

Module 7A: Creating Oblique Surfaces on a 3D Model Through a Multi-Faceted Tube Connector Project

We have so far explored various ways of solving descriptive geometry problems by using the **Work Plane** tool. In this *Module*, we will learn how to create an oblique surface on a 3D model through a real-world project of designing a multi-faceted tube connector for metal structural tubes used in building construction. An oblique surface is one which true shape (abbreviated as “TS” in descriptive geometry terminology) cannot be viewed in any of the six principal views or even the first auxiliary view that is projected from an edge view (abbreviated as “EV” in descriptive geometry terminology and appearing as “a line”) of the surface from a principal view. In order to obtain the true-shape view of an oblique surface, a primary auxiliary view needs to be drawn first to turn the oblique view of the surface into an edge view (“EV”), and a secondary auxiliary view is thereafter projected from the primary auxiliary edge view to create the true-shape view of the surface, as shown in *Figure 7A-1A*. The process of drawing the true-shape (“TS”) view of an oblique surface using traditional board drafting techniques or even the 2D tools of AutoCAD is time-consuming. Fortunately, the 3D tools of AutoCAD and Inventor can be used to create the oblique surface as a 3D feature first; and the 2D working drawing tools of both programs can create the true-shape view of such oblique surface with a few mouse clicks. In this *Module*, we will learn how to use the Inventor program’s **Work Plane** and **Split** tools to create a 3D feature with an oblique surface.

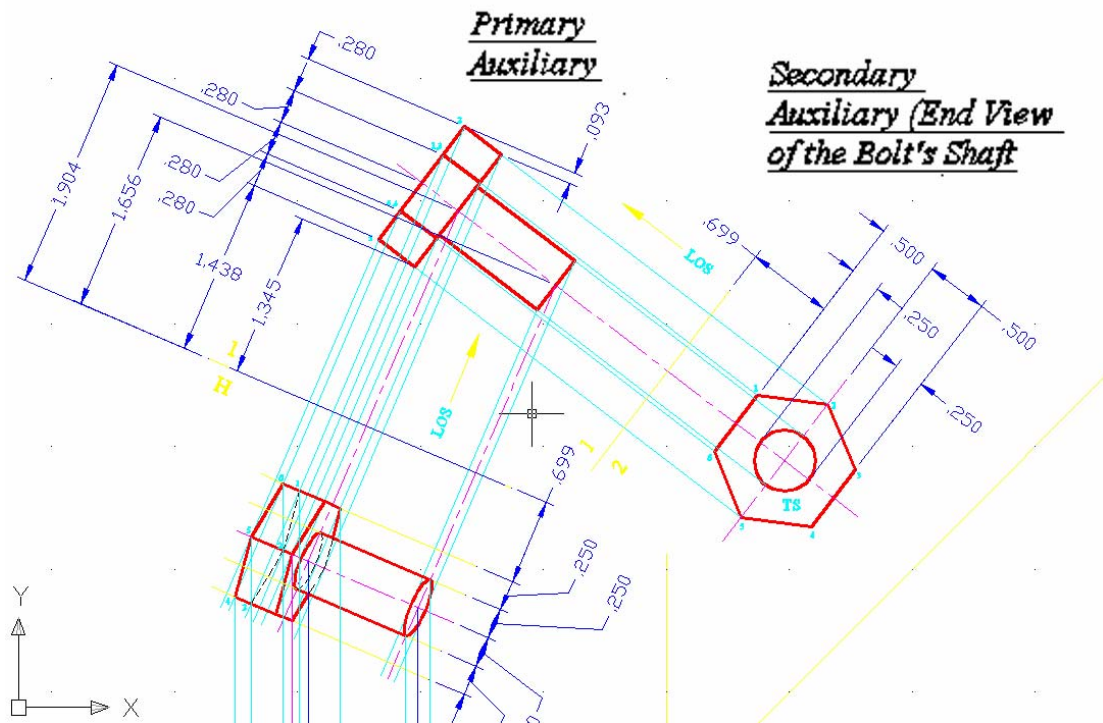


Figure 7A-1A: The bolt's shaft turned to an oblique angle in the principal view (bottom); primary auxiliary edge view (top left); and secondary auxiliary true-shape view (right).

Section 1:

Creating The Basic Polygon with Oblique Surfaces

Step 1: Creating an extruded cube without any oblique surface

We will create a 6 x 6 x 6 cubic-inch cube first. Launch Inventor, start a new **Sheet Metal (in).ipt** file under the **English** tab. Turn **Visibility** on for the **XY Plane**. "Sketch1" is created by default on the **XY Plane** (the one parallel to your computer's screen); rename *Sketch1* as *Square Sketch* in the **Model** panel; use the **Project Geometry** tool to project the **Center Point** onto the sketch for a snap point; use the **Polygon** and **General Dimension** tools to draw a 6-inch square snapped-centered at the projected **Center Point** (Figure 7A-1B); click the **Return** button to exit the Sketch. You can use the **Two Point Rectangle** instead of the **Polygon** tool to draw the square; however, for obvious reason of convenience that I will let you investigate on your own, we will use the **Polygon** tool in this case.

Next, use the **Extrude tool** with **Join as Type**, 6 in for **Distance** and **Midplane** for **Direction** to create the cube with the *Square Sketch*, and rename the **Extrude** feature **Cube** in the **Model** panel (*Figure 7A-1C*).

Next, use the **Extrude tool** with **Join as Type**, 6 in for **Distance** and **Midplane** for **Direction** to create the cube with the *Square Sketch*, and rename the **Extrude** feature **Cube** in the **Model** panel (*Figure 7A-1C*).

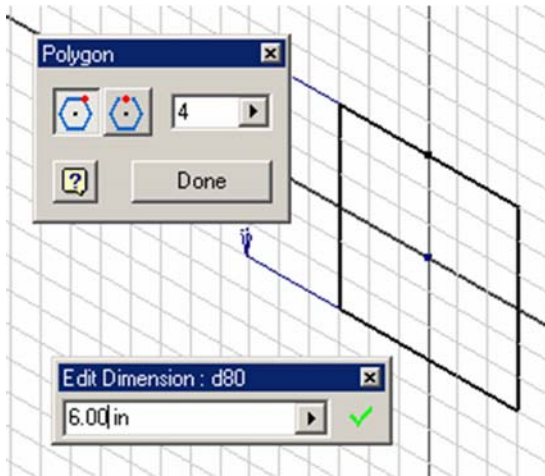


Figure 7A-1B: The Square Sketch.

