor creates the corner relief at the corner of intersection among the three lateral panels (Figure 4B-3P).

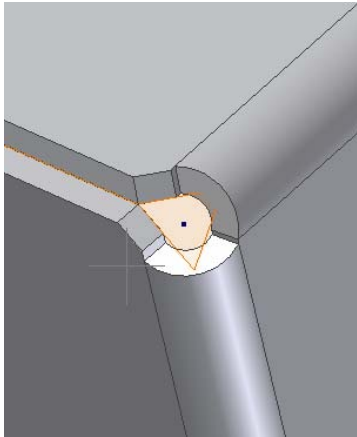


Figure 4B-3P: The corner of intersection among the three lateral panels.

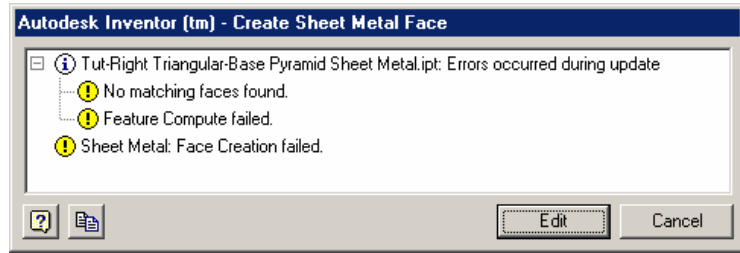


Figure 4B-3Q: The “No matching faces found” error message in the Create Sheet Metal Face dialog window.

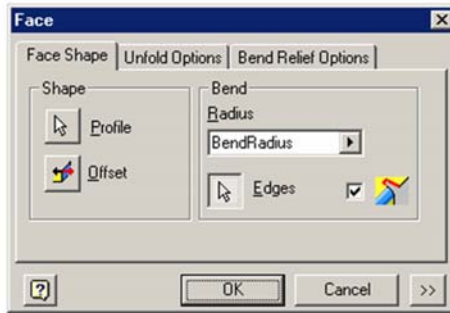
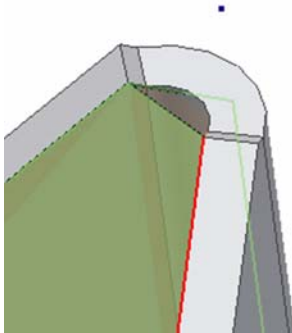


Figure 4B-3R: Selecting an appropriate edge for the Bend feature.

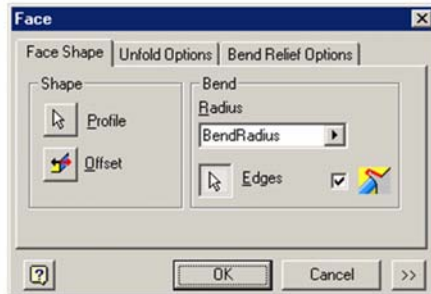
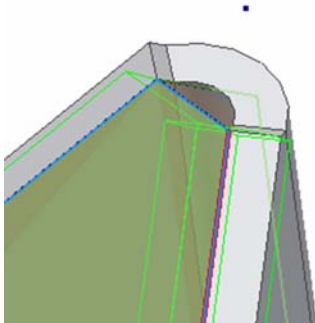


Figure 4B-3S: The green outlined Bend feature geometry indicator

Occasionally, when adding a new **Face** feature, Inventor will open the **Create Sheet Metal Face** dialog window with a “**No matching faces found**” error message (Figure 4B-3Q); this is because there is ambiguity on which matching congruent face to choose for adding a **Bend**; if this occurs, then click the **Edges** button in the Face dialog window, and click-select the appropriate edge (Figure 4B-3R); the green outlined **Bend**

feature geometry indicator will appear (*Figure 4B-3S*); click the **OK** button to create the **Face** feature. If the intended design is geometrically possible, then the **Face** feature will be created; otherwise, space for corner relief needs to be increased, by one of the two following methods:

1. Changing the sketch profile. Select the profile sketch in the **Model** panel; right-click for the shortcut menu and choose the **Edit Sketch** option; in the profile sketch that opens, click-select, hold and drag the short edge line downward to increase space for corner relief (*Figure 4B-4A*), or use the **General Dimension** tool to apply a larger dimension for the gap; next, click the **Return** button to exit the sketch. Next, try using the **Face** tool to create the needed face (*Figure 4B-4B*); once the needed space for corner relief is provided, the **Face** with **Bend** feature can be created;
2. Modifying the **Derived Part**. Click- Select the **Derived Part** feature in the **Model** panel; right-click for the shortcut menu and choose the “**Open Base Component**” option to open the file that originates the **Derived Part** (*Figure 4B-4C*); go to the **Windows**→**Arrange All** menu to arrange both the **Derived Part**-originating file and the sheet metal file within the same screen (*Figure 4B-4D*); switch between the sheet metal part file and the **Derived Part**-originating file by clicking respective screen area; edit the features in the **Derived Part**-originating file first; then, switch to the sheet metal part file and click the **Update** button on the **Command Bar** to update the placed **Derived Part**'s 3D model in the sheet metal file, and edit the related features in the sheet metal part file accordingly.

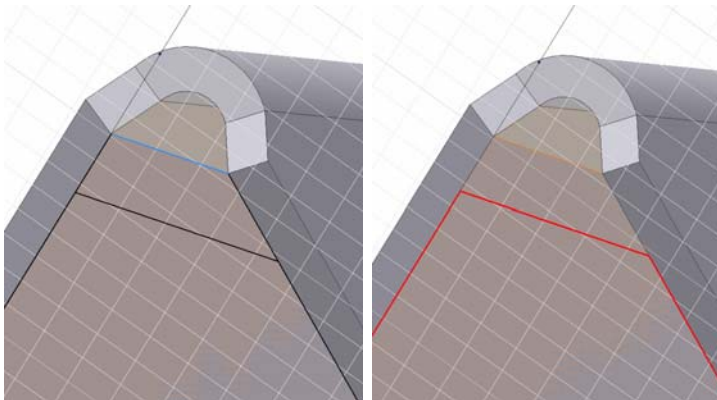
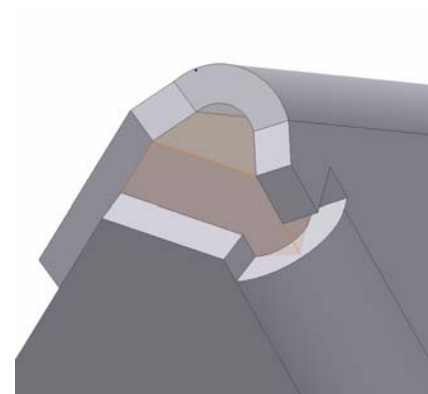


Figure 4B-4A: Editing the profile sketch.



*Figure 4B-4B: Creating the **Face** feature with **Bend** once enough corner relief is provided.*

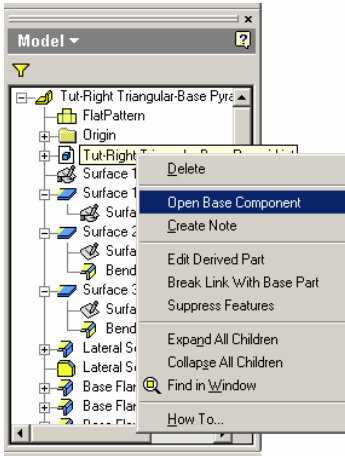


Figure 4B-4C: Choosing the “Open Base Component” option in the shortcut menu to open the file that originates the Derived Part.

