Module 1B:
Parallel-Line Flat Pattern Development of Sheet-Metal Folded Model Wrapping the 3D Space of A Truncated Right Prism

In this *Module*, we will learn how to create a 3D solid model of a truncated right prism in an Inventor **Sheet Metal (in).ipt** file, which can be used as a **Derived Part** feature to be placed in another **Sheet Metal (in).ipt** file for its **Work Surface** reference geometry, to create the folded model of a sheet metal part wrapping the 3D space of the truncated prism, with Face, Flange and other tools, which are special tools for designing sheet metal parts. The basic procedures to complete this job in Inventor are as follows:

- Creating a 3D solid model of a truncated right prism in an Inventor **Sheet Metal (in).ipt** file;

- Starting another Inventor **Sheet Metal (in).ipt** file; using the Derived Component tool to place the 3D model of the truncated hexagon-based right prism as a **Derived Part** for its **Work Surface** reference geometry, from the previous **Sheet Metal (in).ipt** file;

- Selecting the surfaces of the **Derived Part**; using the **Project Geometry** tool to project the edge line profiles; and using the **Face** tool to created sheet metal Face panels with these profiles; and using other tools such as **Flange**, **Contour Flange**, **Corner Chamfers**, and **Extrude**, to complete the 3D folded model of the sheet metal part; and finally, using the **Flat Pattern** tool to create the **Flat pattern** view of the sheet metal part.

These basic procedures will be explained in this *Module*.

Section 1:
Creating the 3D Solid Model of a Regular Hexagonal Prism

Step 1: Creating a regular hexagon-based right-axis prism

Launch Inventor, start a **new Sheet Metal (in).ipt** file under the **English** tab. Hold the Shift key and click-select the **XZ Plane, XY Plane**, and the **Center Point** from

the **Model** panel; right-clock for the shortcut menu and check the **Visibility** option. The **XZ Plane** and **XY Plane** turn light blue and the **Center Point** turns yellow on the drawing screen. "*Sketch1*" is created by default in the **Model** panel. Go to the **View→Isometric** to switch to an isometric view for better visualization (*Figure 1B-1A*). Click the **Return** on the **Command Bar** to dismiss the default sketch feature. Select the default *Sketch1* feature from the **Model** panel, and press the **Delete** key on the keyboard to delete it. Click-select the **XZ Plane** and click the **Sketch** button from the **Command Bar** to start a new sketch, and rename it *Hexagon Profile* in the Model panel. Select the **Project Geometry** tool and click-select the **Center Point** either on the drawing screen or from the **Model** panel to project it onto the new sketch for a snap point; next, click-select the **Polygon** tool from the **Sketch** panel; the **Polygon** tool dialog window opens with 6 in the text field (indicating a 6-sided hexagon) and **Inscribed** button recessed as default settings (with the **Inscribed** option, the center point and one of the corner points are used to define a polygon); move the cursor closer to the projected **Center Point** and click once when the green dot snap indicator appears; move the cursor away horizontally leftwards or rightwards, and click once to establish a corner point (*Figure 1B-1C*); the basic hexagon profile is completed; right-click anywhere in the drawing screen for the shortcut menu and choose the **Done** option to exit the **Polygon** tool. Next, select the **Horizontal** constraint tool and click the visually horizontal edge line at the bottom of the hexagon actually horizontal; next, select the **General Dimension** tool, click-select the top and bottom horizontal edge of the hexagon and drag out a linear dimension feature, type 40 in the **Edit Dimension** text field and check the green check mark to apply the 40-inch "cross-flat" dimension (*Figure 1B-1B*). Click the **Return** button from the **Command Bar** to exit the *Hexagon Profile* sketch. Save the file as *Tut-Truncated Hex Prism Solid.ipt* inside a new folder to be created and named *Tut-Truncated Hex Prism*. Save often at the end of each step throughout the entire project.

