

## Module 3A: Intersection and Development of Sheet Metal Parts in Inventor

We will now use the same *Tut 3-Truncated Cone Top.ipt* file to solve the problems of intersection and development of different sheet metal parts in Inventor, creating the following three parts that intersect the truncated cone:

- A cylinder that intersects the truncated cone along a slanted axis on the **YZ Plane**, from a plane perpendicular to the slanted axis;
- A hexagonal prism that intersects the truncated cone from a plane parallel to the **YZ Plane**, along a straight horizontal axis perpendicular to the **YZ Plane**;
- A cylinder that intersects the truncated cone from a plane at a 45° angle relative to the **YZ Plane** and intersecting the **YZ Plane** at the **Y Axis**.

### *Section 1:*

#### *Cutting the Truncated Cone with the 3D Volumes of Various Geometric Solids*

*Step 1: Cutting the truncated cone with a slanted cylinder that intersects it along a slanted axis on the YZ plane, from a plane perpendicular to the slanted axis*

First of all, will need to create a slanted axis and a work plane perpendicular to it

Open the same *Tut 3-Truncated Cone Top.ipt* file. Click-select the **YZ Plane** from the screen or the **Model** panel; click the **Sketch** button to start a new sketch. Use the **Project Geometry** tool to project the **Center Point** onto the new sketch; use the **Line** tool to draw a slanted line and a horizontal line snapping to the projected **Center Point** (*Figure 3A-5A*). Click the **Return** button to exit the Sketch mode. Rename the **Sketch** feature as *Slanted Axis*. Click-select the **Work Plane** tool from the **Features** panel move the cursor closer to the top endpoint of the slanted line; an orange rectangular outlined work plane symbol appears, and the slanted line turns red, indicating a new work plane perpendicular to the slanted line (*Figure 3A-5B*); click once and a light orange work plane appears; rename it *Slanted Cylinder Work Plane* in the **Model** panel (See *Figure 3AA-5C*).

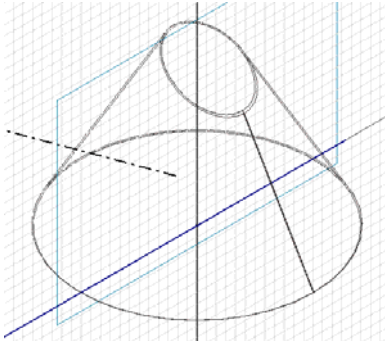


Figure 3AA-5A: Slanted axis sketch.

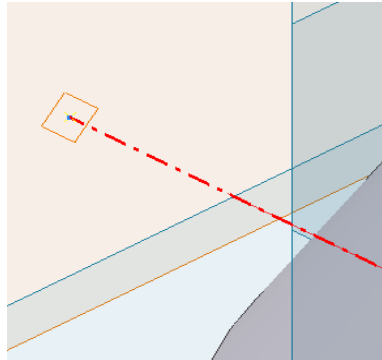


Figure 3A-5B: Starting a new Work Plane feature.

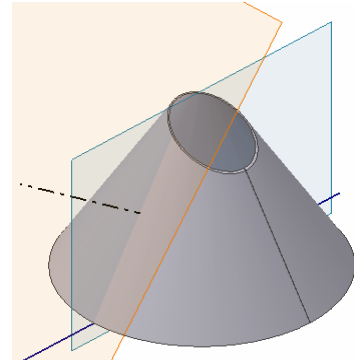


Figure 3A-5C: Completing the creation of a new Work Plane feature.

Next, use the **Rotate** tool to rotate the model and see if the new *Slanted Cylinder Work Plane* is really perpendicular to the slanted line; it appears to be. Affirm its perpendicularity by going to the **Tool**→**Measure Angle** menu to select the **Measure Angle** tool, and clicking the *Slanted Cylinder Work Plane* and the slanted line from the *Slanted Axis Sketch* feature, which turn red when clicked; the Angle text field reads 90 deg after both the **Work Plane** feature and the slanted line have been click-selected (*Figure 3A-5D*). Next, Change the angle of the slanted axis line. Right-click the *Slanted Axis Sketch* feature in the **Model** panel and choose the **Edit Sketch** option. Select the **General Dimension** tool and apply a 45° Angular Dimension between the slanted line and the horizontal line; click the **Return** button to exit the **Sketch** mode. The *Slanted Cylinder Work Plane* instantly changes to a new position, which is still perpendicular to the slanted axis.

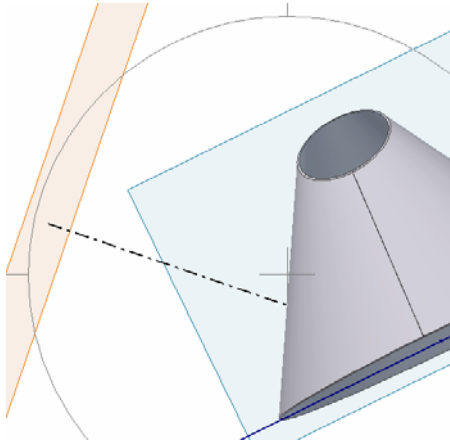


Figure 3A-5C: Rotating the model.

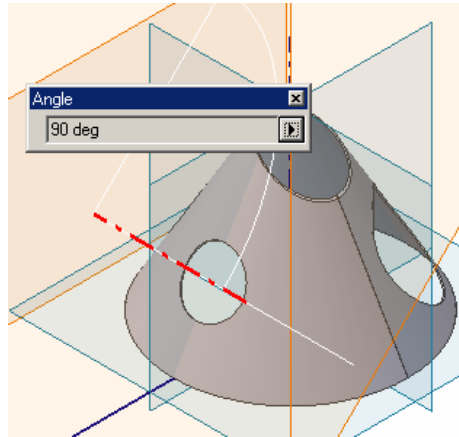


Figure 3A-5D: Checking perpendicularity.

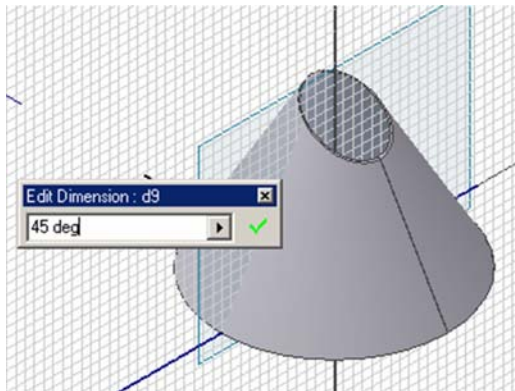


Figure 3A-5E: Applying an angular dimension.