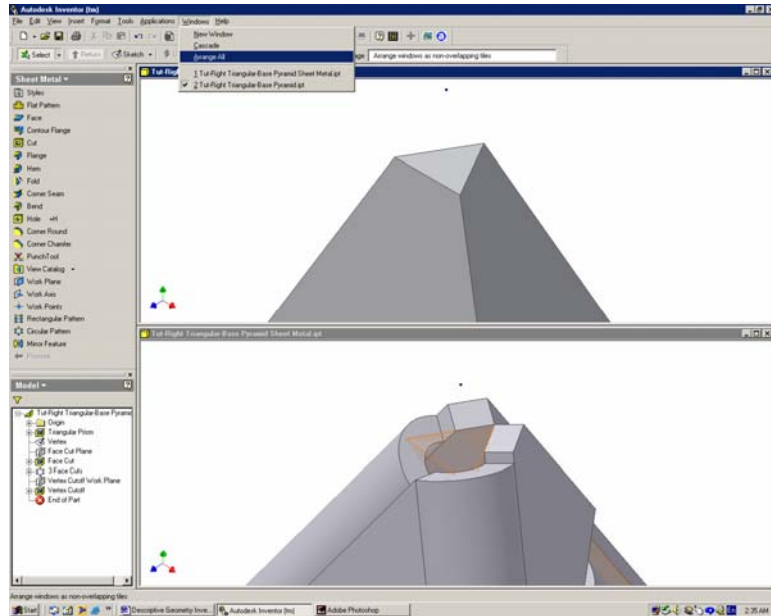


Figure 4B-4D: The Derived Part-originating file (top) and the sheet metal file (bottom) within the same screen.

Step 3: Adding a seam Flange feature connecting the first and the last Face panels

Next, add a seam to the lateral edge of the first lateral **Face** panel so that it will overlay the lateral edge area of the third lateral **Face** panel. First go to the **Tools**→**Document Settings** menu to open the **Document Settings** dialog window, and select the highest accuracy parameter (0.11245) in the pull-down menu under **Angular Dim Display Precision** heading in the **Modeling Dimension Display** section (*Figure 4B-5A*); click the **OK** button to close the dialog window; then go to the **Tools** pull-down menu and select the **Measure Angle** tool, click-select first and third lateral **Face** panel to measure the dihedral angle; it reads 67.3803 deg in the **Angle** window (*Figure 4B-5B*).

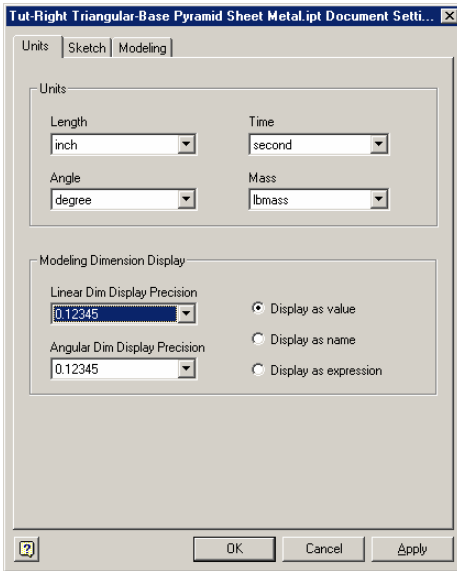


Figure 4B-5A: The **Document Settings** dialog window.

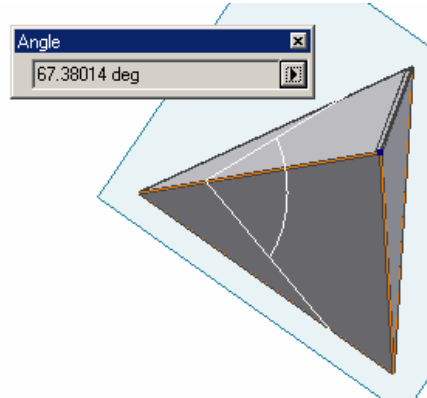


Figure 4B-5B: Measuring the dihedral angle.

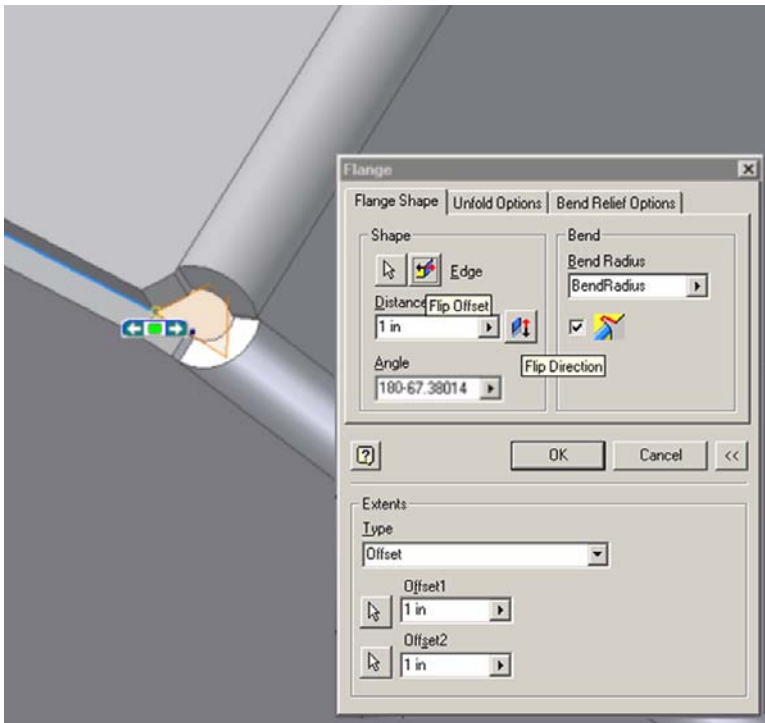


Figure 4B-5C: Selecting the inner lateral edge of the first lateral panel.

Next, select the **Flange** tool from the **Sheet Metal** panel, use the **Zoom Window** tool to zoom in the corner of intersection among the three lateral panels; click-select the

inside lateral edge of the third lateral panel (it turns blue); type 180-67.3803 (degrees) in the **Angle** text field (in this case, the bending angle = $180^\circ - \text{dihedral angle}$); accept the 1 in default value for **Distance**; click the >> button at the lower-right corner of the dialog window to open the **Extents** section; select **Offset** in the pull-down menu under the **Type** heading; type 1 (inch) in both **Offset1** and **Offset2** text field; click any one of the two arrow button on the left of the text field and move the mouse cursor closer to one endpoint of the blue selected edge line and click once at the appearance of the outlined yellow circle snap indicator (*Figure 4B-5C*); move the cursor closer to the other endpoint of the blue edge line and click once at the appearance of the outlined yellow circle snap indicator; the green outlines of the flange's geometry appear (*Figure 4B-5D*); click the **Flip Offset** and then the **Flip Direction** button in the **Flange** dialog window to make sure that the outlined geometry indicator of the new **Flange** feature is shrinking inward and pointing towards the third lateral panel (*Figure 4B-5D*); use the **Rotate** tool to rotate the model to a position where the flange's geometry can be seen inside the third panel (instead of overlapping it) as shown in *Figure 4B-5E*; if not, then click the **Flip Offset** button to make correction; next, use the **Rotate** tool to rotate the model to a position where the flange's geometry can be viewed in its entire length, to make sure that its end portion closer to the third lateral panel does not extend beyond the third lateral **Face** panel, as shown in *Figure 4B-5F*, otherwise, increase the **Offset1** or **Offset2** value (sometime, increase the **Offset** value first to avoid overlapping, then apply a large **Corner Chamfer**, then edit the **Flange** feature's **Offset** value to bring the **Flange** feature closer to the end of the selected edge if needed, as a "work-around" method); click the **OK** button to create the **Flange** feature; and rename the new **Flange** feature *Lateral Seam* in the **Model** panel. Next, select the **Corner Chamfer** tool to add 0.75 (inch) chamfers to both ends of the *Lateral Seam* Flange feature (*Figure 4B-5G*).

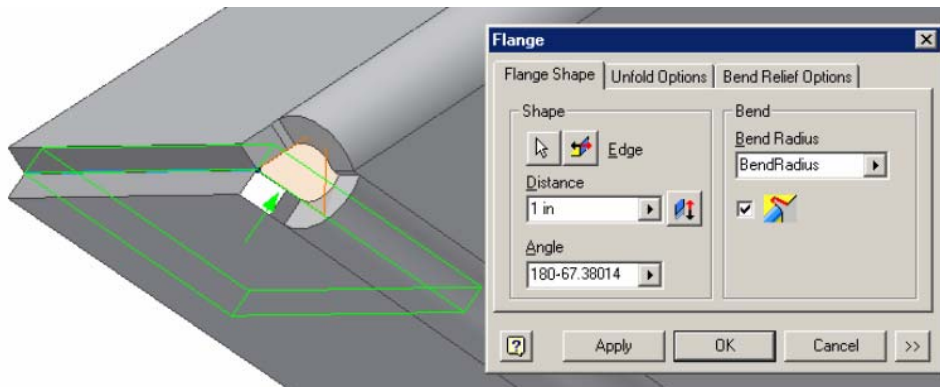


Figure 4B-5D: The green outlined flange geometry.