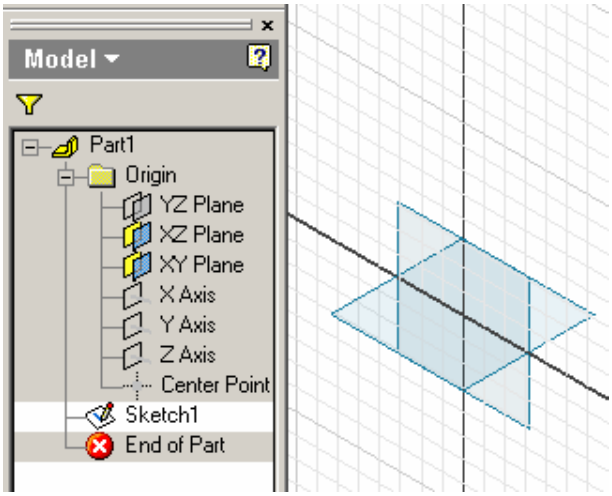


Figure 1B-1A: The default Sketch1.

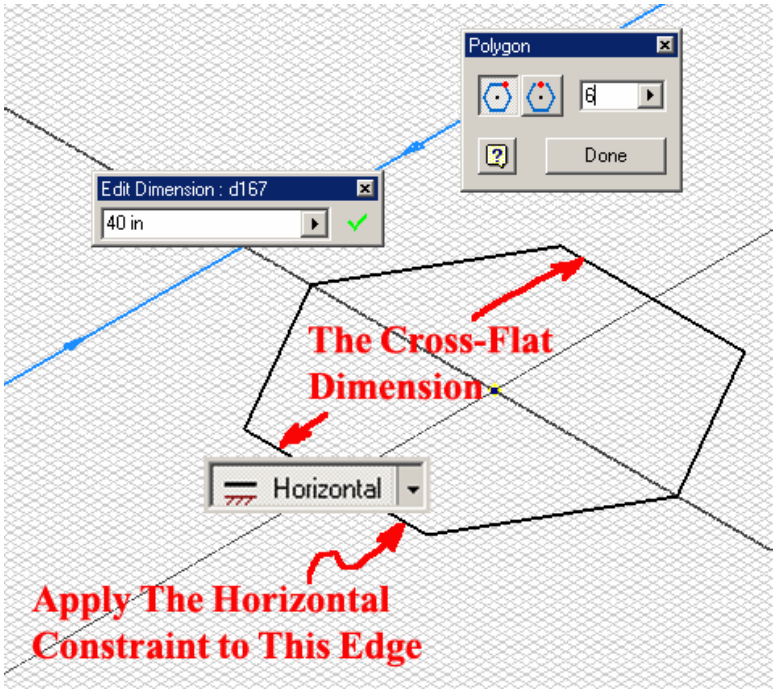
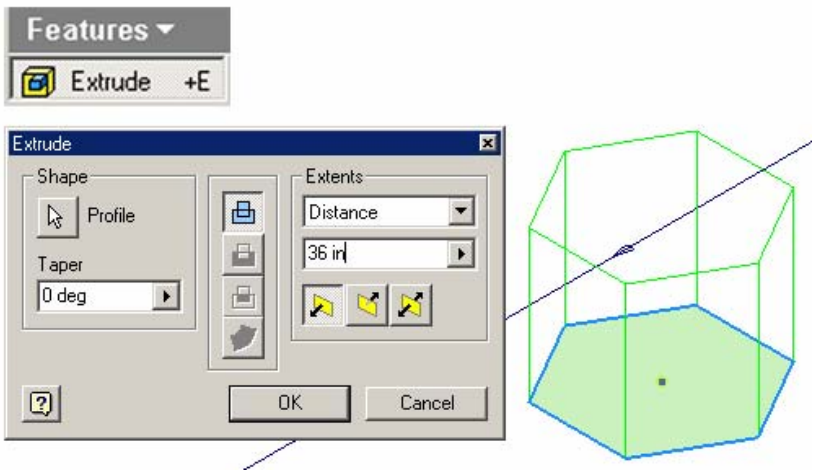


*Figure 1B-1B:The **Polygon** tool with the **Inscribed** option. The other option is **Circumscribed** which selects the center point and the midpoint of an edge to define a **Polygon**.*



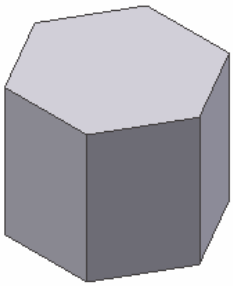


Figure 1B-1C: Extruding the hexagon-based regular right-axis prism.