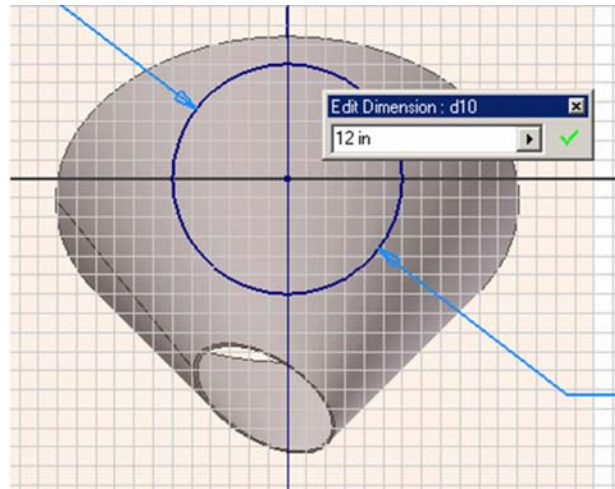


Figure 3A-5F: The sketch for the cylinder.

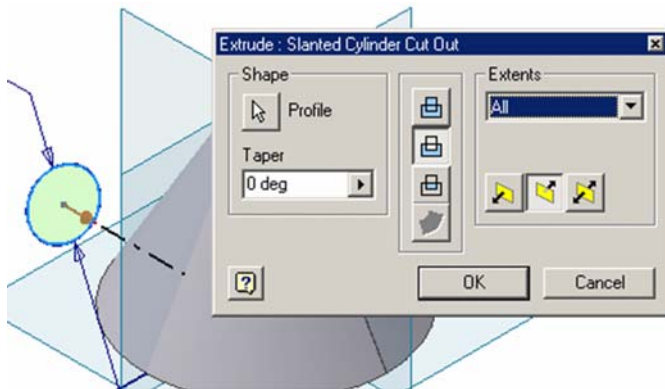


Figure 3A-5G: Cutting the cone.

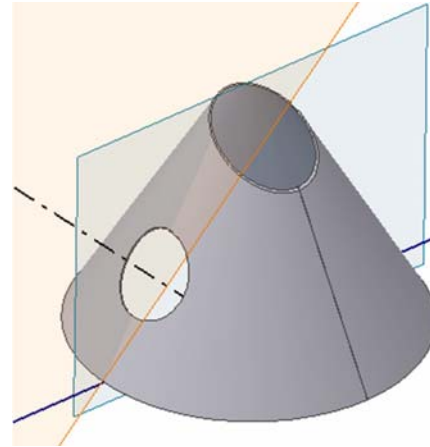


Figure 3A-5H: The cone intersected.

Next, we will intersect or cut the cone with the cylinder extruded along the slanted axis from the work plane perpendicular to the slanted axis. Click-select the *Slanted Cylinder Work Plane*, click the **Sketch** and **Look At** tool buttons; select the **Project Geometry** tool to project the **Center Point** onto the new sketch; select the **Circle Center Point** tool to draw a circle starting from the projected **Center Point** and use the **General Dimension** tool to apply a 12 inch diameter dimension (Figure 3A-5F). Click the **Return** button to exit and rename the Sketch as *Slanted Cylinder* in the **Model** panel. Go to **File**→**Save A Copy As** menu to save a copy of the file as *Tut 2-Slanted Cylinder.ipt*, in the same *Tut 2-Cone and Intersection* folder as in the Module 2. This saves another copy of the file while allowing the user to work on the same file.

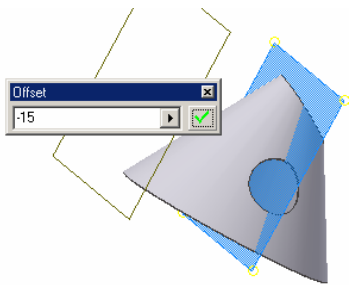
Next, select the **Extrude** tool and choose the **Cut** and **All** options, and a Direction towards the cone, to cut the truncated cone (Figure 3A-5G). The truncated cone is cut or “intersected” by the cylinder along a slanted axis on the **YZ Plane**, from a plane perpendicular to the slanted axis.

***Step 2: Cutting the truncated cone with a hexagonal prism that intersects it from a plane parallel to the YZ plane, along a straight horizontal axis perpendicular to the YZ plane***

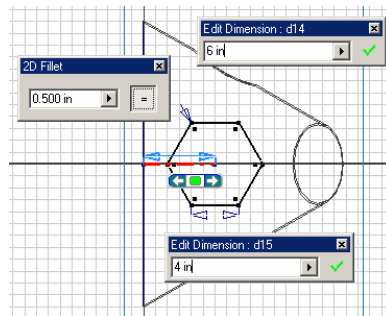
First, -select the **Work Plane** tool, click-select the **YZ plane**, hold the mouse button and drag out; type -15 and click the green check mark in the Plane text field, for a new work plane parallel to the YZ Plane at 15 inches left of the **Center Point** and along the **X-Axis** (Figure 3A-6A). Rename the new Work Plane as *Prism Work Plane* in the **Model** panel. Click-select the *Prism Work Plane* and the **Sketch** and **Look At** tool buttons to start a new sketch and to switch to an orthographic view. Select the **Project Geometry**

tool and click-select the **Center Point** feature in the model panel to project it onto the new sketch; use the **Line** tool to draw a 6-inch “central axis line” from the projected **Center Point** toward the top of the truncated cone; use the **Polygon** tool to draw a hexagon centered at the endpoint of the 6-inch line, and use the **General Dimension** tool to apply a 4-inch **Aligned Dimension** to any side of the hexagon; next, use the **Fillet** tool to apply 0.5-inch corner fillets to the corners of the hexagon (*Figure 3A-6B*). Rename the sketch as *Hex Prism*. Go to **File**→**Save A Copy As** menu to save a copy of the file as *Tut 2-Hex Prism.ipt*.

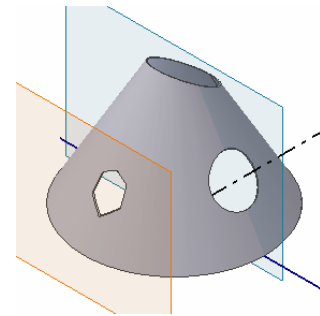
Next, use the **Extrude** tool with **Cut** option and 12 in **Distance** to cut the cone with the hexagonal prism’s volume (*Figure 3A-6C*). The cone is “intersected” by the hexagonal prism (*Figure 3A-6D*). Notice that the intersection outline on the cone still appears somehow as a hexagon as long as the size of the hexagonal shape is small compared to the size of the cone. If the opposite is true, then the intersection outline on the cone will be different, looking like a shield (*Figure 3A-6E*). To see this difference, go back to the *Hex Prism* sketch, change the dimension of the hexagon’s edge from 4 inches to 6 inches, and of the “central axis line” from 6 inches to 8 inches, and update the model. The line of intersection of the hexagon on the cone will look like a shield.