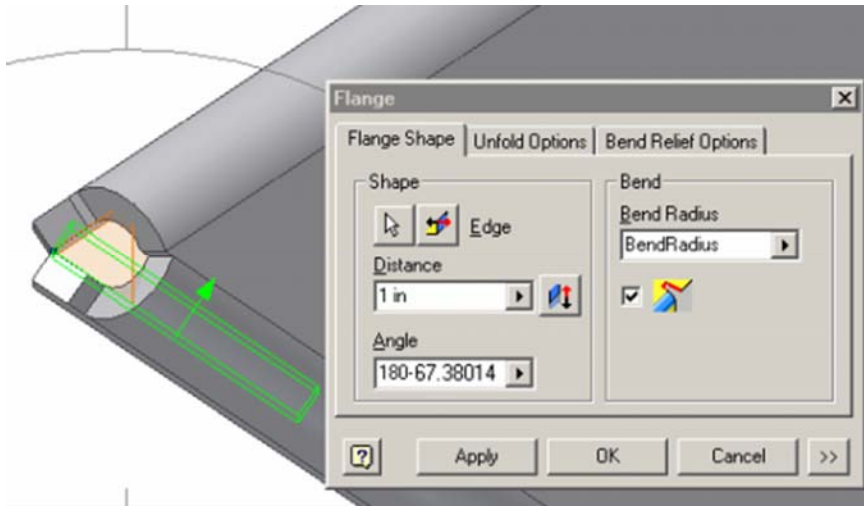


Figure 4B-5E: Checking against overlapping.

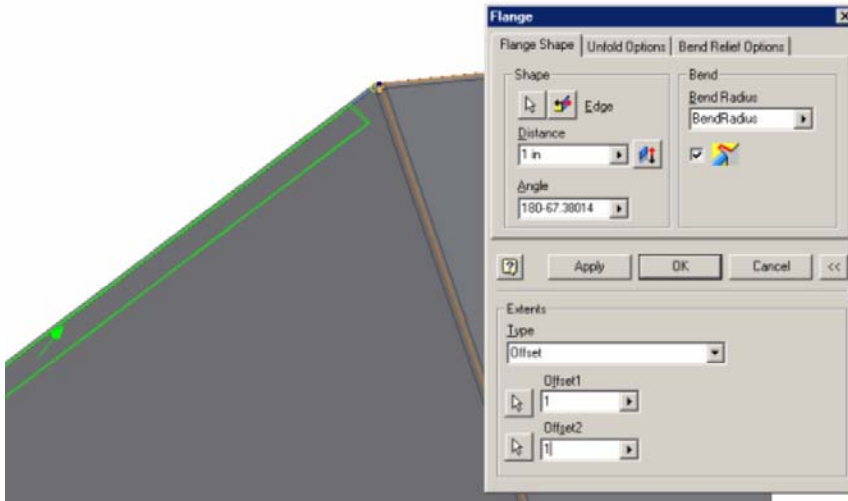


Figure 4B-5F: Checking problem of overlapping.

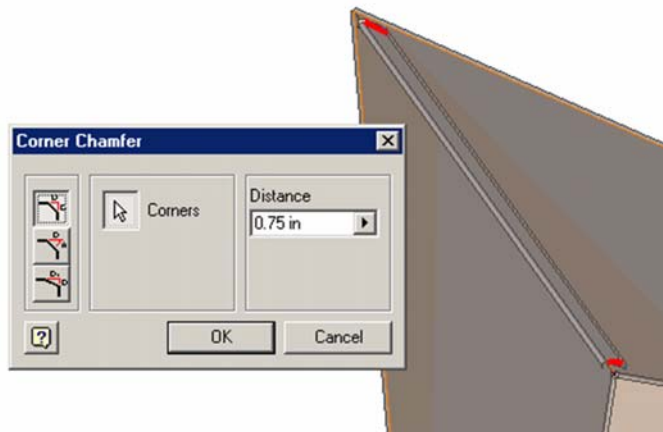


Figure 4B-5G: Adding **Corner Chamfer** features to the Lateral Seam Flange.

Step 4: Adding a Flange features connecting the sheet metal part with a level ground

Next, create three flanges at the base edges to allow the sheet metal part to be attached to a level ground. First, turn on the **Visibility** of the **XZ Plane** in the **Model** panel, then go to the **Tools** menu and select the **Measure Angle** tool to measure the dihedral angle between the **XZ Plane** and anyone of the lateral **Face** panels (since this is a right pyramid with a equilateral triangular base, all dihedral angles are equal and only one measurement is needed), the **Angle** text field reads 106.10101 deg, as shown in *Figure 4B-6A*. Next, with the help of the **Rotate** and **Zoom** tools for selecting the correct edges, select the **Flange** tool with 1 in **Distance**; type 180-106.10101 deg in the **Angle** text field; choose **Offset** as **Type** with 1 (inch) for both **Offset1** and **Offset2** in the **Extents** section; click-select the outer base edge, choose an upward **Flip Offset** direction, and an outward **Flip Direction** (*Figure 4B-6B*); click the **Apply** button to create the first base flange without closing the **Flange** tool's dialog window; continue to create the second and third base flange in the same manner, except that in creating the last flange (the third), clicking the **OK** button instead of the **Apply** button, so as to exit the **Flange** tool. Rename the new **Flange** features "*Base Flange 1*," "*Base Flange 2*," and "*Base Flange 3*" in the **Model** panel. Use the **Rotate** tool with the **Common Views [SPACE]** option selected from the tool's right-click shortcut menu, rotate the model to the front and side views, to check if the base flanges appear horizontal (*Figure 4B-6D*).

Next, use the **Corner Chamfers** tool to add 0.75 (inch) chamfers to both ends of all *Base Flange 1*, 2, and 3 features (*Figure 4B-6E*).

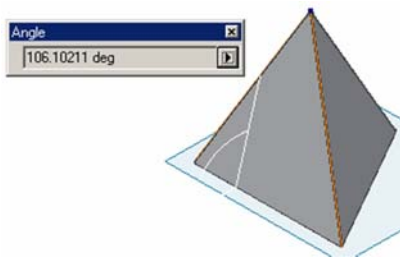


Figure 4B-6A: Measuring the dihedral angle.

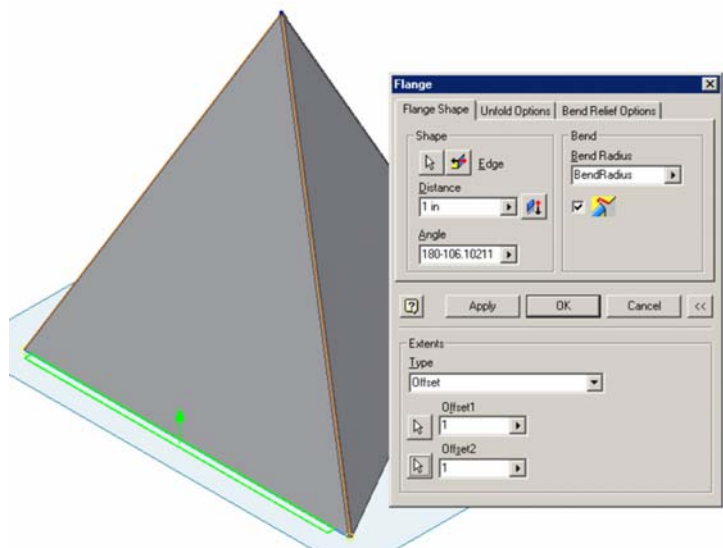


Figure 4B-6B: The Flange tool's dialog window.