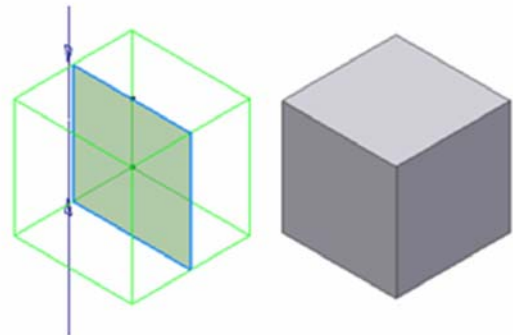
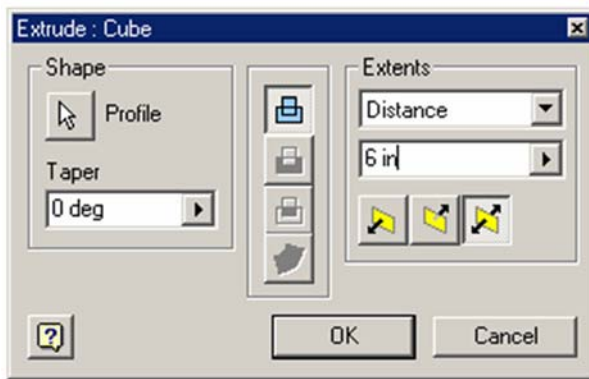


Figure 7A-1C: The Cube.

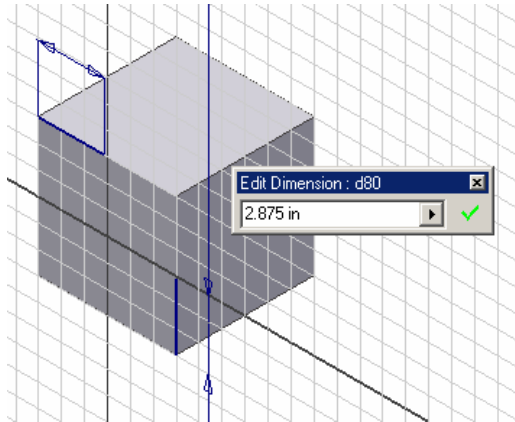


Figure 7A-1D: The *Oblique Corner Construction 1* sketch.

Step 2: Creating two sketches with three corner points and an oblique Work Plane defined by three points

We will create two sketches on two surfaces of the cube so as to provide 3 corner points for the creation of an oblique **Work Plane** feature.

Select the frontal surface of the cube and click the **Sketch** button to start a new sketch; rename it *Oblique Corner Construction 1* in the **Model** panel; use the **Project Geometry** tool to project the upper-left and lower-right corner points of the frontal surface; then use the **Line** tool to draw a horizontal line starting from the projected upper-left corner point rightwards and a vertical line starting from the projected lower-right corner point upwards; then use the **General Dimension** tool to apply a 2.875-inch **Linear Dimension** to both lines; click the **Return** button to exit the *Oblique Corner Construction 1* sketch (Figure 7A-1D).

Next, select the right surface of the cube and click the **Sketch** button to start a new sketch; rename it *Oblique Corner Construction 2* in the **Model** panel; use the **Project Geometry** tool to project the upper-right corner point of the right surface onto the sketch; use the **Line** tool to draw a horizontal line starting from the projected corner point leftwards; use the **General Dimension** tool to apply a 2.875-inch **Linear Dimension** to the line; and click the **Return** button to exit the *Oblique Corner Construction 2* sketch (Figure 7A-1E).

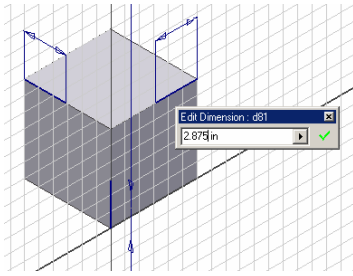


Figure 7A-1E: The Oblique Corner Construction 2 sketch.

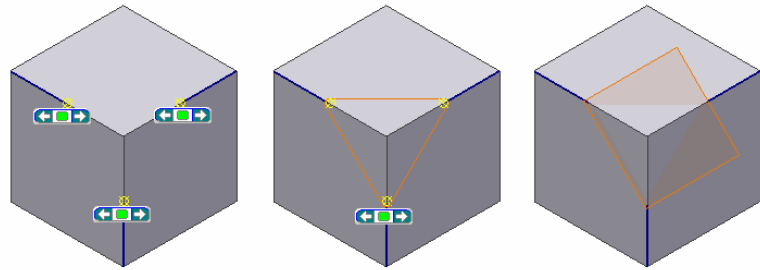


Figure 7A-1F: Creating the 3P Oblique Work Plane.

Next, select the **Work Plane** tool and click-select the three endpoints (they turn red when the cursor is moving closer to them) to create an oblique **Work Plane** feature (Figure 7A-1F); and rename it *3P Oblique Work Plane* in the **Model** panel.

Step 2: Cutting an oblique surface by removing an oblique corner from the cube

Select the **Split** tool from the **Features** panel; in the **Split** tool dialog window, click the **Split Part** button in the **Method** section to select the option; click the **Split Tool** button and then click-select the *3P Oblique Work Plane* either from the **Model** panel or on the 3D model; a red outline appears around the *3P Oblique Work plane* and a red arrow points to the direction on the side where the part will be removed; if the arrow points to the wrong direction, then click either of the **Direction** buttons in the **Remove** section to change it; click the **OK** button to remove the oblique corner (Figure 7A-1G and Figure 7A-1H). Rename the **Split** feature *Oblique Corner Split* in the **Model** panel.

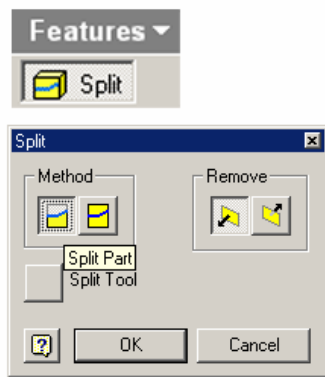


Figure 7A-1G: Cutting off the oblique corner with the **Split** tool from the **Features** panel.