*Figure 3A-6A: The Prism Work Plane*



*Figure 3A-6B: The Hex Prism Sketch.*



*Figure 3A-6D: The cone intersected by the hexagonal prism.*

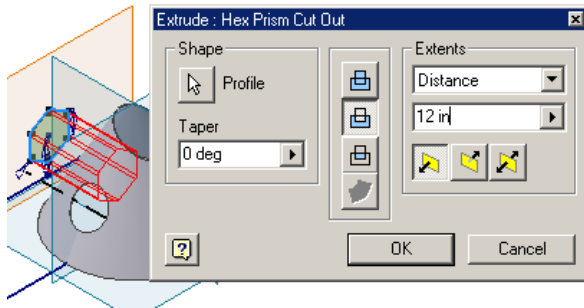


Figure 3A-6C: Cutting the cone.

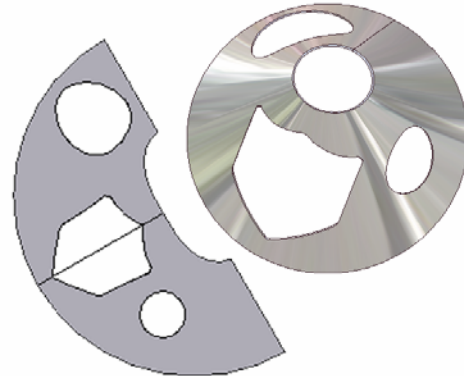


Figure 3A-6E: Intersection outline with a larger cutting hexagonal prism.

**Step 3: Cutting the truncated cone with a cylinder that intersects the truncated cone from a plane at a 45° relative to the YZ Plane and intersecting the YZ Plane at the Y-Axis**

First, select the **YZ Plane**; click the Sketch button to start a new sketch and rename it as *Axis for Angled Plane* in the **Model** panel; use the **Project Geometry** tool to project the **Center Point** onto the sketch; and use the **Line** tool to draw a vertical line from the projected **Center Point**, which corresponds to the Y Axis in the sketch (we will call this line the “Y-Axis line”); change its line **Style** to **Centerline** (Figure 3A-7A). Click-select the **Work Plane** tool; click-select the **YZ Plane** and then the “Y-Axis line” on the new sketch; an **Angle** text field appears (Figure 3A-7B); type 45 (degrees) and click the green check mark. A new work plane at a 45° angle relative to the **YZ Plane** around the Y-Axis is created (Figure 3A-7C). Rename the new work plane feature as *Angled Work Plane* in the **Model** panel.

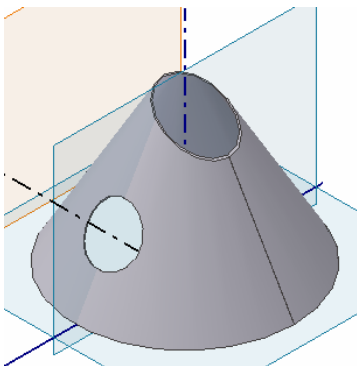


Figure 3A-7A: Vertical “Y-Axis line” on the YZ Plane.

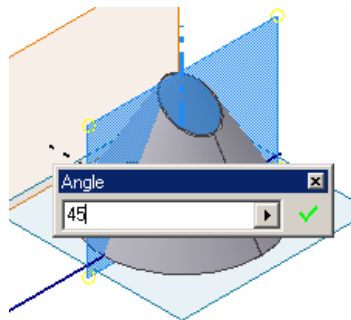


Figure 3A-7B: The Angle text field of the Work Plane tool.

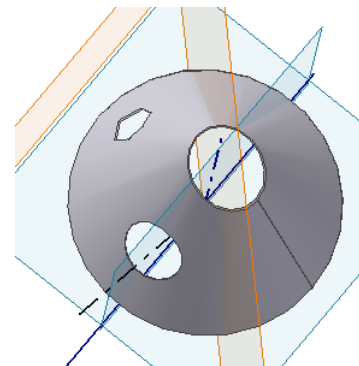
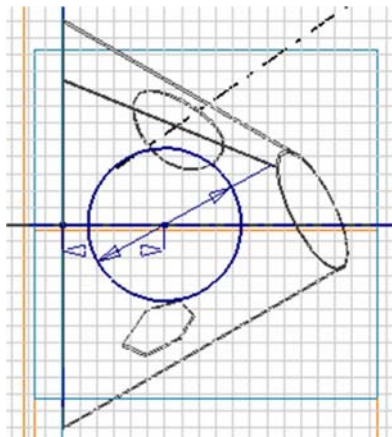


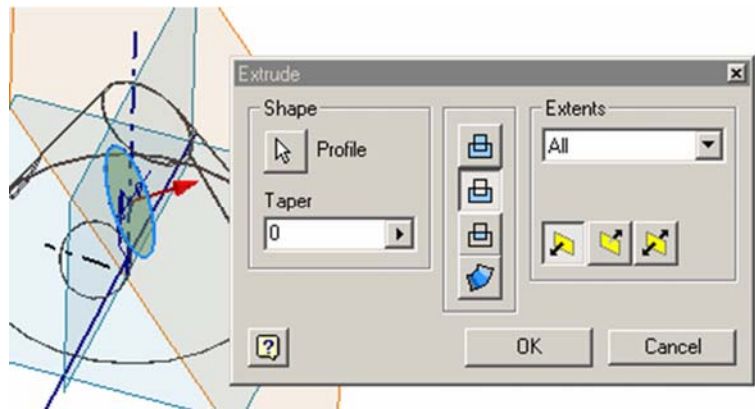
Figure 3A-7C: The angled work plane.

Next, select the *Angled Work Plane*, click the **Sketch** button from the **Command Bar** to start a new sketch; rename the new sketch as *Angled Plane Cylinder Profile* in the **Model** panel; use the **Project Geometry** tool to project the **Center Point** onto the sketch; use the **Circle Center Point** and **General Dimensions** tools to draw a 9-inch diameter circle at 6-inch distance from the projected **Center Point** (*Figure 3A-7D*). Click the **Return** button to exit the sketch. Go to the **File**→**Save A Copy As** menu to save a copy of the file as *Tut 2-Angled Plane Cylinder.ipt* in same *Tut 2-Cone and Intersection* folder.