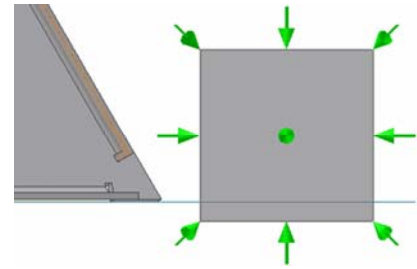
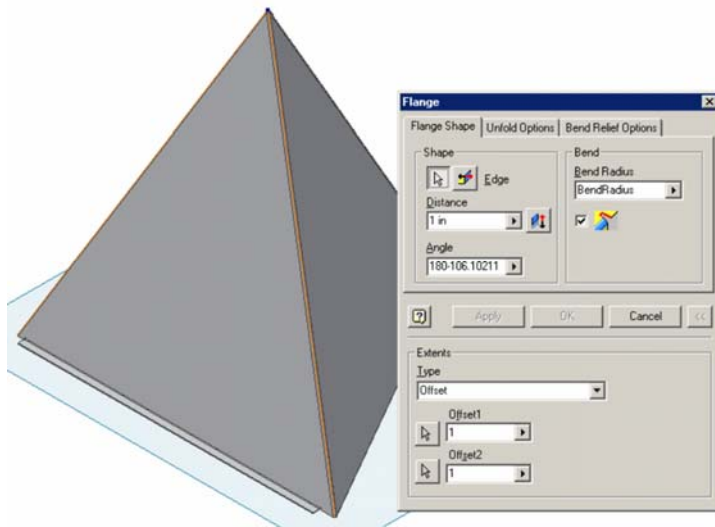


Figure 4B-6D: Checking horizontality of the base flanges.

Figure 4B-6C: Continuing to create the other two flanges.

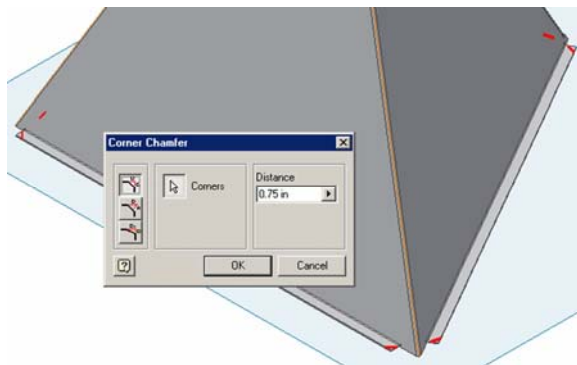


Figure 4B-6E: Adding **Corner Chamfers** to base flanges.

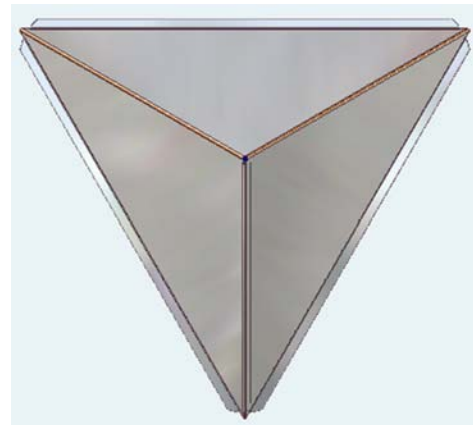


Figure 4B-6F: The top view.

The sheet metal part is completed (Figure 4B-6F). Click-select the outer surface of the 2nd **Face** panel (Figure 4B-6G) and then select the Flat Pattern tool to generate the **Flat Pattern** view (Figure 4B-6H). The features of the sheet metal part are listed in the **Model** panel (Figure 4B-6I).

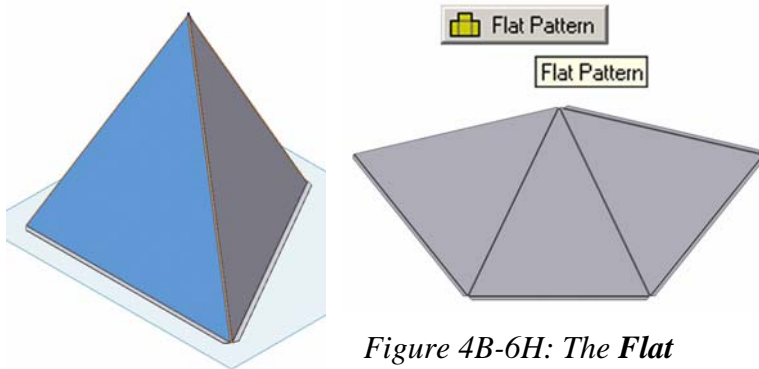


Figure 4B-6G: The isometric view with the 2nd Face panel selected.

Figure 4B-6H: The Flat Pattern view.

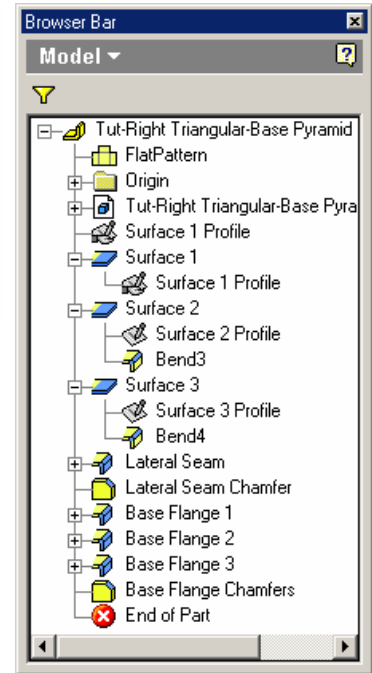


Figure 4B-6I: The features listed in the Model panel.

Section 2:

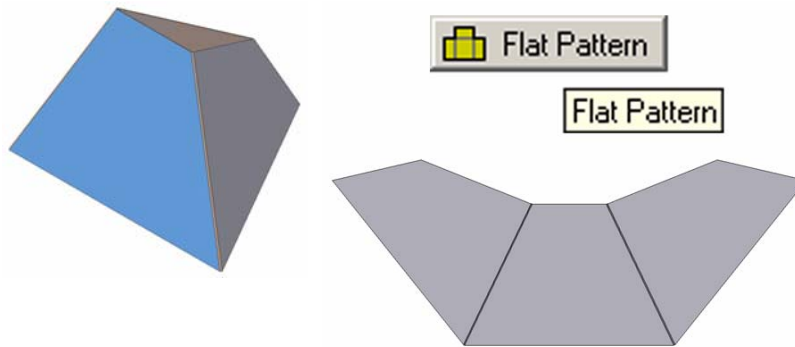
Creating A Sheet Metal Part Enclosing The 3D Space of A Truncated Right-Axis Triangle-Based Pyramid

In this part of the *Module 4B*, we will learn how to create an Inventor sheet metal part enclosing the 3D space of a truncated right pyramid with a cover for the truncated top surface; and we will use the techniques explored in the previous part of the *Module 4B*, and explore some “work-around” techniques on using the **Work Plane** tool in the solution of sheet metal design problems with Inventor that involves a cover for an inclined surface as well as problems of “self-intersecting” surface panels. We will use the previously created *Tut-Truncated Right Triangular-Base Pyramid.ipt* file as a Derived Part to create a sheet metal part. The step-by-step procedures are explained as follows.

Step 1: Creating the three lateral Face panels

Launch Inventor, start a new **Sheet Metal (in).ipt** file under the **English** tab. Click the **Return** button to dismiss the default sketch; select the default *Sketch1* feature in the **Model** panel and delete it with the **Delete** key on the keyboard. Go to the

View→Isometric menu for better visualization in 3D space. Next, use the same techniques as explored in the previous part of the Module 4B to place the 3D model of the *Tut-Truncated Right Triangular-Base Pyramid.ipt* file as a **Work Surface**, and create all lateral **Face** panels, lateral and base seam **Flange** features with **Corner Chamfers**, with the same parameter setups as in the previous part (*Figure 4B-7A*). Select the outer surface of the 2nd **Face** panel and then select the **Flat Pattern** tool to create a **Flat Pattern** view (*Figure 4B-7A*). Save the file as *Tut-Truncated Right Triangular-Base Pyramid Sheet Metal.ipt* in the same *Tut-Pyramid* folder. Save often throughout the entire project.