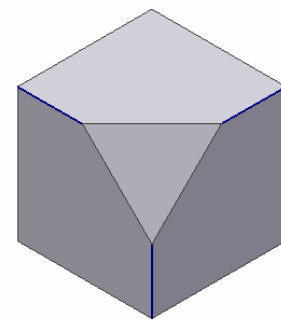
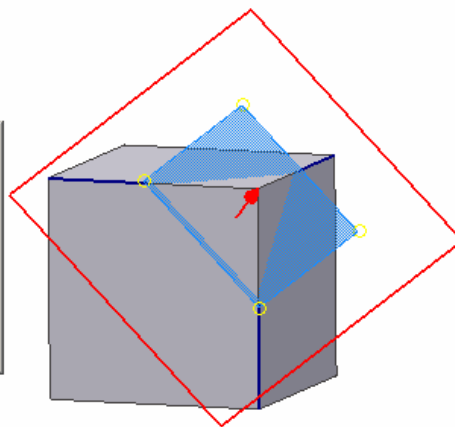


Figure 7A-1H: The oblique surface corner removed from the cube.

Step 3: Duplicating oblique surface and corners on the cube

We will duplicate the oblique surface and corner on the cube with the **Circular Pattern** and **Mirror Feature** tools from the **Features** panel.

First, select the **Circular Pattern** tool; in the **Circular Pattern** tool's dialog window, type 4 ul in the **Count** text field and 360.00 deg in the **Angle** text field; click the **Features** arrow button and then click-select the *Oblique Corner Split* feature from the **Model** panel; next, click the **Rotation Axis** arrow button and then the **Y Axis** feature from the **Model** panel, green outlines for the circularly patterned oblique corners' geometry and blue axis and arrowed symbol of rotation appear around the cube; click the **OK** button to create the **Circular Pattern** feature with 4 oblique corner splits (*Figure 7A-2A*); and rename the feature *Oblique Corner Pattern* in the **Model** panel.

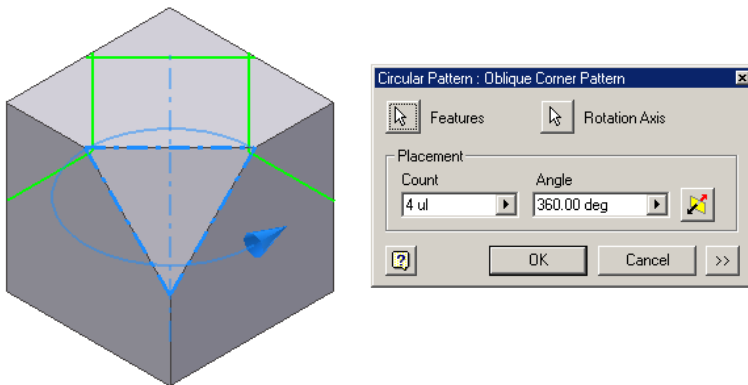


Figure 7A-2A: The Circular Pattern tool.

Next, select the **Mirror Feature** tool; in the **Mirror Feature** tool's dialog window, click the **Features** button and then click-select the *Oblique Corner Pattern* feature in the **Model** panel; click the **Mirror Plane** button and then click-select the **XZ Plane** from the **Model** panel; green outlines for the mirrored oblique corners' geometry and blue outlines for the selected features and solid blue selected mirror plane appear on the 3D model; click the **OK** button to create the mirrored oblique corners (*Figure 7A-2B*). Rename the **Mirror Feature** *Oblique Corner Mirror* in the **Model** Panel (*Figure 7A-2C*). All oblique corners have been created on the cube; and the 3D model is ready for the creation of multi-faceted tube holders.

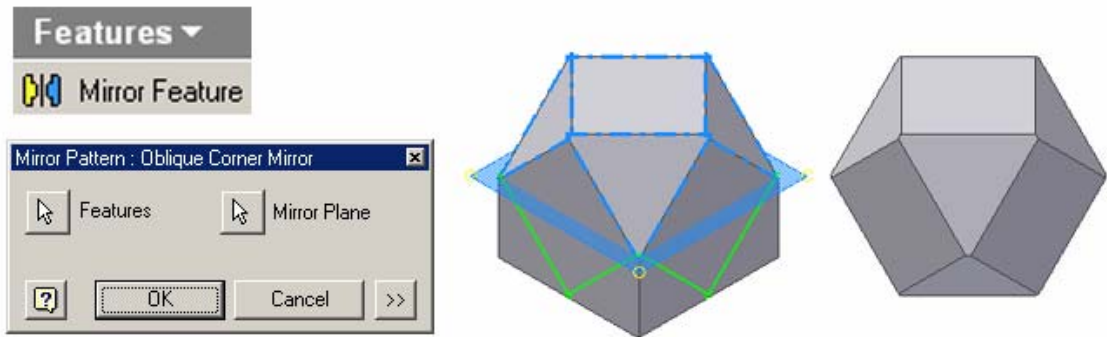


Figure 7A-2B: The **Mirror Feature** tool.

Next, use a variety of tools explored so far, such as **Work Plane**, **Project Geometry**, **Line**, **Circle Center Point**, **Tangent**, **Extrude**, **Circular Pattern**, **Mirror Feature**, **Fillet**, etc., create all tube holders on all square and triangular surfaces of the 3D model created so far, similar but not exactly the same as what is shown in Figure 7A-2D. Use your own creativity and imagination.

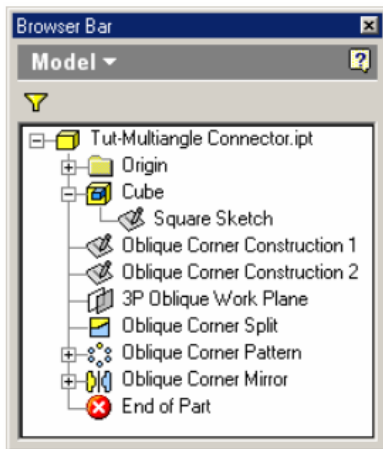


Figure 7A-2C: The features listed in the **Model** panel.