Next, select the **Extrude** tool, choose the 9-inch diameter circle in the *Angled Plane Cylinder Profile* sketch as **Profile**, **Cut** as **Type**, **All** as **Extents** and the **Direction** outward from inside the truncated cone (*Figure 3A-7E*), so as to intersect the cone. The cone has been intersected by all three sheet metal parts (*Figure 3A-7F*). Select the conical surface and click-select the Flat Pattern tool; the flat pattern window opens (*Figure 3A-7G*). The part is completed and the features are listed in the Model panel (*Figure 3A-7I*).



*Figure 3A-7D: Angled Plane Cylinder Profile.*



*Figure 3A-7E: Intersecting the cone.*

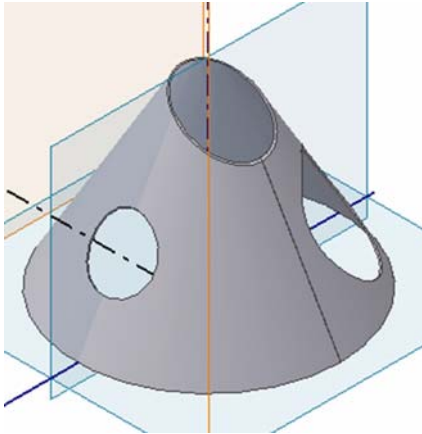


Figure 3A-7F: The cone intersected by the cylinder. The rendered model (bottom).



Figure 3A-7G: The flat pattern.

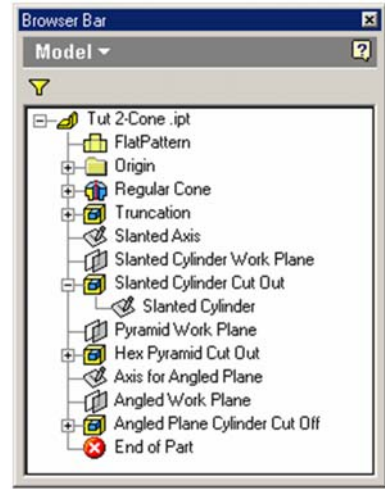


Figure 3A-7I: The Model panel features.



## Section 2:

### ***Creating the Various Sheet metal Parts That intersect the Truncated Cone***

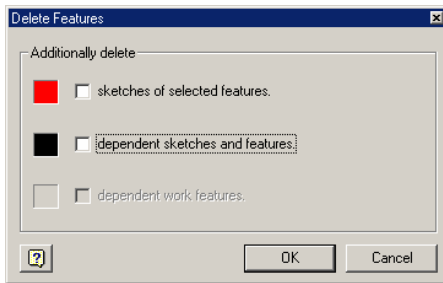
#### ***Step 1: “Cleaning up” the three “Save A Copy As” or “recycling” files***

We will first clean up all three “recycled” files that have been created previously through the **File**→**Save A Copy As** menu.

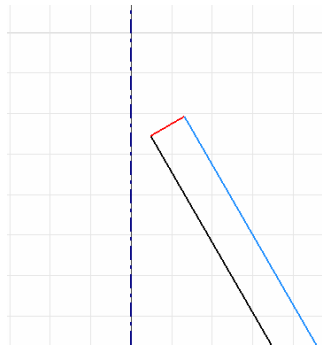
Open the files *Tut 2-Slanted Cylinder.ipt*, *Tut 2-Hex Prism.ipt* and *Tut 2-Angled Plane Cylinder.ipt*; and start deleting irrelevant features from the **Model** panel for all three files. First delete the *FlatPattern* feature. Next, delete the *Regular Cone* feature; in the **Delete Feature** dialog window, uncheck “**sketches of selected features**” and “**dependent sketches and features**” and click the **OK** button (*Figure 3A-8A*). Edit the



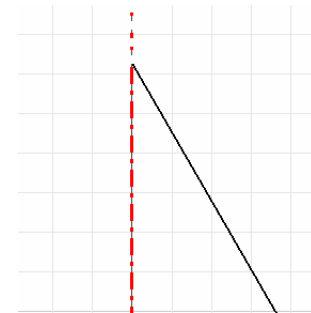
sketch *Regular Cone Profile*, use the **Delete** key on the keyboard to delete the outer wall line (shown blue in *Figure 3A-8B*), and the edge line (red); use the **Extend** and **Trim** tools to extend the angled line to the vertical axis line and trim off the tip of the axis line (*Figure 3A-18C*); use the **Line** tool draw a line connecting the bottom endpoints of the axis line and the angled line at the base of the cone (*Figure 3A-8D*). The profile for cutting the new sheet metal parts is a triangle (*Figure 3A-8E*). Next, go to the Model panel to delete the *Truncation* feature, accepting the default option of deleting “**sketches of selected feature**,” and clicking the **OK** button (*Figure 3A-8F*). Delete all other irrelevant features, so that only the *Regular Cone Profile* for revolving cut, the work planes (as well as the axis sketch used to create the angled work plane) and sketches for the new sheet metal part remain (*Figure 3A-8G*).



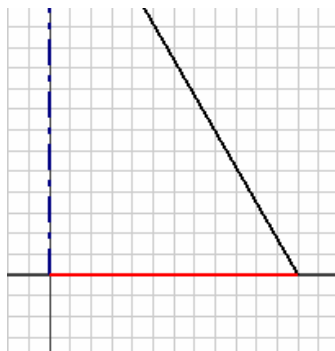
*Figure 3A-8A: The Delete Feature dialog window.*



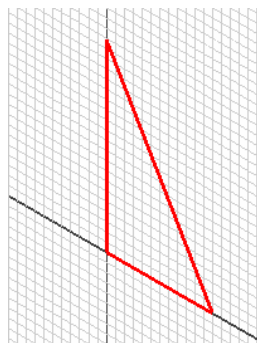
*Figure 3A-8B: Delete the unneeded line segments.*