*Figure 4B-7A: The completed lateral **Face** panels of the sheet metal part enclosing the 3D space of a truncated right pyramid. Isometric view (left). **Flat Pattern** view (right).*

Step 2: Creating the top Flange features from two lateral Face panels to connect and support the top Face panel

Next, create the two top **Flange** features from the two lateral **Face** panels on the left and right hand sides, so as to connect and support the top **Face** panel to be created. First, go to the **Tools** menu and choose the **Measure Angle** tool to measure the dihedral angle between the top **Work Surface** of the placed **Derived Part** and one of the lateral panels on the side (in this case, the right one, as shown in *Figure 4B-8A*). Next, click-select the **Flange** tool; in the **Extents** section, choose **Offset** as **Type** (the other option is **Width**. Edge option can not be used in this case because of potential problem of self-intersecting), and type 1 (inch) in both **Offset1** and **Offset2** text fields; click-select the respective arrow buttons in the dialog window, and the inner edge of the lateral panel (it turns blue) and the two endpoints with the help of outlined yellow circle indicator (*Figure 4B-8B*), before clicking the **Apply** button, using the **Rotate** and **Zoom** tools to rotate the model to needed positions and to enlarge the view, so as to ensure that the direction of the flange (*Figure 4B-8C*) and its position relative to the ends of the lateral panel are correct, and that there is no problem of self-intersecting, and that appropriate space for the flange on the opposite lateral panel exists (*Figure 4B-8D*); after a visual check-up, it is obvious that a distance between the end of the flange and the corner of the lateral panels opposite

the one that connects the top panel needs to be increased; therefore, increase it to 3 (inches) in the Flange tool's dialog window and; additional visual checkup indicates that it will work; therefore, click the **OK** button to complete the first top flange; and rename the new **Flange** feature “*Top Flange 1*” in the **Model** panel. Next, use the **Rotate** tool to rotate the model to the bottom view so that the first top flange can be clearly seen, and add a 0.75-inch **Corner Chamfer** to both ends of the *Top Flange 1* (Figure 4B-8E); rename the new **Corner Chamfer** features as “*Top Flange 1 Chamfers.*” Repeat the same steps to create the top flange with chamfers from the other lateral **Face** panel and complete the sheet metal part (Figure 4B-8F); rename the new features “*Top Flange 2*” and “*Top Flange 2 Chamfers.*”

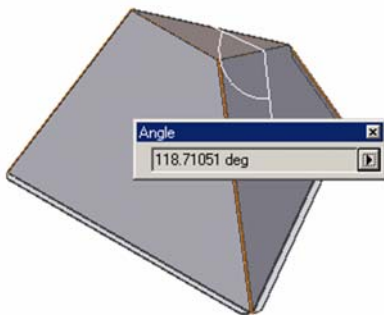


Figure 4B-8A: Measuring the dihedral angle.

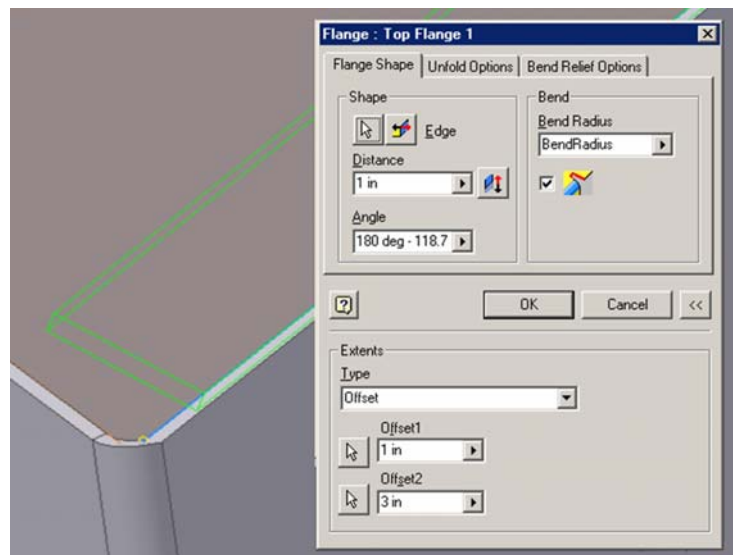


Figure 4B-8B: Selecting the inner edge and the *Offset1* and *Offset2* endpoints.

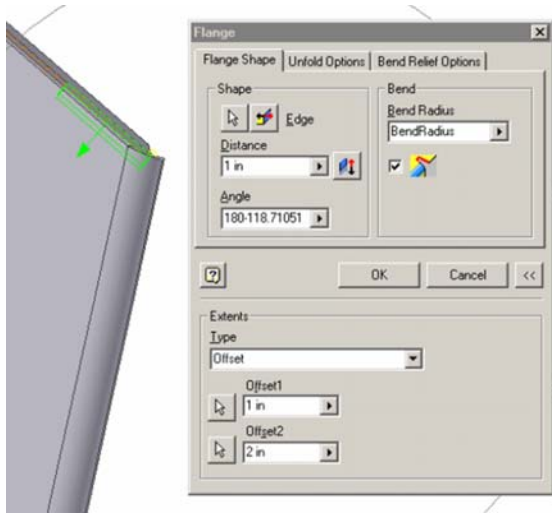


Figure 4B-8C: Rotating and zooming to a needed position area.

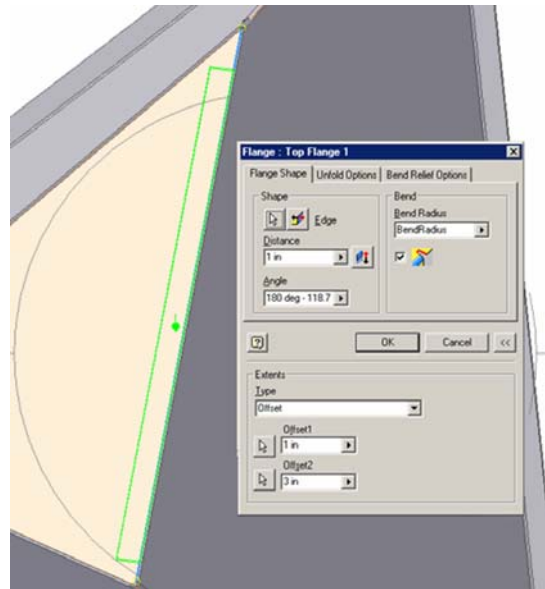


Figure 4B-8D: Checking for problem of self-intersecting.

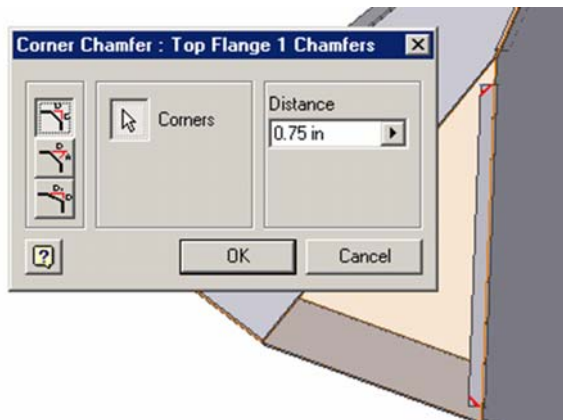


Figure 4B-8E: Rotating the model to a bottom view position, and adding **Corner Chamfers**.

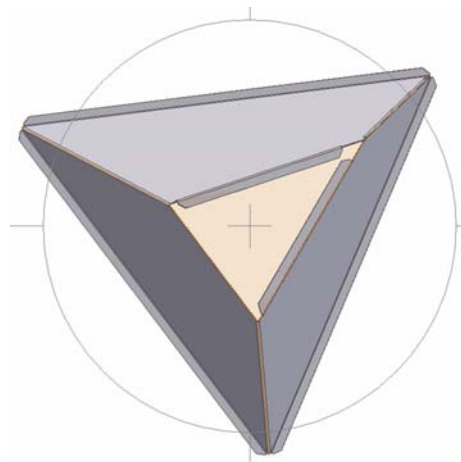


Figure 4B-8F: The completed part so far.

Step 3: Creating the cover for the truncated top surface

Next, start creating the cover for the truncated top surface. Click-select the **Work Plane** tool and then click-select the truncated top surface of the **Derived Part**, hold the mouse button and drag out a new **Work Plane**, an **Offset** text field appears; type 0.001 in

for distance (the tiny distance to separate the top **Face** panel to be created, from the existing lateral **Face** panels, and to allow Inventor to create the **Flat Pattern** view); next, click the green check mark to finish (*Figure 4B-9A*). Rename the new **Work Plane** feature “*Truncated Top Work Plane*” in the **Model** panel.