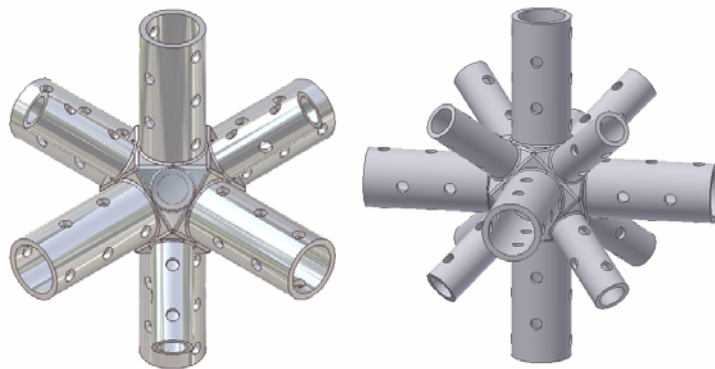


Figure 7A-2D: The final product.

Section 2: **Adding Features On The Oblique Surfaces**

In this Section of the Module 7A, we will discuss how to add features on oblique surfaces.

Step 1: Creating tube holders on an oblique surfaces

We will create a tube holder on one of the oblique surfaces, and then use the **Circular Patterns** and **Mirror Features** tools to add copies of the tube holders to all oblique surfaces.

While the 3D model is in **Isometric** view, select the oblique surface in the center of the model (any of the oblique surfaces will work; but for convenience, we select this particular one); click the **Sketch** button from the **Command Bar** to start a new sketch; use the **Project Geometry** tool to project the triangular edge lines onto the new sketch; use the **Circle Center Point** tool to draw a circle centered approximately at the center of the triangle and with a diameter that places the circle within the triangle (*Figure 7A-3A*); select the **Tangent** tool, click-select one of the edge of the triangle and the circle, the circle becomes “tangent” to or touches the edge of the triangle; select another edge of the triangle and the circle again, the circle becomes “tangent” to or touches two edges of the triangle (*Figure 7A-3B*); select the third edge of the triangle and the circle again, the circle not only becomes “tangent” to or touches all three edges of the triangle, its size enlarges as well; this circle provides a center point snap for additional concentric circles to be drawn; next, use the **Circle Center Point** and **General Dimensions** tools to draw two concentric circles of 1.75 inches and 2.375 inches diameters respectively (*Figure 7A-3C*); select the first circle and go to the **Command Bar** to change its line **Style** to **Construction** (or select the first circle together with the triangle’s edge lines and use the **Delete** key on the keyboard to delete all of them); click the **Return** button to complete the sketch; and rename the **Sketch** feature as *Tube Holder Sketch* in the **Model** panel.

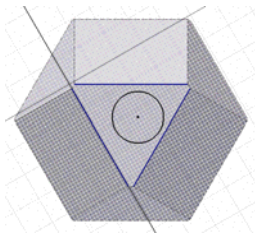


Figure 7A-3A:
Drawing a circle
within the
triangle.

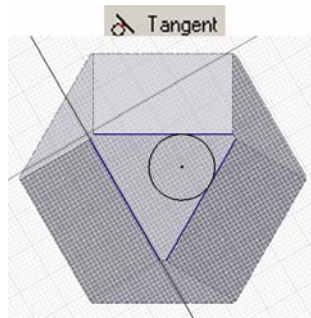


Figure 7A-3B:
Applying tangent
constraint to the
circle and the edge
of the triangle.

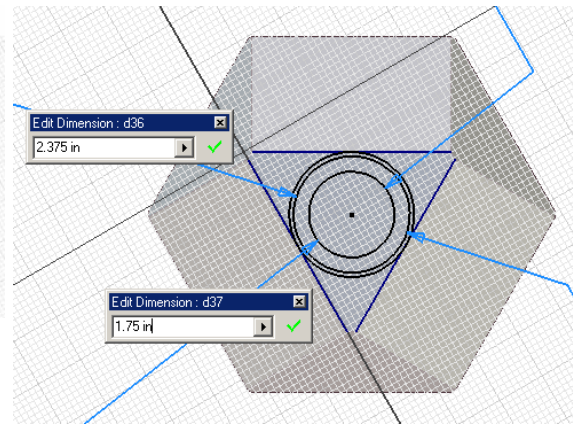


Figure 7A-3C: Changing the size of the first circle with Tangent tool and drawing additional concentric circles in the Tube Holder Sketch.

