Module 2: Radial-Line Sheet-Metal 3D Modeling and 2D Pattern Development: Right Cone (Regular, Frustum, and Truncated)

In this tutorial, we will learn how to build a 3D model of a sheet metal part wrapping a right cone (regular, frustum and truncated), which is a solid of revolution, and which flat pattern is one of the two types of "radial-line development" (one in which element lines "radiate" from a common point called the vertex); we will accomplish this task with the **Revolve** and **Extrude** tools, which are generic 3D modeling tools from Inventor's **Features** panel. The basic procedures to complete this job in Inventor are to first draw a cut-off section profile of a regular or frustum of a cone (as shown in *Figure 2-1C*); and to use the Revolve tool to revolve this profile around an axis line into a 3D solid, with 359.999° (instead of 360° since there needs to be a gap for Inventor to flat the pattern out). To cut the regular cone into a truncated cone, a profile that includes an angled line is drawn and used to cut the regular cone.

After the truncated cone is created, we will move to *Module 4: Intersection and Development of Sheet Metal Parts in Inventor*, which introduces the topic of intersection and development of conic and other sheet metal parts, with the same tools previously learned but used in different ways. In addition, we will study how to use the **Work Plane** tool to solve important descriptive geometry problems in Inventor.

Section 1: Creating the Sheet Metal Part Wrapping A Regular Cone

Launch Inventor, start a new **Sheet Metal (in).ipt** file under **English** tab. Turn **Visibility** on for **XZ** and **XY Planes**, plus the **Center Point** from the **Model** panel. "*Sketch1*" is created by default in the **XY Plane** (the one parallel to your computer's screen). Go to **View→Isometric** for better visualization in 3D space; and click the **Look At** tool button to return to orthographic view and start creating the cross-sectional profile of the cone.

For a frustum of a cone (one with the top cut off with a plane parallel to the base plane, or in this case, "horizontally"), use the Project Geometry tool to project the **Center Point** for a snap first; then use the **Line** tool to draw a slanted line of the cross-section profile, plus a line of revolution staring from the projected **Center Point** (go to the **Style** pull-down menu to change the **Style** of the line of revolution to **Centerline**); then use the Offset tool to draw another slanted line parallel to the first one; use the General Dimensions tool to apply a 0.120 **Aligned Dimension** between the two slanted lines; then use the Line tool again to draw two short line segments between the upper and lower endpoints of the slanted lines, so as to complete the profile, as shown in *Figure 2-1A*, make sure that the parallel lines representing the inside and outside walls of the conic sheet metal part are parallel, and connected with a short edge line perpendicular to both.

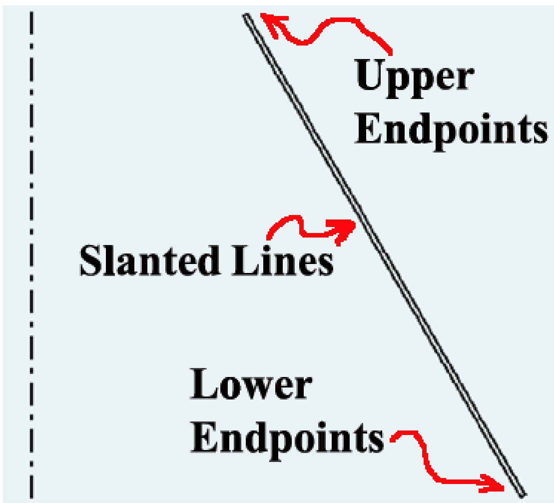


Figure 2-1A: Profile sketch (left) and 3D solid feature of a frustum of cone (right)