Next, select the Extrude tool from the Features panel; in the Extrude tool's dialog window, select Join as Type, type 36 (inch) for Distance in the Extents section; and an upward **Direction**, to create the hexagon-based regular right-axis prism (*Figure 1B-1C*).

Step 2: Truncating the regular hexagon-based right-axis prism

Select the **YZ Plane** and click the Sketch button on the **Command Bar** to start a new sketch; rename it *Truncation Profile* in the **Model** panel.

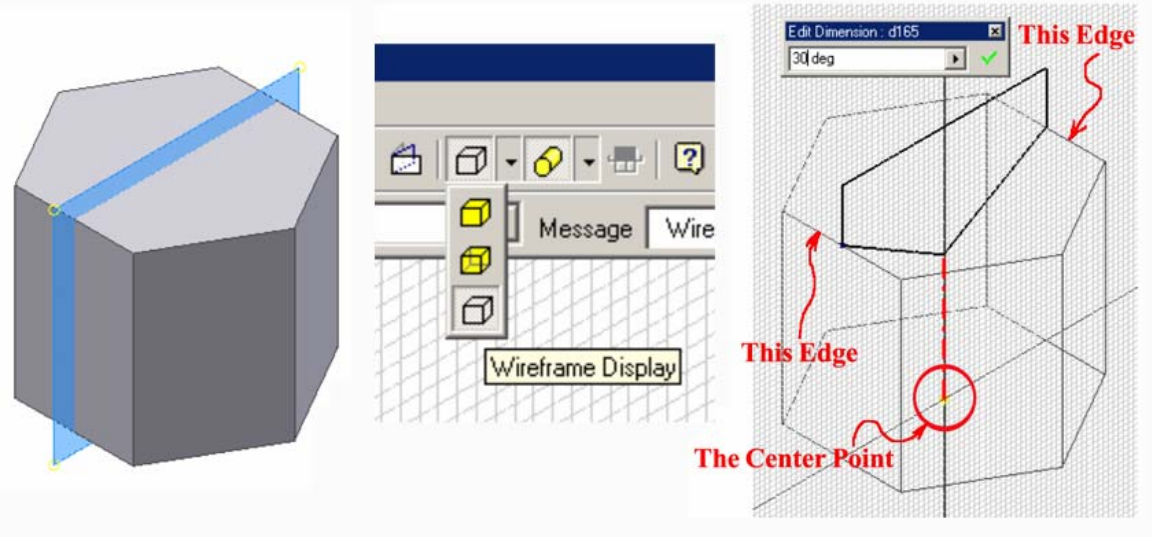
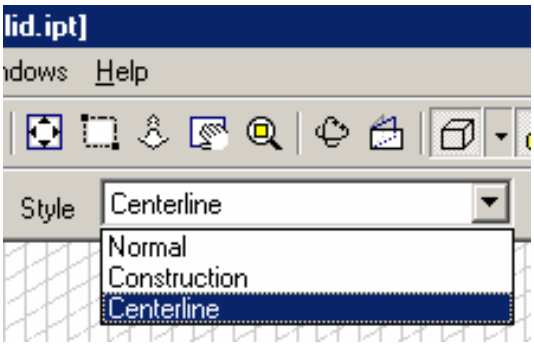


*Figure 1B-1D: Selecting the **YZ Plane** (left); changing the **Display** mode (middle); The Truncation Profile sketch (right); the **Style** pull-down menu on the **Command Bar** (bottom).*