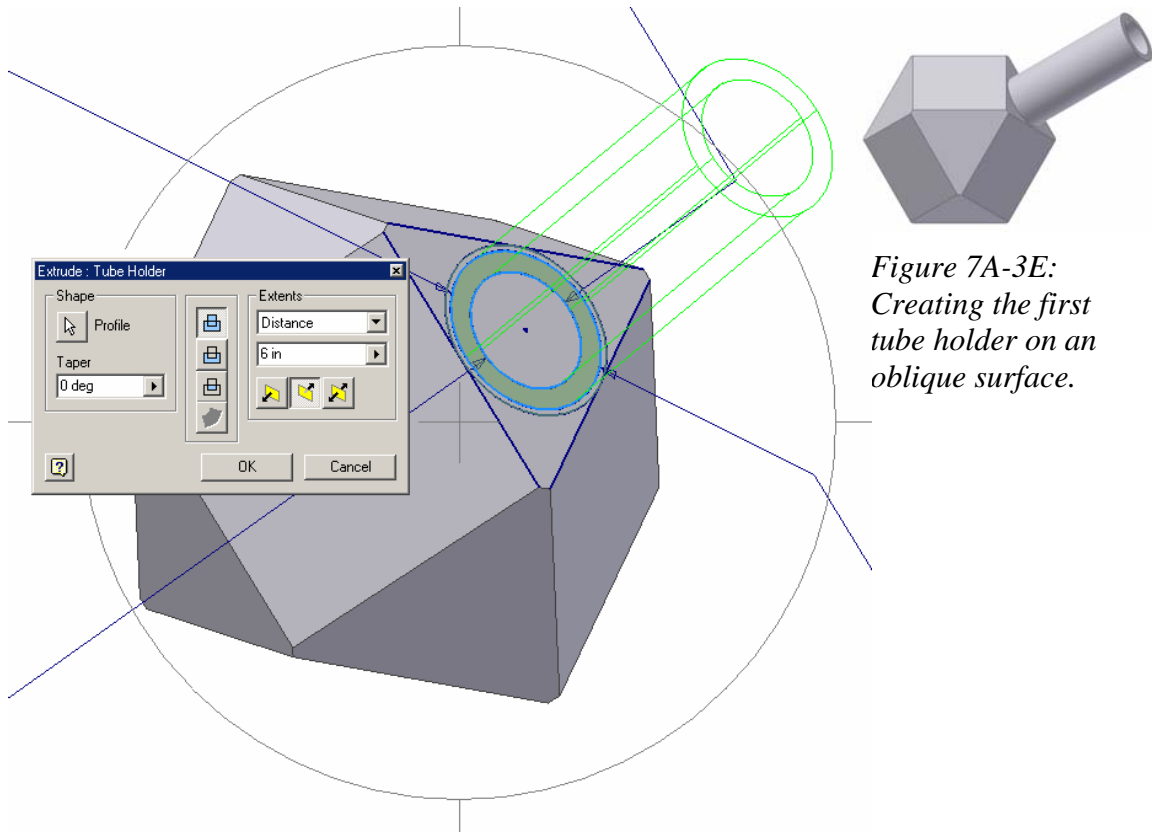


Figure 7A-3E:
Creating the first
tube holder on an
oblique surface.

Figure 7A-3D: Using the Extrude tool.

Next, select the **Extrude** tool; choose the **Join** as **Type**, type 6 in as **Distance** for **Extents**, the space between the two concentric circles of 1.75 inches and 2.375 inches diameters as **Profile**, and outward from the oblique surface as **Direction** (*Figure 7A-3D*), create the extruded tube holder and rename the **Extrude** feature in the **Model** panel as *Tube Holder* (*Figure 7A-3E*).

Step 2: Creating holes on the tube holder for fasteners

To create fastener holes on the wall of the tube holder, we need to first create two perpendicular work planes passing through the central axis of the tube holder, on which profiles for the fastener holes can be drawn. These work planes can be defined by three points from two construction sketches drawn on the top and base surfaces of the tube holder. The steps are explained as follows.

First, create two construction sketches to provide three points for the definition of work planes. Select the top surface of the tube holder and click the **Sketch** button to start a new sketch; rename the sketch as *Screw Hole Construction 1* in the **Model** panel; use the **Project Geometry** tool to project the circular outer edge of the tube holder onto the sketch; use the **Line** tool to draw two perpendicular lines centered at the center of the projected circle and extending beyond the circle's circumference (the two lines should be parallel to grid lines), with the help of Inventor's **perpendicular** and **Parallel** indicators (an alternative method is to use the **Perpendicular** and **Parallel** tools to constrain the lines drawn); use the **Trim** tool to trim off the excess line segments beyond the circle (*Figure 7A-3E*); click the **Return** button to exit the **Sketch** mode. Next, select the triangular oblique surface at the base of the tube holder and click the **Sketch** button to start a new sketch; rename the sketch as *Screw Hole Construction 2* in the **Model** panel; use the **Project Geometry** tool to project the profile of the *Screw Hole Construction 1* onto the sketch (*Figure 7A-3F*); click the **Return** button to exit the **Sketch** mode.

Next, create the work planes for the fastener holes' profiles. For better visualization, first go to the **Command Bar** to switch to **Wireframe Display** mode. Select the **Work Plane** tool; move the mouse closer to each of the three points from both *Screw Hole Construction 1* and *Screw Hole Construction 2* sketches, as shown in *Figure 7A-3G*, and click once when the red snap point indicator appears; a new **Work Plane** feature is created (*Figure 7A-3H*); rename it *Screw Hole Plane Work Plane 1* in the **Model** panel. Use the **Work Plane** tool again and move the mouse closer to each of the three points from both sketches, as shown in *Figure 7A-3J*, and click once when the red snap point indicator appears; a new **Work Plane** feature is created (*Figure 7A-3K*); rename it *Screw Hole Plane Work Plane 2* in the **Model** panel.

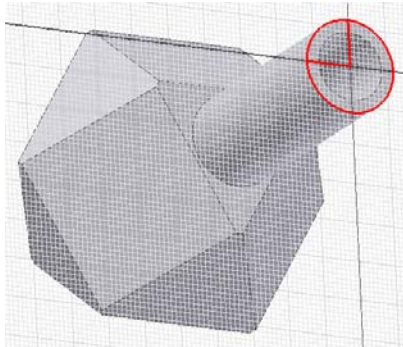


Figure 7A-3E: The Screw Hole Construction 1 profile on the top surface of the tube holder.

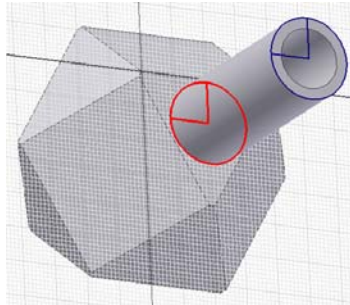


Figure 7A-3F: Using the Project Geometry tool to create the Screw Hole Construction 1 profile on the base of the tube holder.

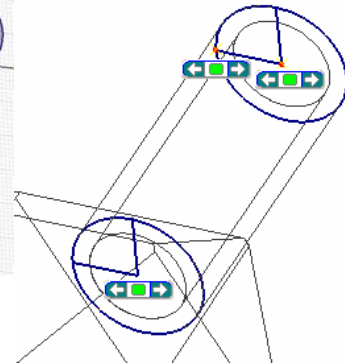


Figure 7A-3G: Picking the three points from both sketches to define a new work plane.

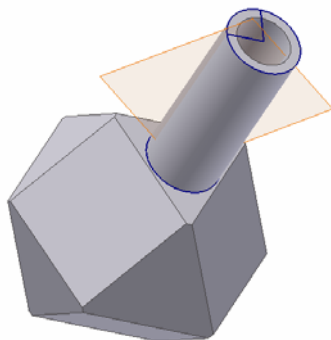


Figure 7A-3H: Creating Screw Hole Plane Work Plane 1.

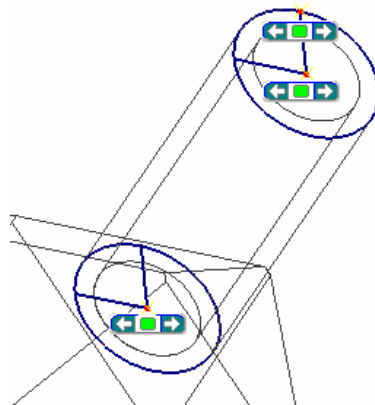


Figure 7A-3J: Picking the three points from both sketches to define the second work plane for fastener holes.