Next, select the *Truncated Top Work Plane* and click the **Sketch** button to start a new sketch; rename the new **Sketch** feature “*Truncated Top Surface Profile*” in the **Model** panel; click the **Look At** tool icon in the **Command Bar** and then the “*Truncated Top Surface Profile*” **Sketch** feature in the **Model** panel to switch to an orthographic view, and use the **Zoom Window** tool to zoom in closer to the area of the truncated top surface, which is in the shape of an isosceles triangle (*Figure 4B-9B*); zoom window again the top corner of the “isosceles triangle;” use the **Project Geometry** tool to project the two inclined inner edges of the lateral **Face** panels onto the *Truncated Top Surface Profile* sketch; change the projected lines’ **Style** from **Reference** to **Normal** in the **Command Bar**; next, use the “Circle-Extend-Trim” method to prevent potential problem of “open loop”, using the **Circle Center Point** tool to draw a construction circle, the **Extend** tool to extend the edge lines to the circle (*Figure 4B-9C*); the **Delete** key on the keyboard to delete the circle and the **Trim** tool to trim off the excessive line segments (*Figure 4B-9D*); next, use **Zoom All** and then **Zoom Window** tools to access an enlarged view of the lower left corner. The horizontal edge at the bottom of the triangle in this view is the edge of the top **Face** panel to be created, which connects the edge of the middle lateral **Face** panel; due to the way the **Bend** of the **Face** feature connects the two **Face** panels, there can be problems of self-intersecting when the flat pattern is generated, as shown in *Figure 4B-9E*. To avoid this potential problem, add bend relief as explained in the next paragraph.

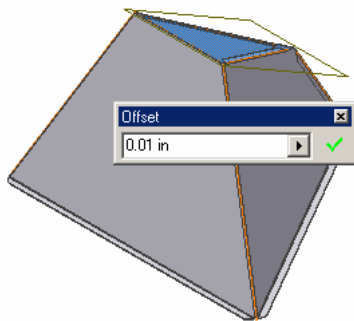


Figure 4B-8A: Creating the “Truncated Top Work Plane.”

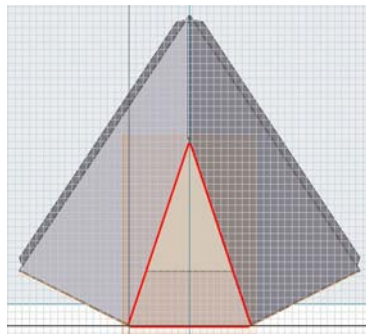


Figure 4B-9B: The “isosceles triangle” shape of the top surface.

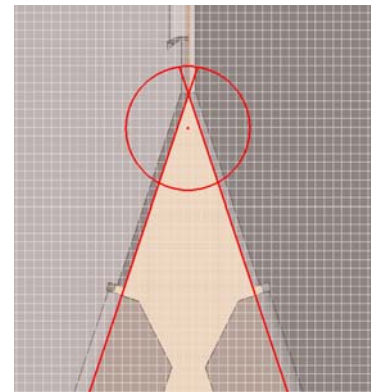


Figure 4B-9C: Preventing open loop problem.

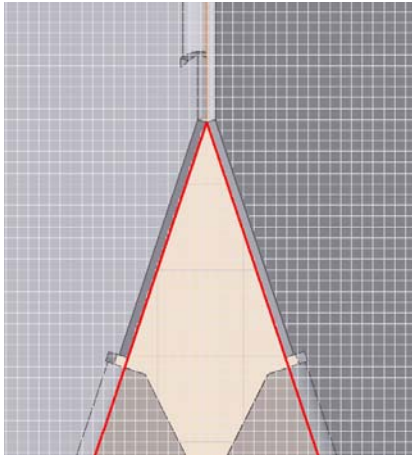


Figure 4B-9D: The top corner.

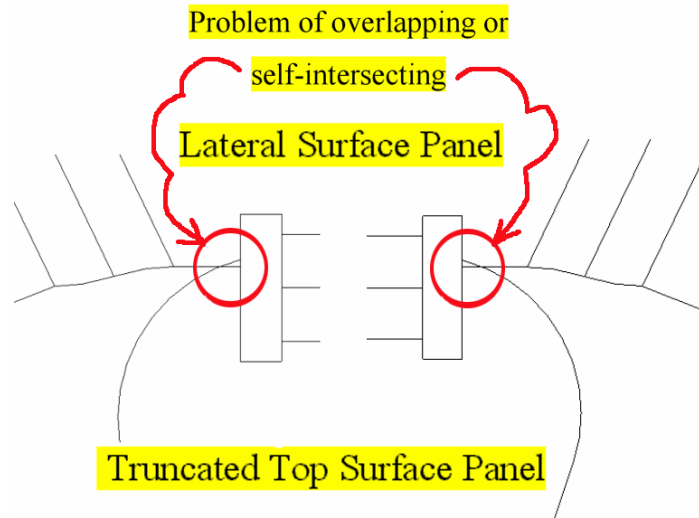


Figure 4B-9E: Potential problems of over-lapping or self-intersecting between two panels connected by a **Bend** feature.

Use the **Project Geometry** to project the horizontal outer edge of the lateral panel, change the projected lines' **Style** from **Reference** to **Normal** in the Command Bar; then use the **Fix** tool to fix this line to position; use the **Line** tool to draw a horizontal line above it and a vertical line on the right of the vertical edge between the straight portion of the lateral panels and the rounded **Bend** feature; next, selected the **General Dimension** tool to apply a 0.2-inch vertical linear dimension between the two horizontal lines (which allows a gap or relief between the top and lateral **Face** panels, for cutting the sheet metal material, when the flat pattern is developed, as shown later in *Figure 4B-10C*); and then a 0.25-inch horizontal linear dimension between the vertical line and the vertical edge between the straight portion of the lateral panels and the rounded **Bend** feature, by clicking the vertical edge (a projected line from the vertical edge selected by the **General Dimension** tool is automatically generated) and the vertical line (*Figure 4B-9F*); next, use the **Trim** tool and the **Delete** key on the keyboard to delete the dimension, the projected vertical edge and trim off all excessive line segments (*Figure 4B-9G*); next, repeat the same procedures to the right corner of the top surface, and use the Trim tool to cut off the middle segment of the top horizontal line (*Figure 4B-9H*) and to leave clean profiles of the left corner (*Figure 4B-8I*) and right corner (*Figure 4B-8J*) of the sketch; next, click the **Return** button or right-click for a shortcut menu and choose the **Finish Sketch** option to exit the *Truncated Top Surface Profile* sketch (*Figure 4B-9K*).

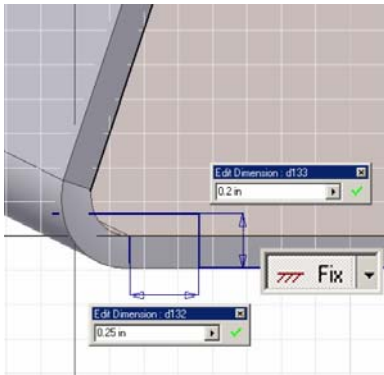


Figure 4B-9F: Projecting and fixing the horizontal outer edge line.

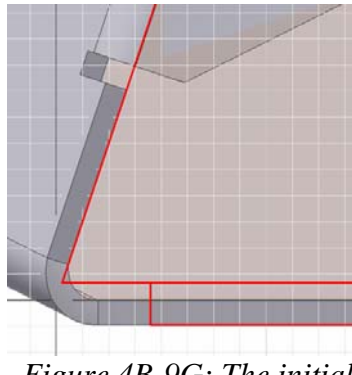


Figure 4B-9G: The initial sketch.

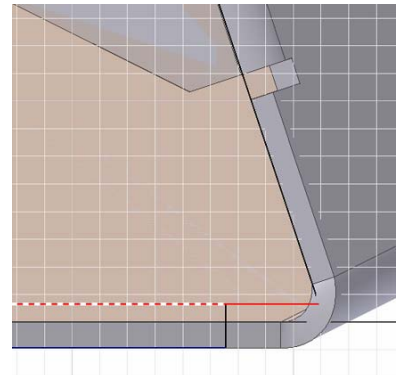


Figure 4B-9H: Trimming excessive line segment.