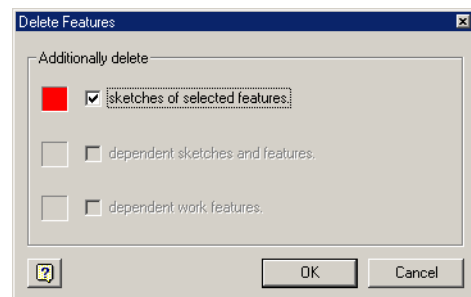
*Figure 3A-8C: Extending and trimming.*



*Figure 3A-8D: The horizontal base line.*



*Figure 3A-8E: The profile for revolve cut.*



*Figure 3A-18F: Deleting the Truncation feature and sketch.*

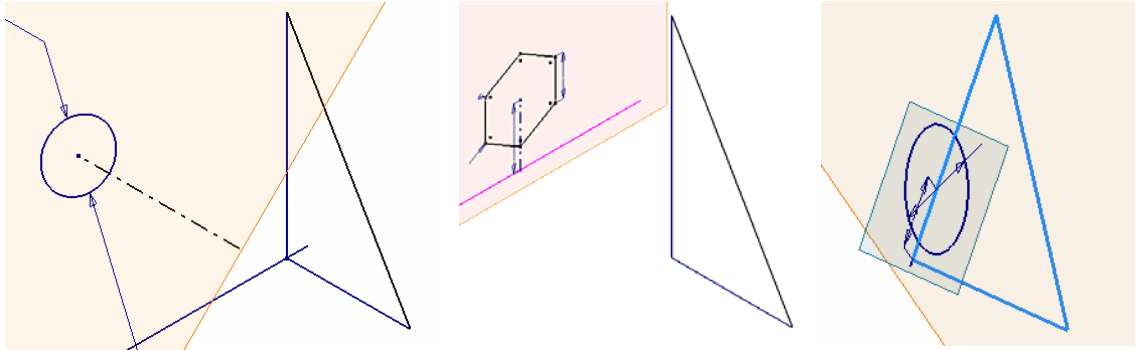


Figure 3A-8G: The remaining features for Tut 2-Slanted Cylinder.ipt (left), Tut 2-Hex Prism.ipt (middle) and Tut 2-Angled Plane Cylinder.ipt (right).

**Step 2: Creating the basic 3D models of sheet metal parts wrapping the volumes of a slanted cylinder and a hexagonal prism that intersect the truncated cone by “recycling” existing files**

We will now create the new sheet metal parts separately in the three “recycled” files. In the *Tut 2-Slanted Cylinder.ipt* file, select the **Extrude** tool and the circular profile, and extrude a 20-inch cylinder towards the triangular profile of the cone (*Figure 3A-9A*). Rename this feature as *Slanted Cylinder* in the **Model** panel. Next, select the **Shell** tool and apply to the cylinder a **Shell** feature with 0.12 in **Thickness** and both top and base surfaces removed (*Figure 3A-9B*). Rename the **Shell** feature as *Shell for Slanted Cylinder*. The basic cylindrical tube is complete. Next cut an opening in the tube for flat pattern development. Select the edge surface of the part (*Figure 3A-9C*); and click the **Sketch** button to start a new sketch; use the **Project Geometry** tool to project the **Center Point** onto the sketch; and use the **Line** tool to draw a horizontal line starting from the projected Center Point, and change its line **Style** to construction (*Figure 3A-9D*); use the **Offset** tool to offset this line to both sides at 0.005 inch distance for a total width of opening of 0.01 inch; and use the **Line** tool to draw two short vertical lines to connect them into a rectangle (*Figure 3A-9E*). Rename this Sketch as *Opening*. This is the profile used to cut a 0.01inch opening. Next, select the **Extrude** tool and choose **Cut** as **Type** and **Full** as **Extents**, the rectangle in the *Opening* sketch, and a **Direction** with an arrow pointing to the other edge surface, cut an opening on the 3D model. Rename this new **Extrude** feature as *Opening for Pattern*.

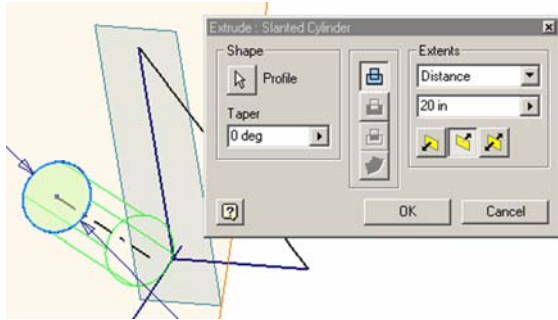


Figure 3A-9A: Creating the cylinder.

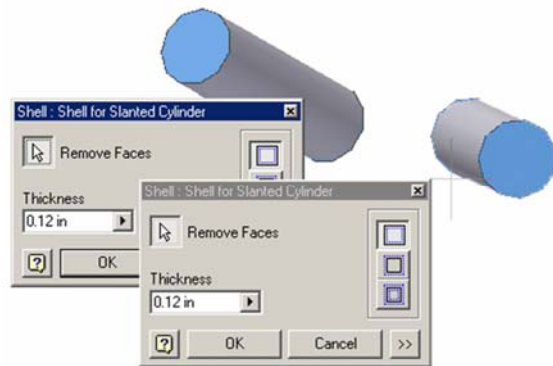


Figure 3A-9B: Shell the cylinder.

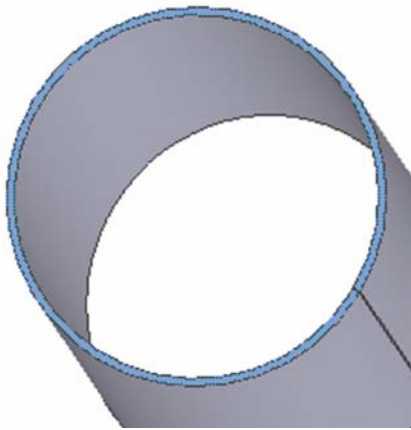


Figure 3A-9C: The edge surface.

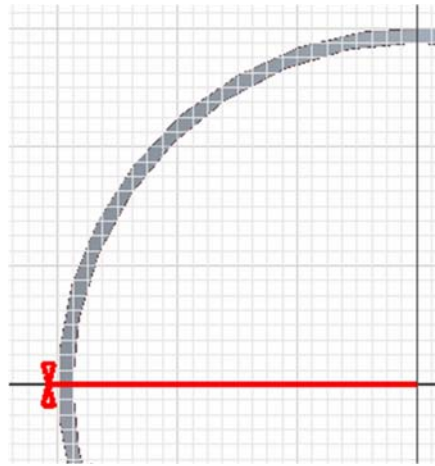


Figure 3A-9D: The construction line.