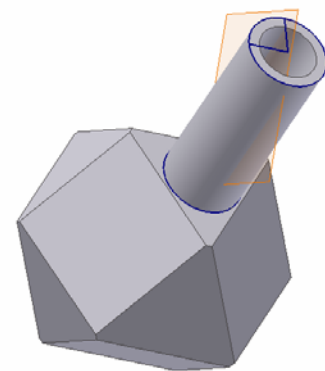


Figure 7A-3K: Creating Screw Hole Plane Work Plane 2.

Next, create the profile sketches on the work planes and **Extrude** cut features for fastener holes. Select the *Screw Hole Plane Work Plane 1*(Figure 7A-3L), click the **Look At** button to switch to a normal orthographic view, and click the **Sketch** button to start a new sketch; rename it *Screw Hole Sketch 1* in the **Model** panel; use the **Project Geometry** tool to project the top edge of the tube holder onto the sketch; use the **Offset** and **General Dimension** tool to create an offset line 1.5 inches from the projected top edge line, and another offset line at 3.0 inches from the first offset line (the two offset lines provide midpoint snaps for the center of the fastener holes); next, use the **Circle**

Center Point and **General Dimension** tools to create two 0.75-inch diameter circles, as shown in *Figure 7A-3M*. Next, use the **Extrude** tool with **Cut** for **Type**, **All** for **Extents**, **Midplane** for **Direction**, and the two circles for **Profile**, create the two fastener holes (*Figure 7A-3N*); rename the new **Extrude** feature 2 *Screw Holes On Plane 1* in the **Model** panel.

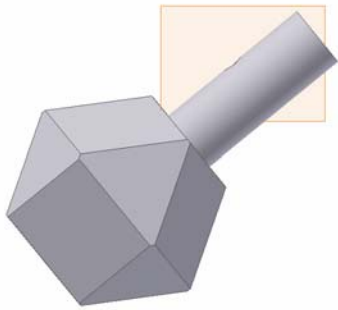


Figure 7A-3L: Selecting the Screw Hole Plane Work Plane 1.

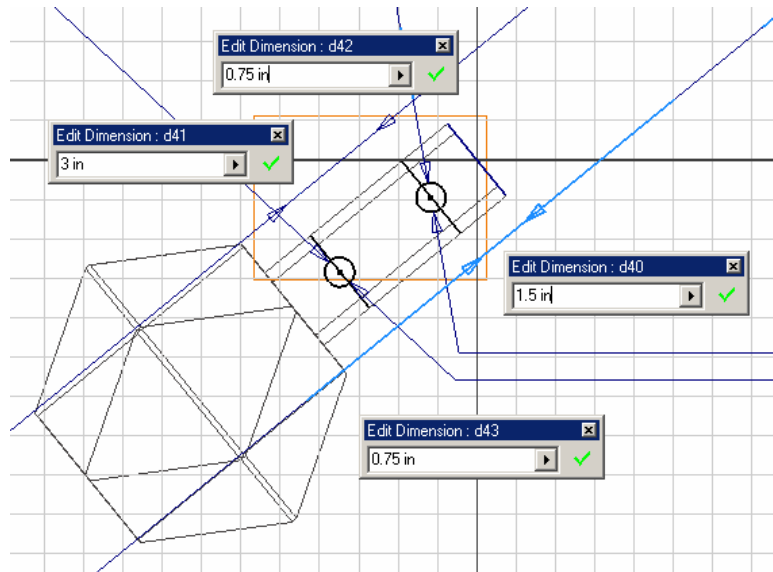


Figure 7A-3M: The Screw Hole Sketch 1.

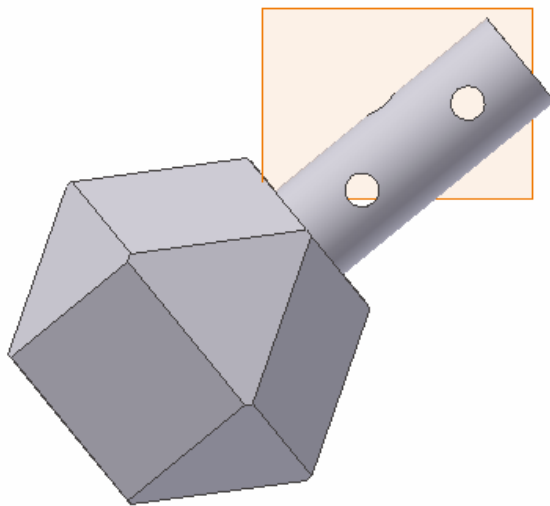


Figure 7A-3N: Creating two fastener holes.