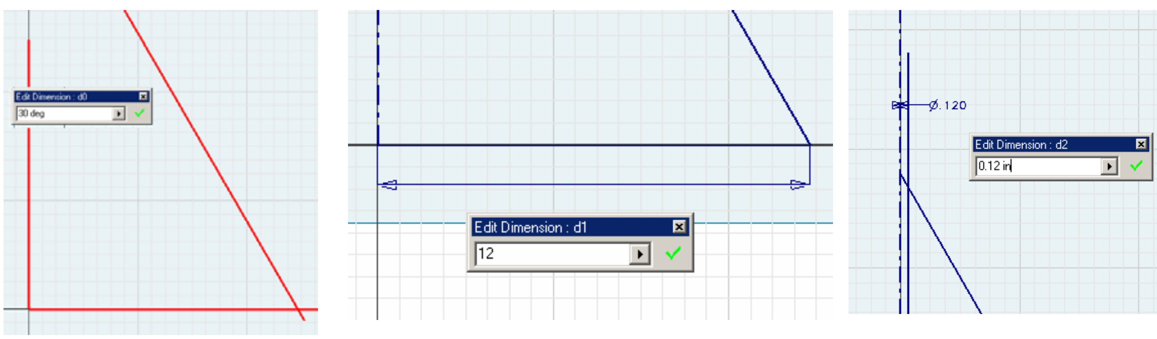
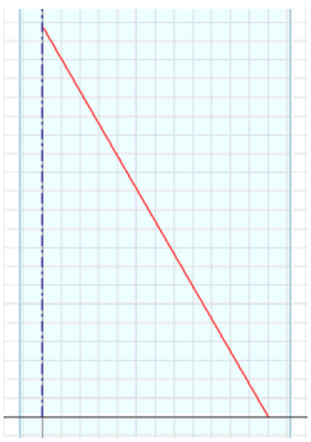
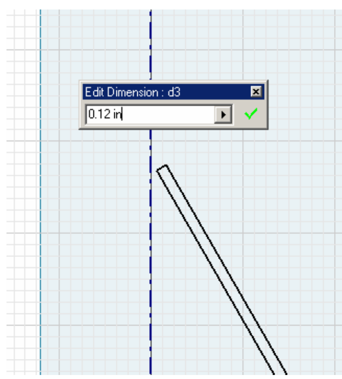


Figure 2-1B: ***Center Point****, vertical line of axis, horizontal line, and angled line (Left); applying dimensions (middle); a gap at the vertex (right).*

For a regular cone (one with the vertex intact) that will be used to create a truncated cone later, we need a profile as shown in *Figure 2-1B*. Notice that not only the parallel lines representing the inside and outside walls of the conic sheet metal part are parallel; but also there needs to be a gap at the vertex of the cone, approximately the same size as the thickness of the sheet metal material (in this case, 0.120 inch); or half of this (0.060 inch) from the line of axis of revolution. Use the **Project Geometry** tool to project the **Center Point** onto the sketch first; then use the **Line** tool to draw the vertical line of axis of revolution starting from the projected **Center Point**, and change the line **Style** to **Centerline**; then draw another vertical line next to the axis of revolution (in this case, to the right of it; and we called this line "gap line"); then use he **General Dimensions** tool to apply a ϕ0.120-inch diametrical dimension as shown in *Figure 2-1B* (right); next, use the Line tool to draw a horizontal line starting from the projected **Center Point** with the help of snap, and an angled line (for the conic wall) starting at the endpoint of the horizontal line passing the ; then use the **General Dimension** tool to apply a 30° angular dimension between the angled line and the axis of revolution, and 12-inch linear dimension to the horizontal line. Next, use the Trim tool to trim off the segment of the angled line beyond the "gap line" (*Figure 2-1B*). Next, use the Offset and General Dimension tools to draw the second conic wall line parallel to the first one; then use the **General Dimension** tool to apply an **Offset** distance value of 0.120 (same as the sheet metal material thickness); next, use the **Line** tool to draw two short line segments connecting the upper and lower endpoints of the slanted conical wall line and complete the profile (*Figure 2-1C*). Click the **Return** button to exit the **Sketch** mode. Create a folder named *Tut 2-Cone and Intersection*, in a convenient directory, save the file as *Tut 3-Truncated Cone.ipt* inside this folder. Save often.





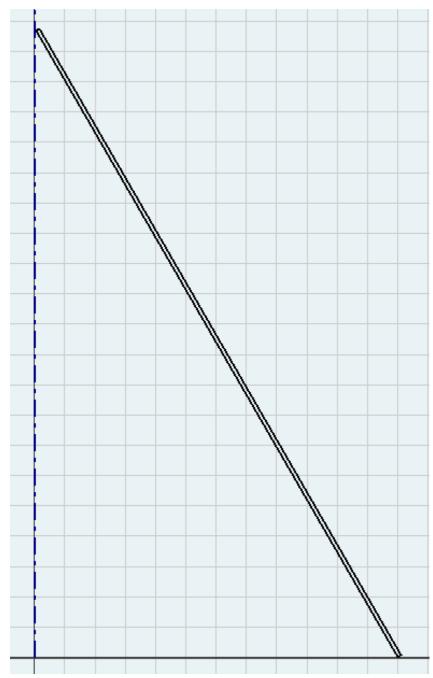


Figure 2-1C: Inside wall profile line (left); offset outside wall line (middle); completed section for regular cone (right).