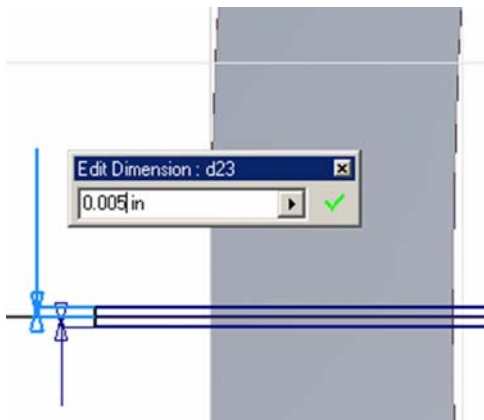


Figure 3A-9E: The profile for an opening

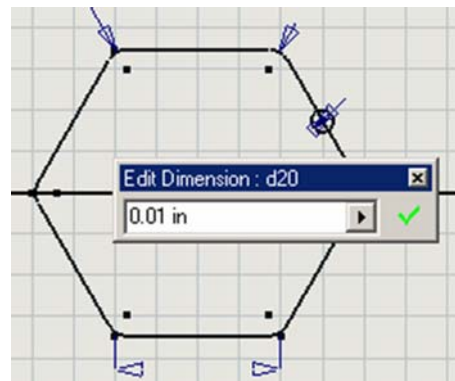
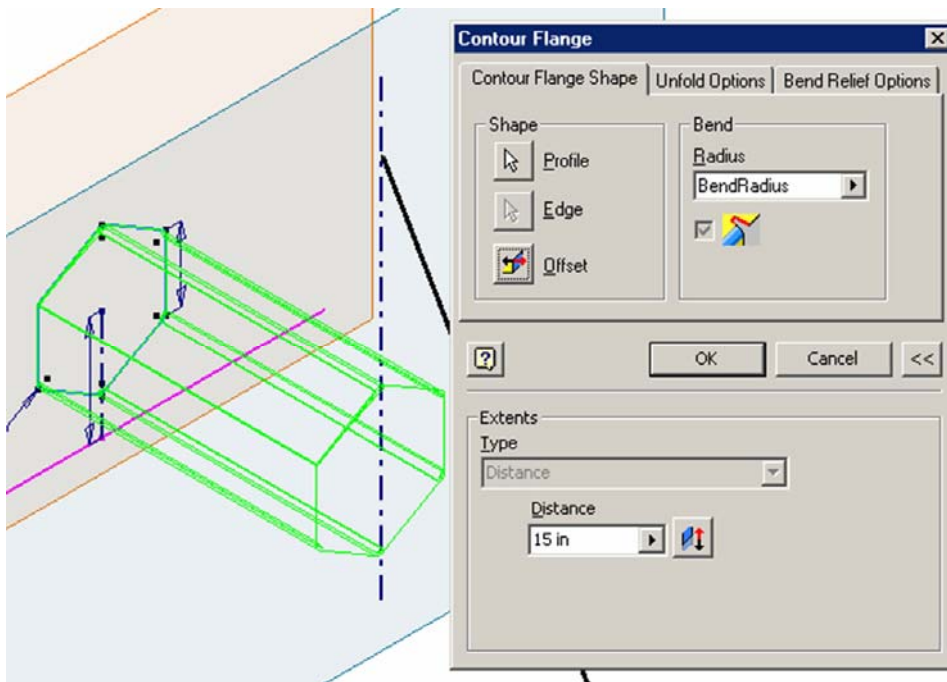


Figure 3A-10A: The construction circle.

In the *Tut 2-Hex Prism.ipt* file, select the *Hex Prism Profile* sketch feature in the **Model** panel, right-click for the shortcut menu and choose the **Edit Sketch** option; use the **Project Geometry** tool to project the edge line of one side of the polygon; select the **Circle Center Point** tool and pick the midpoint snap of the projected edge line as the center to draw a construction circle, and use the **General Dimension** tool to apply a 0.01 inch diameter to the circle (*Figure 3A-10A*); next, select the **Trim** tool and use the circle as trimming edge to trim off the portion of the edge line inside of the circle, and delete the circle with the **Delete** key of the keyboard; click the **Return** button to exit the sketch. Next, click-select the **Contour Flange** tool from the **Sheet Metal** panel; in the tool's dialog window, click the **Profile** button and click-select the *Hex Prism Profile*; green geometry outline appears on the screen; click the **Offset** button and watch how the green outline shape changes to make sure that the contour flange is falling inside of the profile; type 15 (inches) in the **Distance** text field, and click the **Flip Direction** button on its right side and watch how the green outline shape changes to make sure that the **Contour Flange** feature is projected towards the cone (with the help of **Rotate** tool, you can determine that 15 inch is about the right distance for the hexagonal prism to sufficiently reach the inside space of the cone); click the **OK** button (*Figure 3A-10B*) to create the Contour Flange feature; and rename this feature *Hex Prism* in the Model panel. The basic sheet metal part is complete.



*Figure 3A-10B: Contour Flange tool window.*

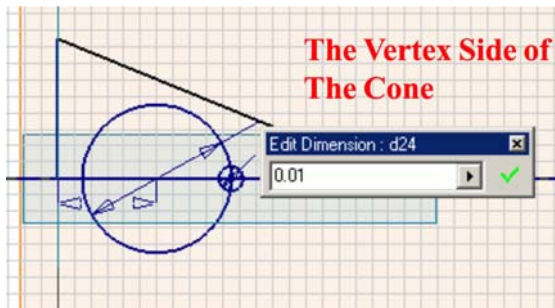


Figure 3A-11A: Cutting an opening in the circular profile used to create the sheet metal part wrapping the 3D volume of a cylinder.

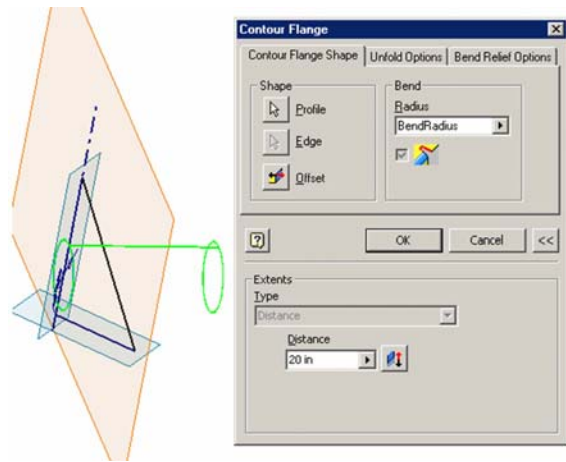


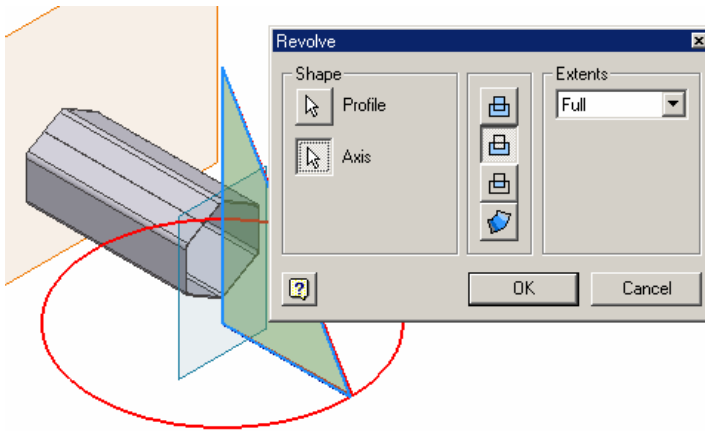
Figure 3A-11B: Creating the cylindrical sheet metal part.