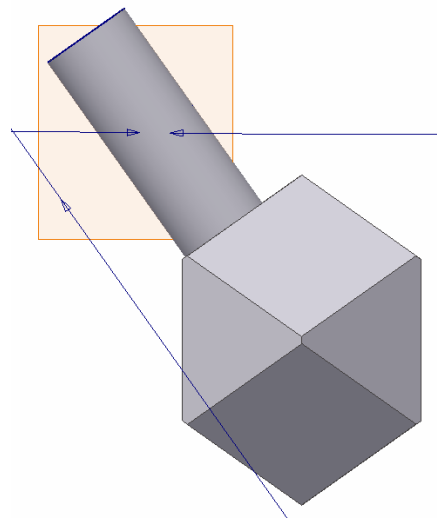
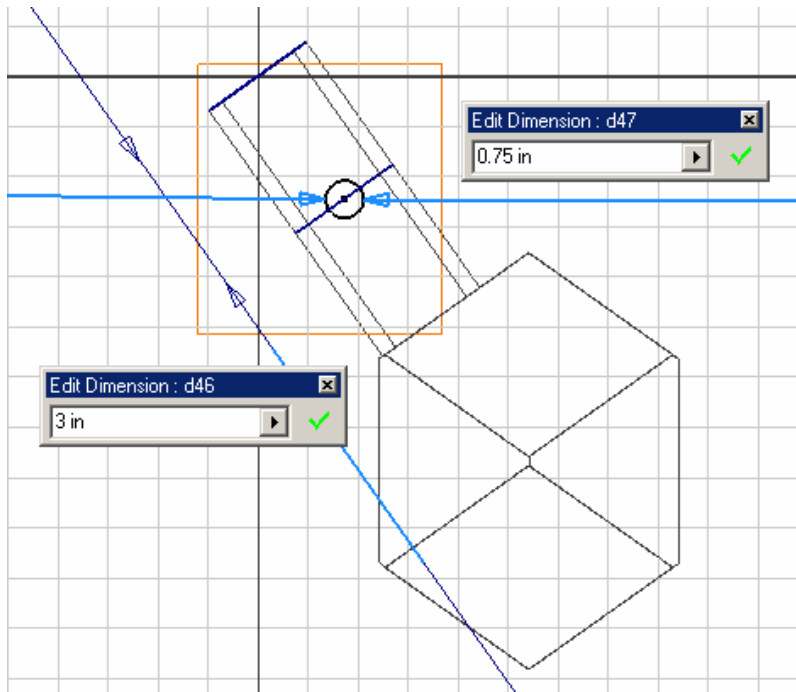
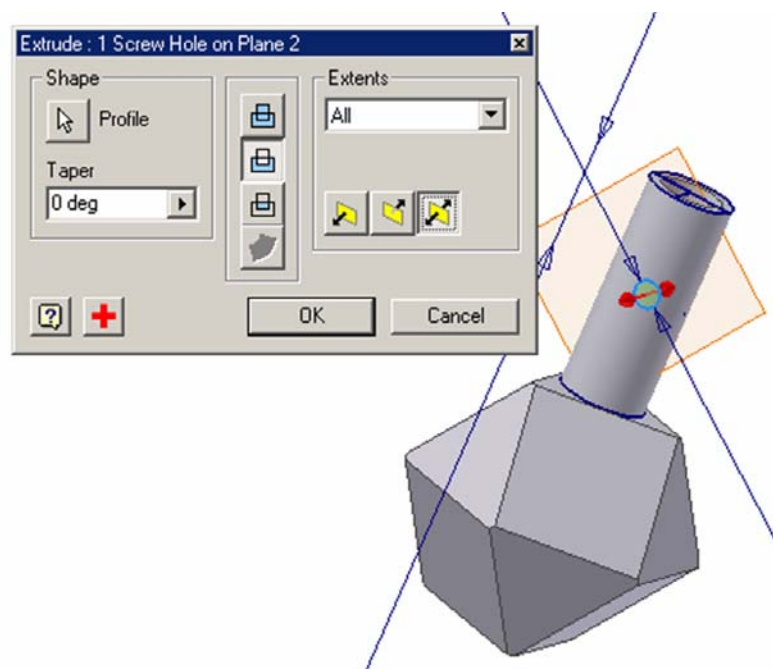


Figure 7A-3P: Selecting the Screw Hole Plane Work Plane 2.



*Figure 7A-3Q:
The Screw Hole
Sketch 2.*



*Figure 7A-3R:
Creating one
fastener hole.*

Next, select the *Screw Hole Plane Work Plane 2* (Figure 7A-3P), click the **Look At** button to switch to a normal orthographic view, and click the **Sketch** button to start a

new sketch; rename it *Screw Hole Sketch 2* in the **Model** panel; use the **Project Geometry** tool to project the top edge of the tube holder onto the sketch; use the **Offset** and **General Dimensions** tool to create an offset line 3.0 inches from the projected top edge line (the offset line provides a midpoint snap for the center of the fastener hole); next, use the **Circle Center Point** and **General Dimension** tools to create a 0.75-inch diameter circle, as shown in *Figure 7A-3Q*. Next, use the **Extrude** tool with **Cut** for **Type**, **All** for **Extents**, **Midplane** for **Direction**, and the circle for **Profile**, create a fastener hole (*Figure 7A-3N*); rename the new **Extrude** feature *1 Screw Holes On Plane 2* in the **Model** panel.

Step 3: Adding tube holders with fastener holes to all oblique surfaces

Select the **Circular Pattern** tool; in the **Circular pattern** tool's dialog window, click the **Features** button and click-select the *Tube Holder*, *2 Screw Holes On Plane 1*, and *1 Screw Hole On Plane 2* features from the **Model** panel, holding down the **Shift** key on the keyboard for multiple selections (*Figure 7A-3S*); next, click the **Rotation Axis** button and then click-select the **Y Axis** from the **Model** panel; type 4 ul in the **Count** text field; and type 360.00 deg in the **Angle** text field; click the **OK** button to create additional tube holders with fastener holes, with the **Circular Pattern** feature (*Figure 7A-3T*); and rename it *Tube Holder Pattern* in the **Model** panel.

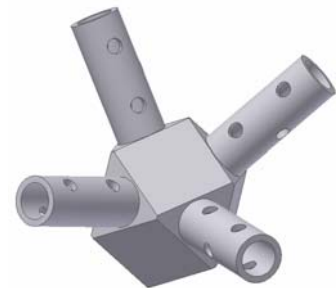
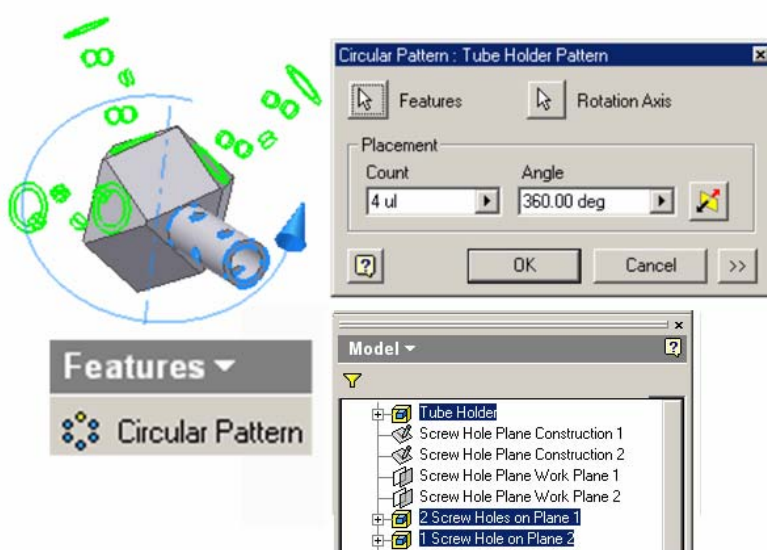


Figure 7A-3T:
Additional tube holders
with fastener holes.

Figure 7A-3S: Using the **Circular Pattern** tool to create additional tube holders with fastener holes.