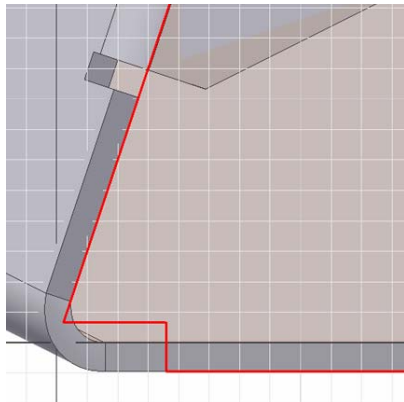


Figure 4B-9I: The profile at the lower-left corner.

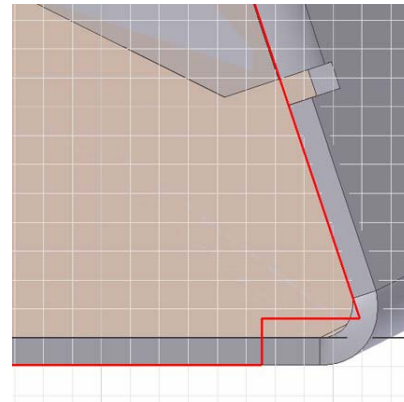


Figure 4B-9J: The profile at the lower-right corner.

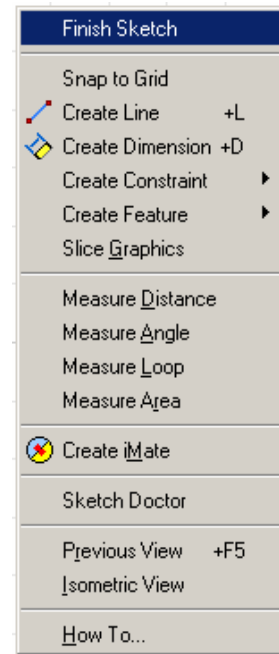


Figure 4B-9K: Finish Sketch.

Next, create the truncated top surface **Face** panel. Since the profile for the truncated top surface panel is created on the *Truncated Top Work Plane* that is not touching but a short gap of 0.001 inch above the lateral **Face** panel, Inventor will not generate a bend between the existing lateral **Face** panel and the new top panel by default, unless an edge is selected with the **Edges** arrow button in the **Bend** section of the **Face** tool's dialog window; therefore, we have two options: either assigning an edge for the

Bend feature (in this case, the inner edge of the existing lateral **Face** panel (*Figure 4B-10B*) and creating the top panel with a **Bend** connecting it to the existing lateral **Face** panel all at once; or creating a default separate top surface **Face** panel (*Figure 4B-9A*), and later add a bend with the **Bend** tool (*Figure 4B-9C*). in either case, rename the new **Face** feature “*Top Panel*” in the **Model** panel after it is created.

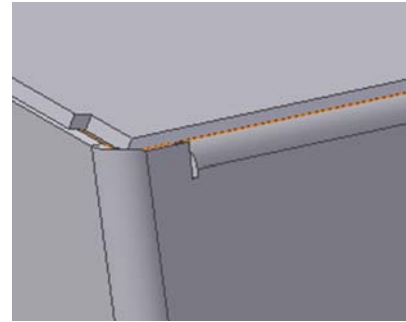
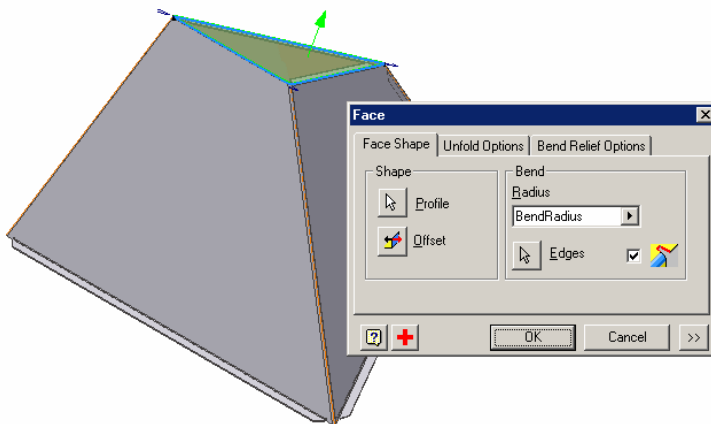


Figure 4B-10A: Creating a separate top panel without the Bend feature.

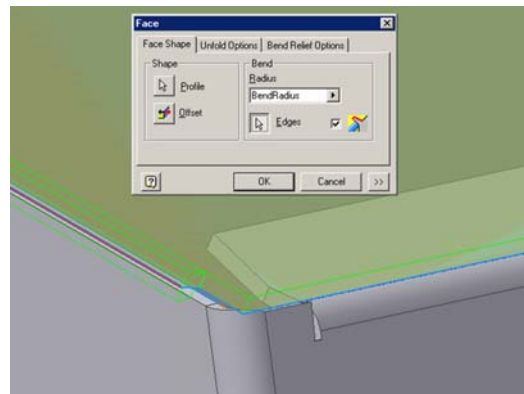
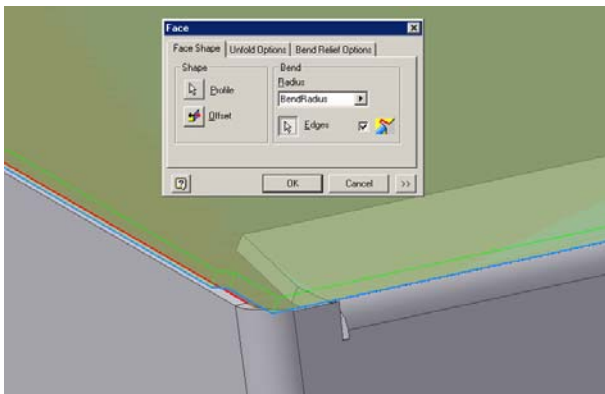


Figure 4B-10B: Selecting the inner edge of the lateral panel to create a Bend feature between the existing lateral Face panel and the new top face panel to be created.

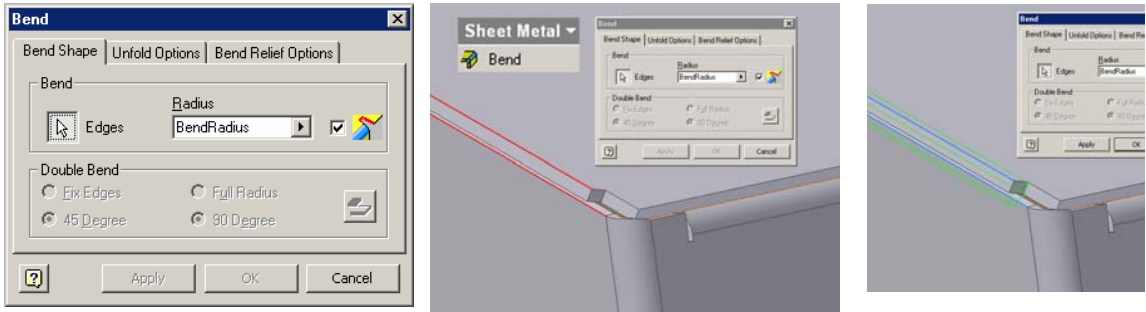


Figure 4B-9C: The **Bend** tool dialog window (left); selecting the same inner edge (middle) and creating a separate **Bend** feature (right).