Click-select the Revolve tool icon from the Feature panel; the Revolve tool dialog window opens; in the Extents section, change from **Full** to **Angle** and type 359.999 in the text field; click the **Profile** button and select the sectional profile; click he **Axis** button and select the axis line, green 3D geometry outlines appears on the screen; click the **OK** button (*Figure 2-2A*), and the 3D solid of the regular cone appears on the screen (See *Figure 2-2B*).

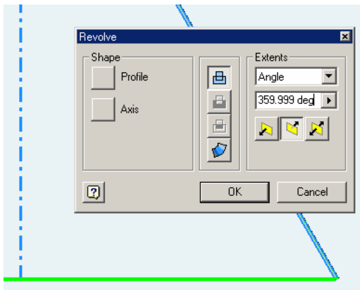


*Figure 2-2A: The **Revolve** tool window.*

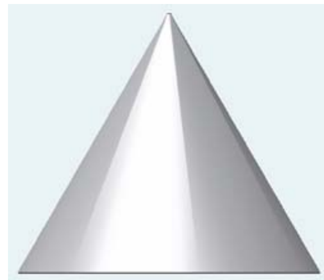


Figure 2-2B: 3D solid of the regular cone.

Section 2:
Truncating the Regular Cone

Click-select the **XY Plane** from the **Model** panel (or on the screen); click the **Sketch** button from the **Command Bar**; start to draw an angled truncating line and additional lines to make it a closed profile, with the help of the green snap indicator; then use the **General Dimension** tool to constrain the angled line to 30° with respect to the top horizontal line (*Figure 2-3A*). Next, use the **Project Geometry** tool and click-select the left edge of the 3D cone model; the edge line is projected onto the sketch. Next, use the Trim tool to trim off the top portion of the edge line above the truncating line (*Figure 2-3B*). Next, use the **General Dimension** tool to apply a linear dimension on the edge line for a height of 18-inch; the truncated edge line jumps up with the profile. Click the **Return** button to exit the sketch. Click-select the **Extrude** tool from the **Features** panel; in the tool window, choose **Cut** for **Type**, **All** for **Extents**, and **Mid Plane** for **Direction**, click the **OK** button to complete the truncated cone (*Figure 2-3D*). Save the file; and go to File→**Save A Copy As** menu to save it as a new file as *Tut 3-Truncated Cone Top.ipt*, in the same *Tut 2-Cone and Intersection* folder. Close the current file.

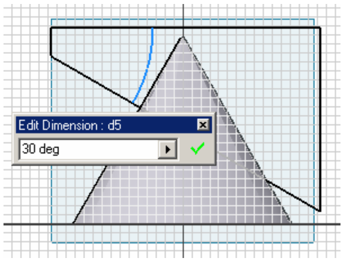


Figure 2-3A: Basic truncating profile before height is applied.

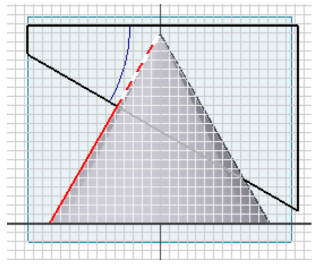


Figure 2-3B Projected and trimmed edge line

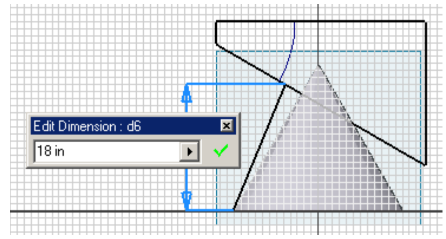


Figure 2-3C: Height dimension applied.

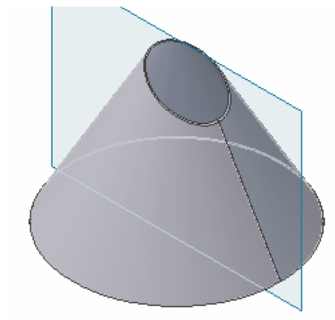


Figure 2-3D: The truncated cone.