In the *Tut 2-Angled Plane Cylinder.ipt* file, right-click the *Angled Cylinder Profile* sketch feature in the **Model** panel and choose the **Edit Sketch** option; click the **Look At** button for an orthographic view. Next, use the **Project Geometry** tool to project the **Center Point** onto the sketch; use the **Line** tool to draw a horizontal construction line starting from the projected **Center Point** and extending beyond the circle's edge on the vertex side of the cone; select the **Trim** tool to trim this horizontal construction line against the circular profile for the cylinder (the "cylinder circle") at the point of intersection between the circular profile and the construction line, giving an endpoint snap at this point of intersection; next, draw a construction circle with a 0.01-inch diameter (Figure 3A-11A); select the **Trim** tool and use the construction circle to trim off the portion of the "cylinder circle" inside of the construction circle; delete the construction circle and the horizontal construction line; click the **Return** button to exit the sketch. Next, click-select the **Contour Flange** tool from the **Sheet Metal** panel; in the **Extents** section of the dialog window, type 20 in the **Distance** text field; click the **Offset** button to make sure that the **Contour Flange** feature is falling inside of the profile; click the **Flip Direction** button to make sure that the contour flange is projected from the inside space of the cone outwards; click the **OK** button (Figure 3A-11B). Rename this feature *Angled Plane Cylinder* in the **Model** panel.

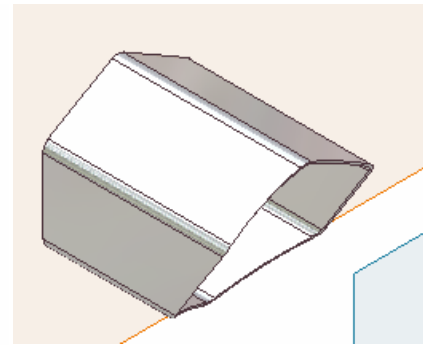
### ***Step 3: Cutting the basic 3D models of sheet metal parts wrapping the volumes of a slanted cylinder and a hexagonal prism with the volumes of the truncated cone***

Next, cut the sheet metal parts with a conical volume in all of the above three files. Select the **Revolve** tool; in the tool's dialog window, select **Cut** as **Type**, **Full** in the **Extents** pull-down menu; click the **Profile** button and click-select the triangular profile

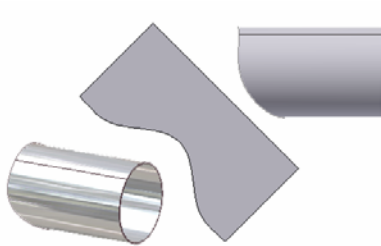
of the cone, the surface of the triangle turns light green and the outline of the conical volume is shown in dark green; next click the Axis button and then move the cursor of the mouse closer to the vertical edge line of the triangle (which corresponds to the **Y-Axis** passing through the vertex of the cone; and which line **Style** has been previously changed to **Centerline**), the axis line turns red, click the axis line and a red circle appears on the base of the triangle, indicating a cut; click the **OK** button (*Figure 3A-12A*). The basic sheet metal piece is “intersected” or cut by the cone (*Figure 3A-12B*). Rename the **Revolve** feature as *Cut from Cone* in the **Model** panel of all of the above three files. Repeat the same procedures for all three files. The complete parts and flat patterns are shown in *Figure 3A-12C*, *I-12D* and *I-12E*.



*Figure 3A-12A: The **Revolve** tool.*



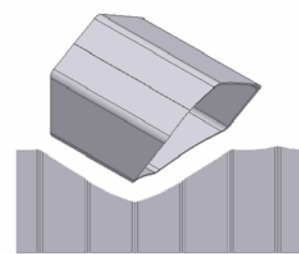
*Figure 3A-12B: The “intersected” hexagonal prism.*



*Figure 3A-12C: The intersected cylinder from an angled plane*



*Figure 3A-12D: The intersected cylinder from a plane perpendicular to a slanted axis*

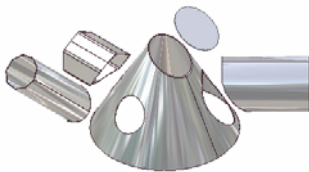


*Figure 3A-12E: The intersected from a plane parallel to the **YZ Plane**.*

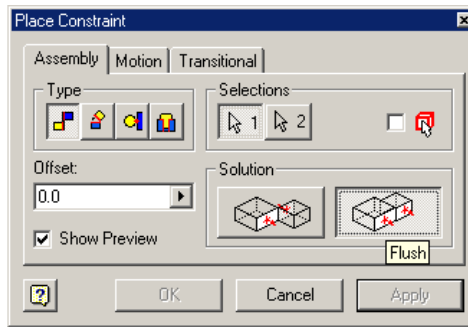
### **Section 3:** ***Assembling The Intersecting Sheet Metal Parts***

In the final step of this Module, we will learn how to use the Measure Angle, Measure Distance and Measure Area tools to make geometric calculations in Inventor. We will first assemble all parts into an Inventor assembly file, using the same methods as have been explained in the *Module 1A*.

Create a new Inventor **Standard.iam** assembly file under **English** tab; use the **Place Component** tool to import all parts into the file (*Figure 3A-13A*); next, used the **Place Constraint** tool with **Mate** as **Type** and **Flush** as **Solution** to mate pairs of all **XY**, **YZ**, and **XZ Planes** between each part and the assembly file's own **Planes** (*Figure 3A-13B*). Save the assembly file in the same folder as *Tut 2-Cone Cyl & Proisim Intersection Assembly.iam*.



*Figure 3A-13A: use the Place Component to import the parts.*



*Figure 3A-13B: All parts assembled*

### **Section 4:** ***Making Geometrical Calculations With Inventor's Measure Tools***

The assembly file is ready for making geometric calculations on Inventor 3D models. First try to measure the angle between the truncated top of the cone and the curved surface of the cone. Go to **Tools**→**Measure Angle** menu to select the **Measure Angle** tool; move the cursor closer to the surface of the truncated top of the cone, a red outline of the surface appears, click-select the surface; then move the cursor closer to the curved surface of the cone, red outline plus a center line appears, click-select a point on the right side of the cone and the text field reads **60 deg** (degrees), and a white angle symbol appears on the top face of the cone (*Figure 3A-13C*). Next, click on the right-pointing black triangle next to the text field for a shortcut menu and choose the **Reset** option. Click the top surface again and then click a point on the curved surface of the cone on the left side, the text field displays **120 deg**. This demonstrates the Inventor's

capability of measuring angles between a flat surface and a curved surface on a particular point on the surface.