Since the **YZ Plane** is hidden inside the *Hex Prism* **Extrude** feature, go to the **Display** mode selector on the **Command Bar**, click the inverse black arrow for a pull-down tool icon menu and select the **Wireframe Display** button to change the **Display** mode for a better visualization. Next, use the **Project Geometry** tool to project the **Center Point** and the two side edges onto the *Truncation Profile* sketch (when projected, the edges are projected as points since they are perpendicular to the **YZ Plane** where the *Truncation Profile* sketch is drawn); use the **Line** tool to draw a vertical line starting from

the projected **Center Point** with the help of the green dot snap indicator that appears on the screen at the projected **Center Point** when the cursor is moved closer to it, and go to the **Style** pull-down menu on the **Command Bar** to change its **Style** from **Normal** to **Centerline**; draw two vertical lines, and an inclined line starting from the point projected from one of the edge lines and extending beyond the centerline, also with the help of the green dot snap indicator that appears on the screen when the cursor is moved closer to it; next, draw a horizontal line above the top of the model; next, select the **General Dimension** tool to apply an 30-degree angular dimension between the horizontal line and the inclined line; next, use the **Trim** tool to trim off the unneeded segments of the centerline and the inclined line; next, use the **Line** tool to draw another inclined line connecting the point of intersection between the centerline and the first inclined line, and the point projected from another edge line of the model. The *Truncation Profile* sketch is complete (*Figure 1B-1D*); click the **Return** button to exit the sketch.

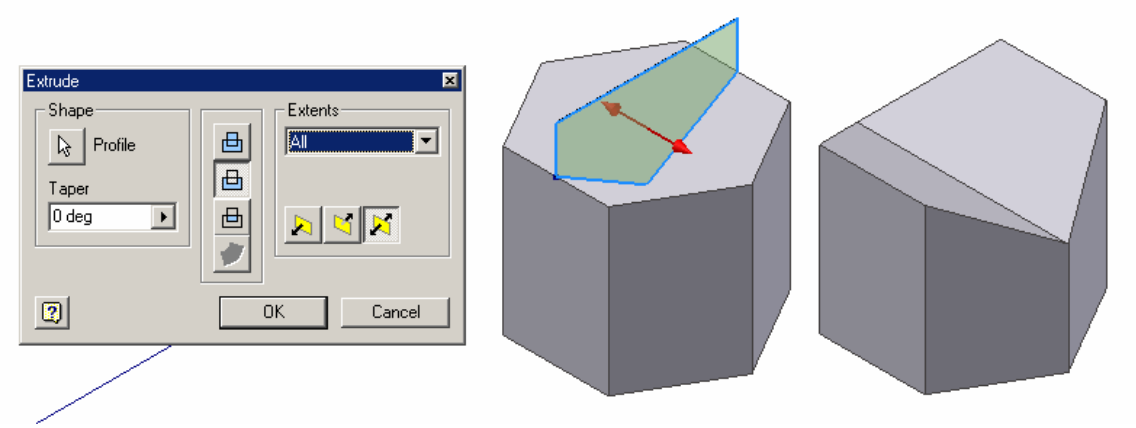


*Figure 1B-1E: Truncating the prism with the **Extrude** tool.*

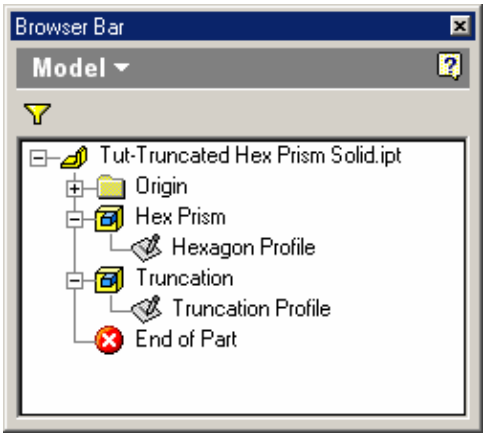


Figure 1B-1F: The features of the model

Next, select the **Extrude** tool and choose the **Cut** option for **Type**, **All** for **Extents** and **Midplane** for **Direction**, click the **OK** button to cut the prism with the profile in the *Truncation Profile* sketch (*Figure 1B-1E*). The features of the model are listed in the **Model** panel (*Figure 1B-1F*). The solid model file is completed. Save and close the file.