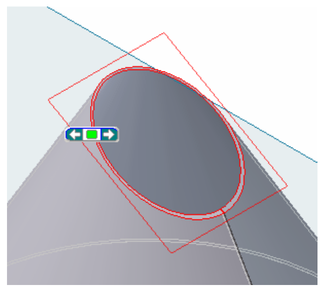


Figure 2-4A: Click-select the cone's truncated surface; and pick up the work plane

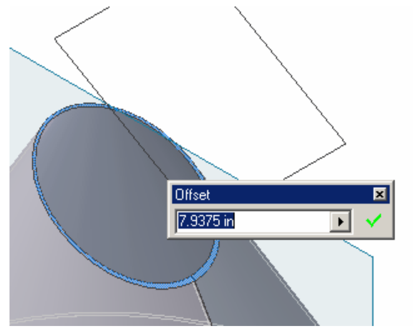


Figure 2-4B: Click, hold and drag out a work plane.

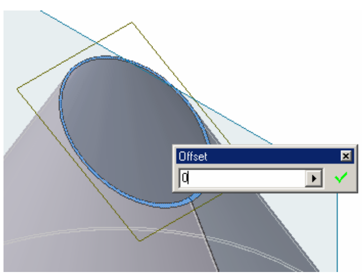


Figure 2-4C: Type in the Distance value.

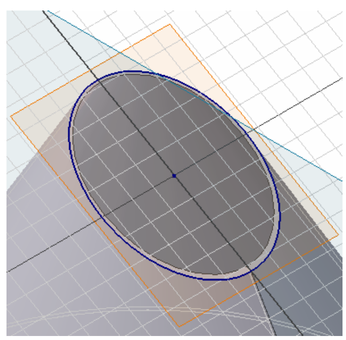


Figure 2-4D: Start a sketch on the new plane.

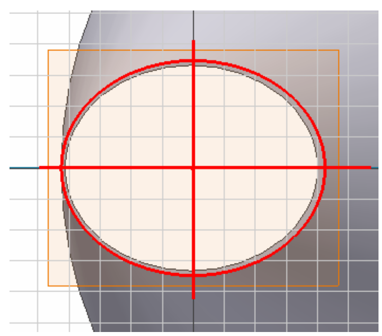


Figure 2-4E: the projected outline, Center Point, and axis.

Section 3:
Creating the Top of The Truncated Cone

We will create the top piece of the truncated cone in a way similar to what has been explained in *Module 1A*, but with a different approach on the creation of the profile sketch. The shape of the top appears to be an ellipse; however, due to the way the first 3D feature of the regular cone is created with 359.999° instead of full 360°, there is a gap in the edge outline of the truncated surface, making any projected profile sketch an open, not a closed path, which can not be used without some repair by the "**Sketch Doctor Examiner**" functions (*Figure 2-4F*), which is not always successful. We will try to avoid this problem, as will be explained in the following steps.

Open the *Tut 3-Truncated Cone Top.ipt* file; and create a new work plane on the truncated surface of the cone. Click-select the truncated surface, then the **Work Plane** tool from the **Sheet Metal** (or **Features**, or **Solid**) panel; move the cursor closer to the surface; red outlines of the surface and a rectangular work plane outline appear (*Figure 2-4A*); hold down the mouse button and drag up any distance; a dimension text field appears (*Figure 2-4B*); highlight the value and type 0, click the green check mark (*Figure 2-4C*). The new work plane is created; rename the it *Top Work Plane* in the Model panel. Click-select the new plane; click the **Sketch** button; a new sketch screen appears (*Figure 2-4CD*). Use the **Project Geometry** tool, click on the edge outline of the truncated surface on the 3D model, and the Center Point feature in the Model panel. Draw one horizontal and one vertical axis lines, both starting from the projected **Center Point** and extending beyond the elliptical outline of the truncated surface, with the help of the green snap indicators (*Figure 2-4E*). Next, use the **Trim** tool to trim off the portion of the axis lines beyond the elliptical outline (*Figure 2-4G*). Delete the open projected elliptical outline. This provides three snap points for the new, closed ellipse (the **Center Point**, the major axis endpoint, and the minor axis endpoint, which are essential in the construction of an ellipse in any CAD program). Next, click-select the **Ellipse** tool from the **Sketch** panel; click on the projected **Center Point**, then one end point of one axis (major or minor); next, drag the curve out until it reaches the endpoint of another axis at click once at the appearance of the green snap indicator. The new ellipse is exactly the same as the outline of the truncated surface (*Figure 2-4H*); rename the sketch *Top Face Sketch* in the Model panel. A closed elliptical profile is now available (*Figure 2-4I*). Next, click-select all irrelevant features from the **Model** panel and press the Delete key on the keyboard to delete them, so that only the *Top Work Plane* and the *Top Face Sketch* remain (*Figure 2-4J*). Finally use the **Face** tool (with the **Offset** direction arrow upwards) to create the top piece (*Figure 2-4K*). The features of the 3D model are listed in the Model panel (*Figure 2-4L*). Save and close the file.