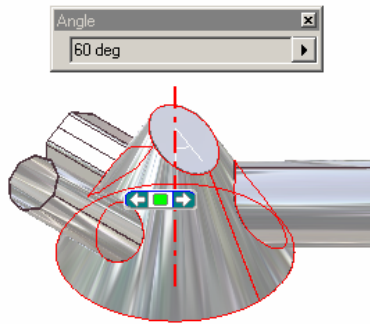


Figure 3A-13C: Calculation the angle between an axis and a conical surface.

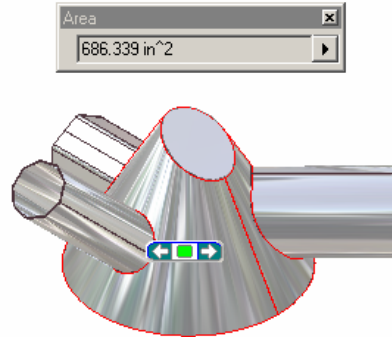


Figure 3A-13D: Calculating the area.

Next, try the **Measure Area** tool. Go to **Tools**→**Measure Area** menu, or, while the **Measure Angle** tool is still active with its window open, click the black right-pointing triangle for the shortcut menu and choose the **Measure Area** option; click-select the curved surface of the cone, the text field reads **686.339 in<sup>2</sup>**, meaning 686.339 square inches (Figure 3A-13D).

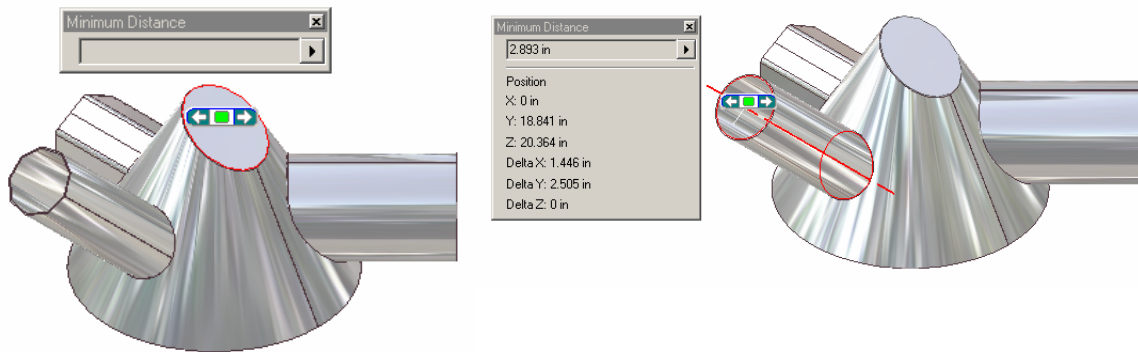


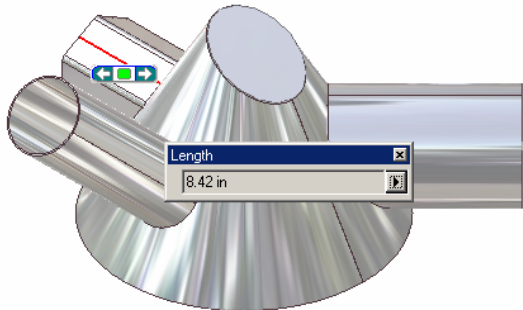
Figure 3A-13E: Measuring the **Minimum Distance** between two non-parallel planes. Picking the first plane, or the surface of the top of the cone (left); and pick the second plan, or the surface of the top sectional edge of the cylinder (right).

Next, try the **Measure Distance** tool. Go to **Tools**→**Measure Distance** menu, or, while the **Measure Angle** tool is still active, click the black right-pointing triangle for the shortcut menu and choose the **Measure Distance** option; click-select the surface of the top of the cone (the first plane); and then click-select the surface of the top section edge of the cylinder (the second plane); the text field reads **2.893 in** and the title above the text



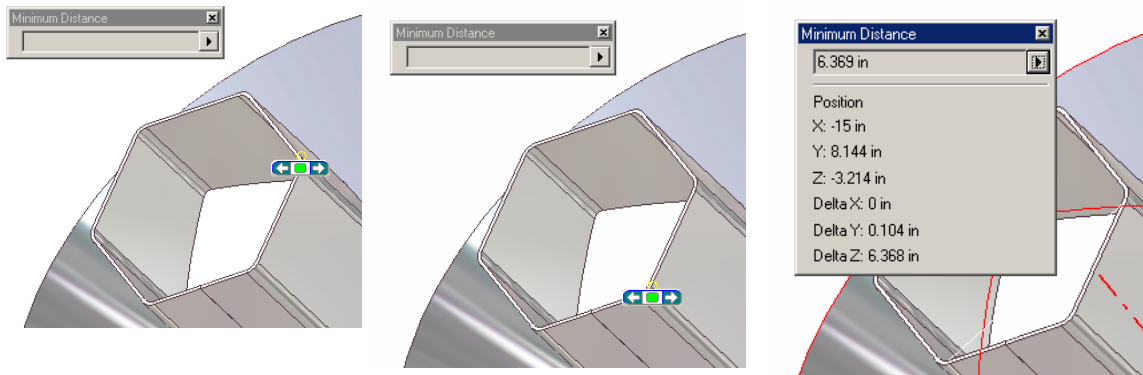
field reads **Minimum Distance**, indicating a shortest distance of 2.893 inches between the two planes (*Figure 3A-13E*).

Next, click the black right-pointing triangle for the shortcut menu and choose the **Reset** option. Click-select an edge line on the part, and the text field displays the **Length** (*Figure 3A-13F*).



*Figure 3A-13F: Measuring the **Length**.*

Next, click the black right-pointing triangle for the shortcut menu and choose the **Reset** option. Move the cursor closer to the first point on the model (*Figure 3A-13G*, left), a yellow outlined circle indicator appears, click-select the point; then move the cursor closer to the second point (*Figure 3A-13G*, middle), a yellow outlined circle indicator appears, click-select the point; the text field displays the **Minimum Distance**.



*Figure 3A-13G: Measuring the **Length** between two points on the 3D model. Picking the first point (left); picking the second point (middle); Displaying the **Minimum Distance** (right).*

This completes the Modules 3. Congratulations! You have learned how to:

- Create the 3D models and flat patterns of sheet metal parts wrapping the volume of cylinders and prisms;
- Create the 3D models and flat patterns of intersecting cones, prisms, and cylinders;
- Solving problems of minimum distance between two non-parallel planes, and between two points; of dihedral angles between a curved conic surface and a flat surface, and of area of a curved surface, in Inventor 3D modeling environment.