Section 2: Creating the Sheet Metal Part Wrapping The 3D Space of A Regular Hexagonal Prism

The basic procedures of creating the sheet metal part wrapping the 3d space of a regular hexagonal prism in Inventor have been explained in the introductory paragraphs of this *Module*. They include:

- Placing the **Derived Part** (*Figure 1B-2A*);
- Creating **Face** Profile sketches on the surface of the **Derived part** (*Figure 1B-2B*);
- Creating needed **Flange** features for connecting adjacent **Face** panels (*Figure 1B-2C*);
- Creating needed **Corner Chamfers** on the ends of **Flange** features (*Figure 1B-2D*);
- Creating **Work Plane** features for the top and base **Face** panels (*Figure 1B-2E*);
- Creating the top and base **Face** panels with appropriate **Bend** features (*Figure 1B-2F*);
- Completing the 3D folded model and 2D **Flat Pattern** view of the sheet metal part (*Figure 1B-2G*).

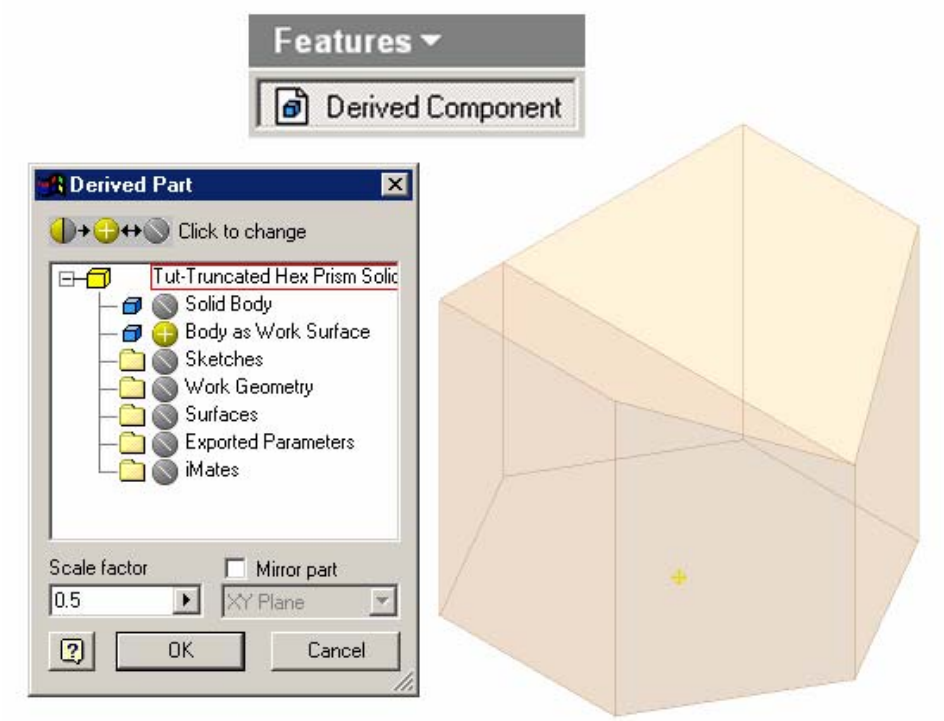


*Figure 1B-2A: Placing the **Derived Part**.*

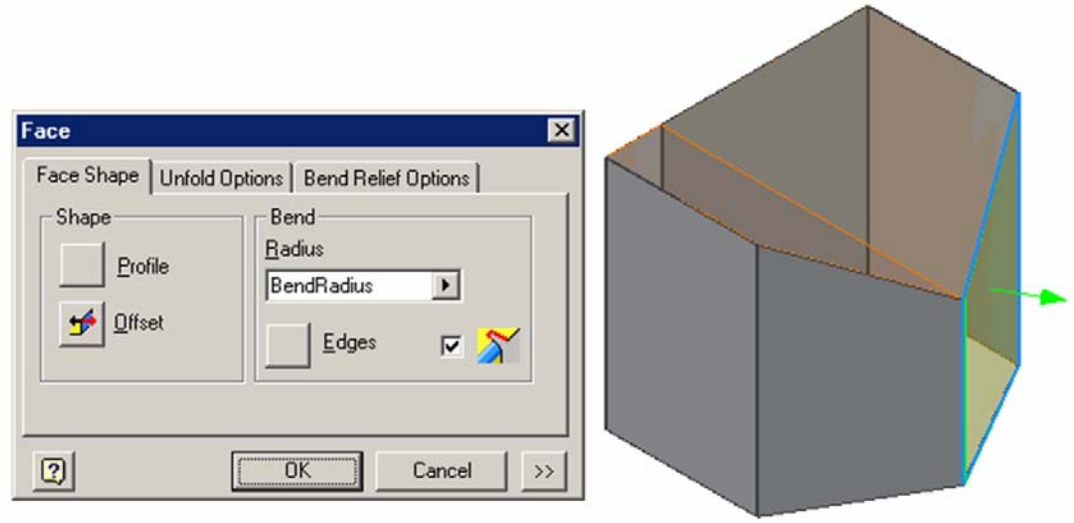


*Figure 1B-2B: Creating **Face** Profile sketches on the surface of the **Derived part**.*

These procedures will be explained in details in subsequent Modules; and this project will be used to text the students' ability to independently solve descriptive geometry problems through sheet metal design projects at the end of the course, as a hands-on semester exam or exploration project.

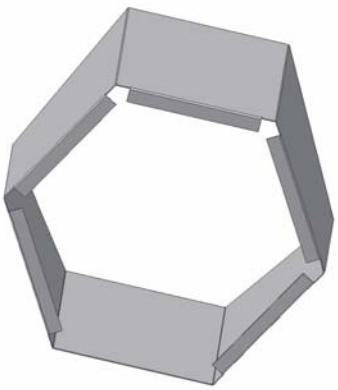


*Figure 1B-2C: Creating needed **Flange** features for connecting adjacent **Face** panels.*

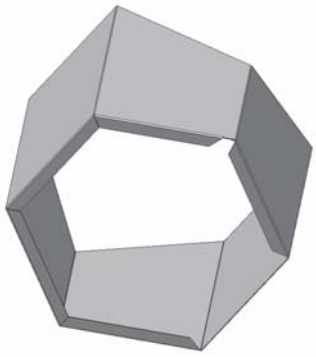


*Figure 1B-2D: Creating needed **Corner Chamfers** on the ends of **Flange** features.*

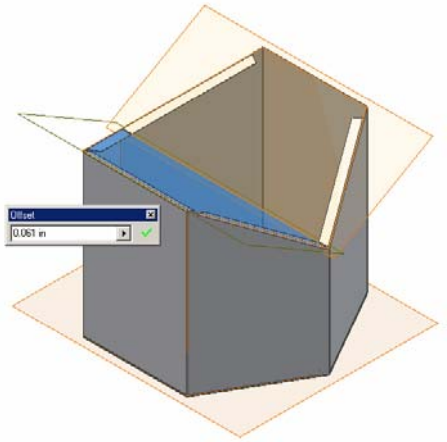


*Figure 1B-2E: Creating **Work Plane** features for the top and base **Face** panels.*