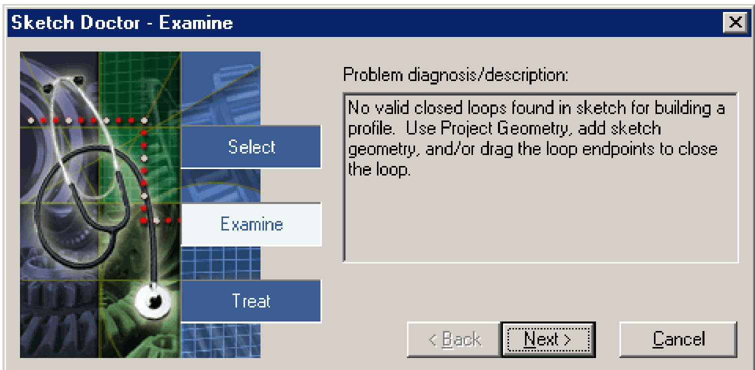


*Figure 2-4F: The "**Sketch Doctor Examiner**."*

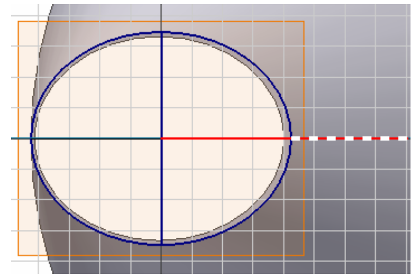


Figure 2-4G: Trim off the portion of the axis beyong the projected elliptical outline

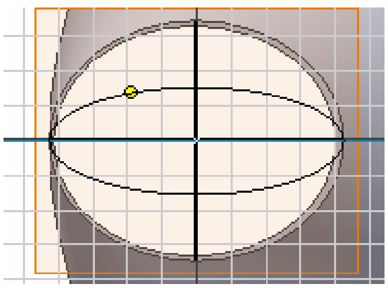


Figure 2-4H: Draw the new elliptical outline.

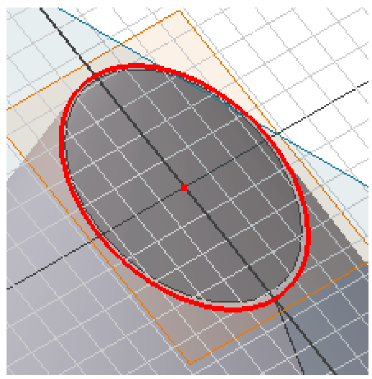


Figure 2-4I: The closed outline.

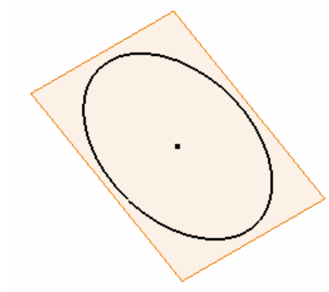


Figure 2-4J: Profile for the top piece.

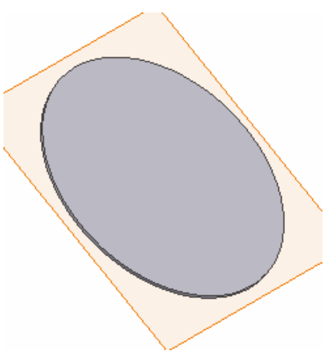


Figure 2-4K: The top piece.

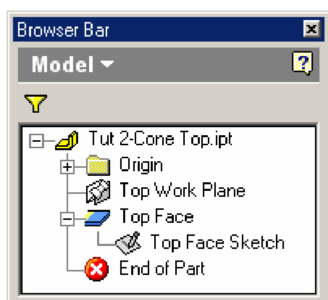


Figure 2-4L: The features listed in the Model panel.

This completes the Modules 2. Congratulations! You have learned how to:

- Create the 3D models and flat patterns of sheet metal parts wrapping the volume of right-axis cones (regular, frustum and truncated).