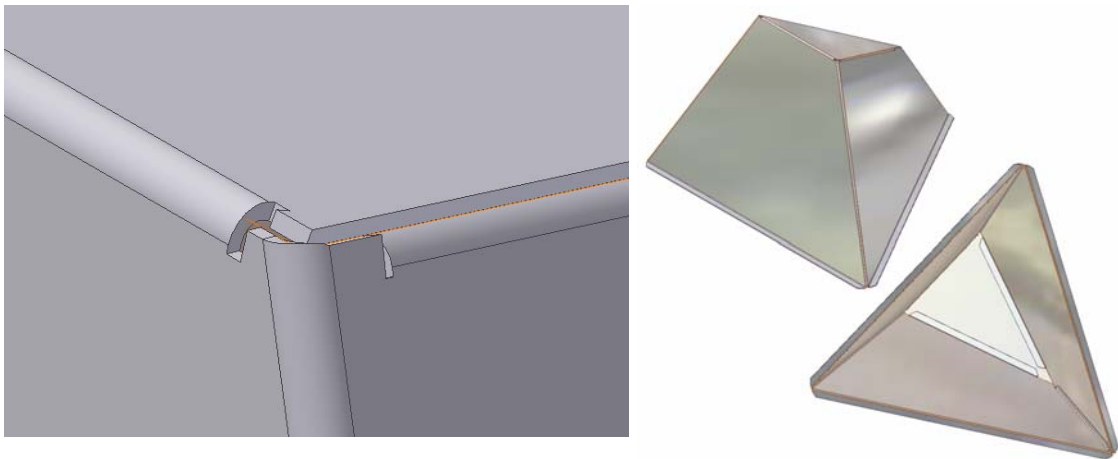
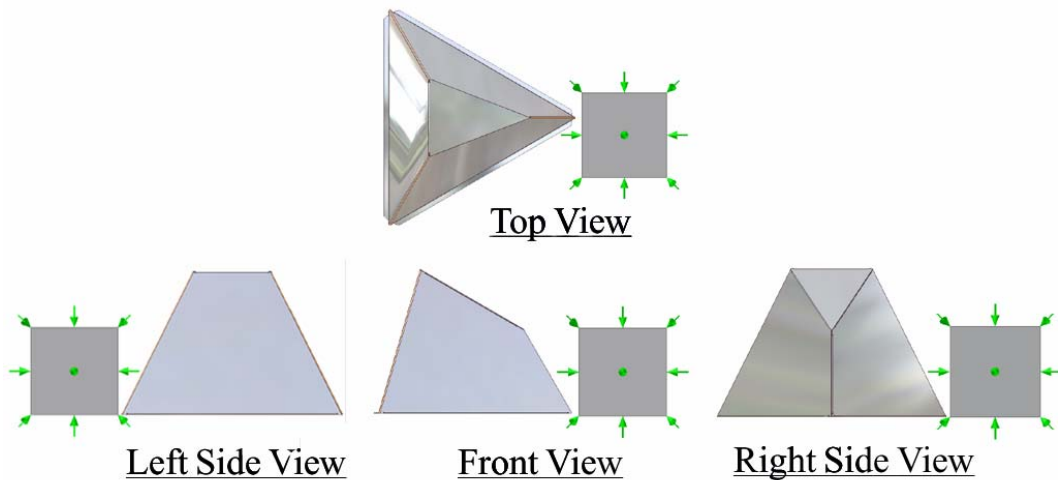


Figure 4B-9D: The **Top Panel** (left); the isometric views of the finished part (right). The orthographic views (Bottom).



Next, click-select the Flat Pattern tool from the **Sheet Metal** panel again to open the **Flat Pattern** window opens (*Figure 4B-10A*); use the Zoom window tool to zoom in the area where the bend occurs between the top **Face** panel and the middle lateral **Face** panel, the bend relief is shown (*Figure 4B-10B*); the **Flat Pattern** is actually a 3D model by itself and can be rotated to different views with the **Rotate** tool (*Figure 4B-10C*); in addition, it can be viewed in **Wireframe Display** mode (*Figure 4B-10C*). The features of the model are listed in the **Model** panel (*Figure 4B-10E*).

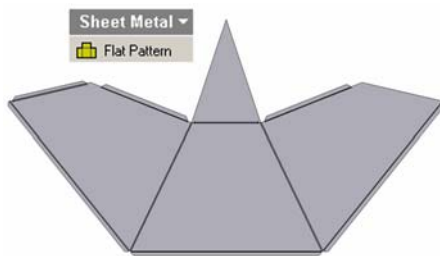


Figure 4B-10A: The Flat Pattern.

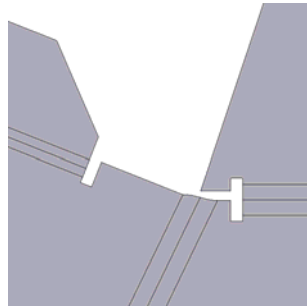


Figure 4B-10B: The bend relief.

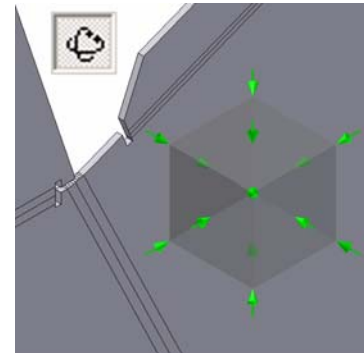


Figure 4B-10C: 3D view of the Flat Pattern.

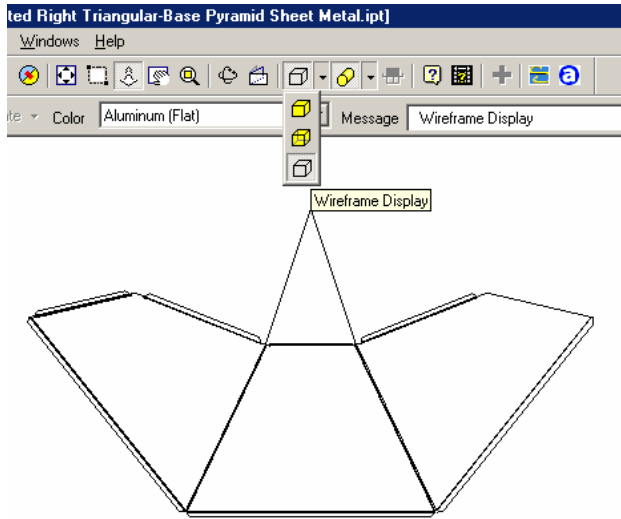


Figure 4B-10D: Wireframe Display of the Flat Pattern.

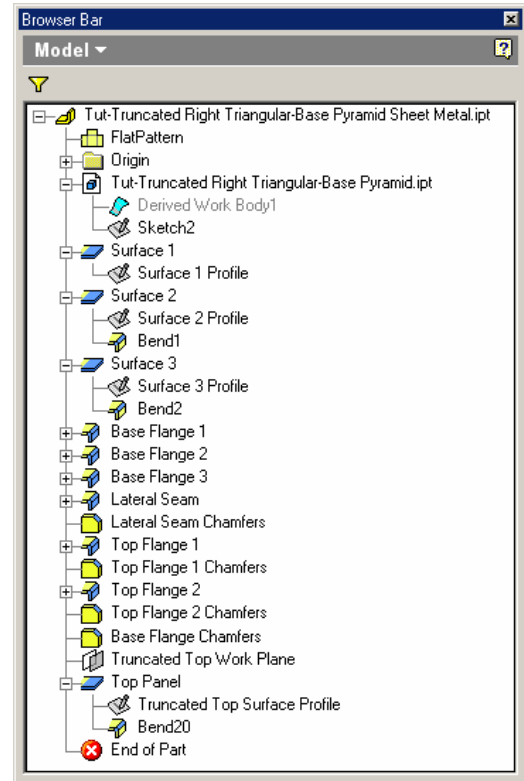


Figure 4B-10E: The features listed in the file's **Model** panel.

Please notice that, due to several factors such as available sizes of sheet metal materials, and the convenience and cost of cutting versus bending, it is sometimes preferred to create separate lateral **Face** panels with **Flange** features on the top and base of all panels as connecting and supporting elements for the separate top and base **Face panels**, in one piece (and one sheet metal .ipt file in Inventor), and separate top and base panels in separate files; and to assemble these separate sheet metal parts through welding, riveting, bolting and other assembling process. In Inventor, the folded models of separate sheet metal .ipt files can be assembled in a **Standard (in).iam** assembly file.

Congratulations! In the *Modules 4A* and *4B*, you have learned important techniques of

- Creating 3D folded models and 2D **Flat Pattern** views of regular and truncated right-axis triangle-based pyramids;

- Using the **Work Surface** of a **Derived Part** for the creation of sheet metal parts;
- Using **Work Plane**, **Work Point** and **Work Axis** tools creatively for the solution of mechanical engineering-related descriptive geometry problems.