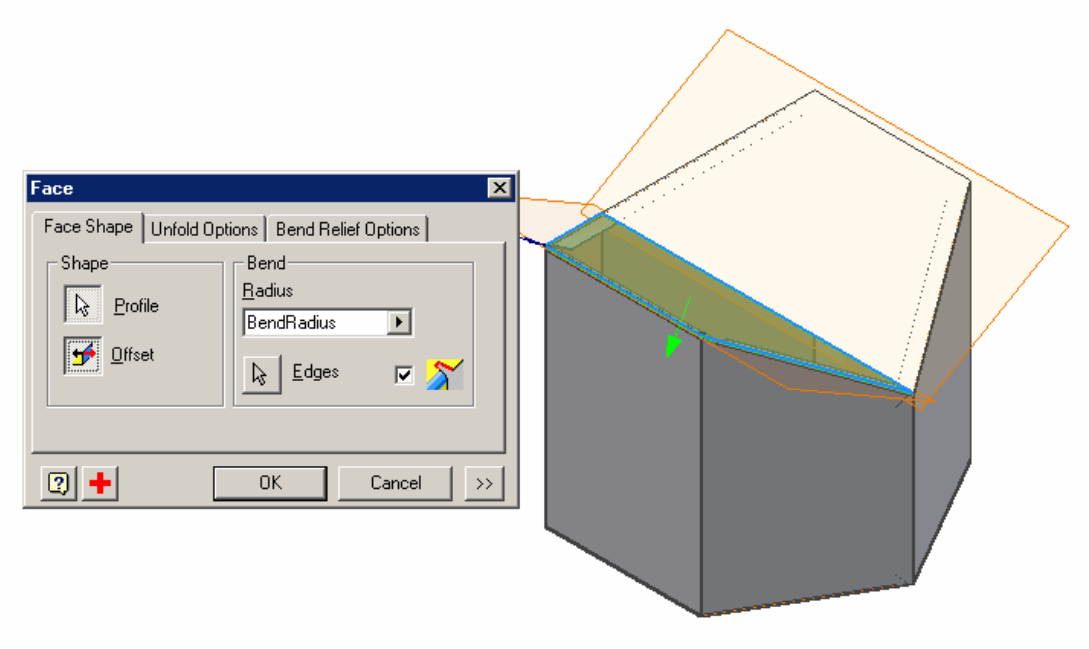


*Figure 1B-2F: Creating the top and base **Face** panels with appropriate **Bend** features.*

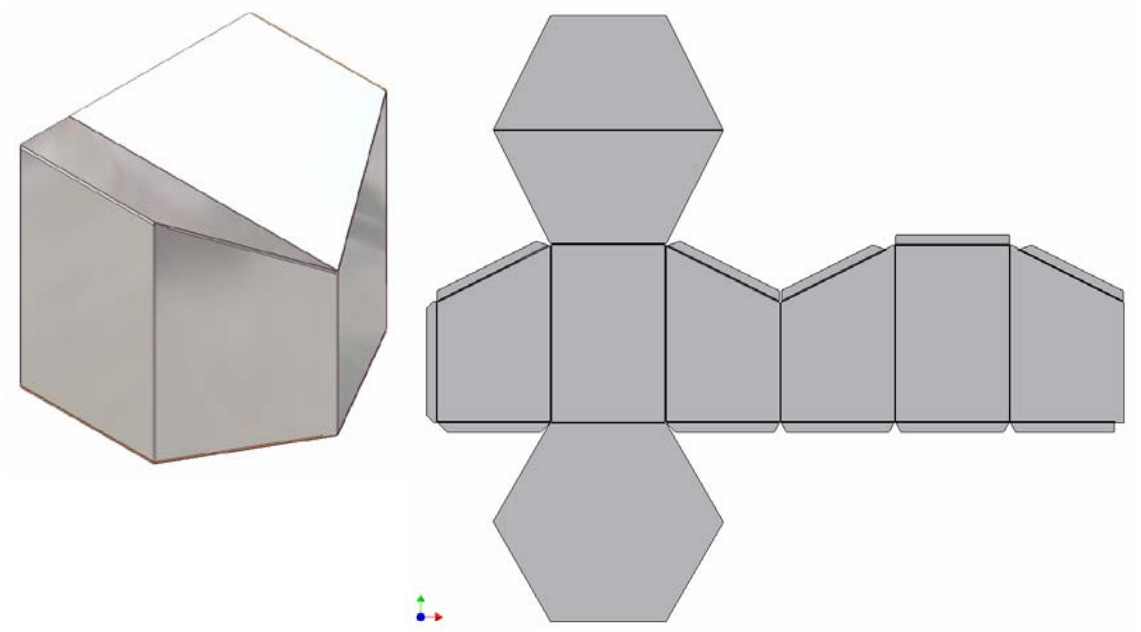


*Figure 1B-2G: The 3D folded model and 2D **Flat Pattern** view of the sheet metal part.*

We shall now proceed to the next *Module*.