Next, select the **Mirror Feature** tool to create the remaining tube holders with fastener holes on all other oblique surfaces; in the tool's dialog box, click the **Features** button, and click-select *Tube Holder Pattern* in the **Model** panel (notice that once this feature is click-selected, all dependent features, i.e., the *Tube Holder*, *2 Screw Holes On Plane 1*, and *1 Screw Hole On Plane* features are all automatically selected and highlighted in the **Model** panel); next, click the **Mirror Plane** button in the tool's dialog box, and click-select the **XZ Plane** from the **Model** panel (*Figure 7A-3U*); click the **OK** button to create the **Mirror Pattern** feature (*Figure 7A-3V*), and rename it *Tube Holder Mirror* in the **Model** panel.

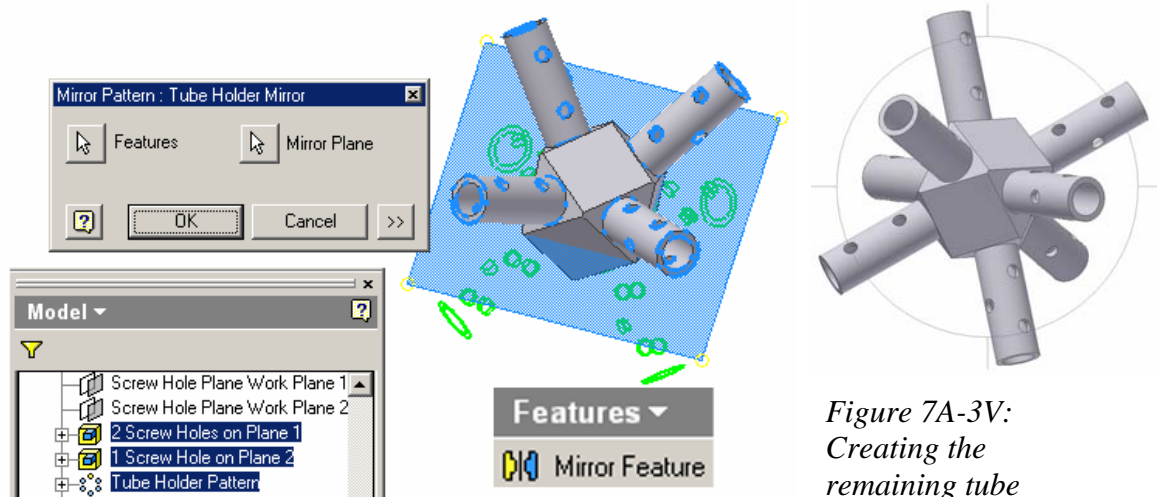


Figure 7A-3U: Using the Mirror Pattern tool.

*Figure 7A-3V:
Creating the
remaining tube
holders with fastener
holes on all other
oblique surfaces.*

Step 4: Creating a tube holder with fastener holes on a rectangular surface parallel to the principal planes

While the 3D model is in **Isometric** view, select the rectangular surface close to the top of the model (any of the rectangular surfaces will work; but for convenience, we select this particular one); click the **Sketch** button from the **Command Bar** to start a new sketch; use the **Project Geometry** tool to project the three perpendicular edge lines onto the new sketch; use the **Circle Center Point** tool to draw a circle centered approximately at the center of the triangle and with a diameter that places the circle within the triangle; select the **Tangent** tool, click-select one of the projected edges and the circle, the circle becomes “tangent” to or touches the edge of the rectangle; select another edge of the rectangle and the circle again, the circle becomes “tangent” to or touches two edges of the rectangle; select the third edge of the rectangle and the circle again, the circle not only becomes “tangent” to or touches all three edges of the rectangle, its size enlarges as well;

this circle provides a center point snap for additional concentric circles to be drawn; next, use the Circle **Center Point** and **General Dimensions** tools to draw two concentric circles of 3.75 inches and 3.0 inches diameters respectively (*Figure 7A-4A*); select the first circle and go to the **Command Bar** to change its line **Style** to **Construction** (or select the first circle together with the triangle's edge lines and use the **Delete** key on the keyboard to delete all of them); click the **Return** button to complete the sketch; and rename the **Sketch** feature as *Large Tube Holder Sketch* in the **Model** panel.

Next, select the **Extrude** tool; choose the **Join** as **Type**, type 8 in as **Distance** for **Extents**, the space between the two concentric circles of 3.75- and 3.0-inch diameters as **Profile**, and outward from the oblique surface as **Direction** (*Figure 7A-4B*), create the extruded tube holder and rename the **Extrude** feature in the **Model** panel as *Large Tube Holder* (*Figure 7A-4C*).

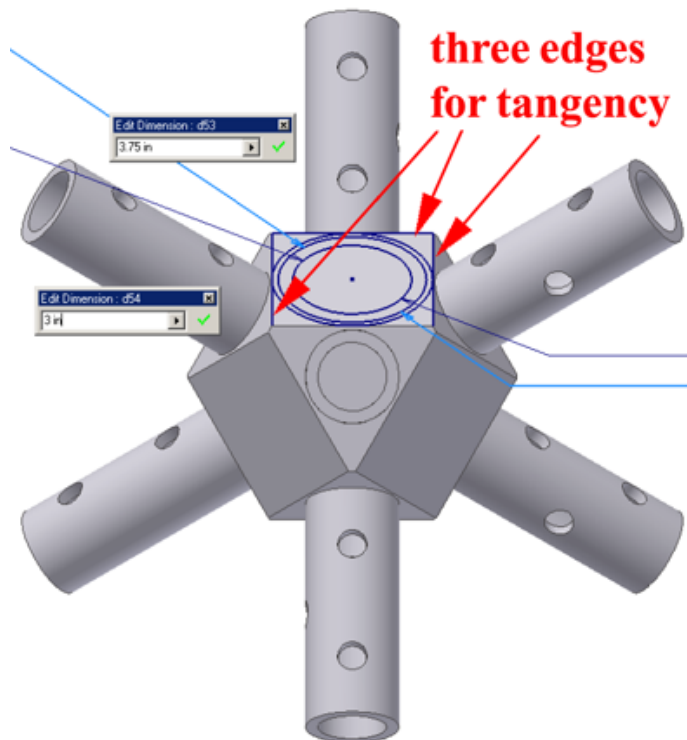


Figure 7A-4A: The Large Tube Holder Sketch.

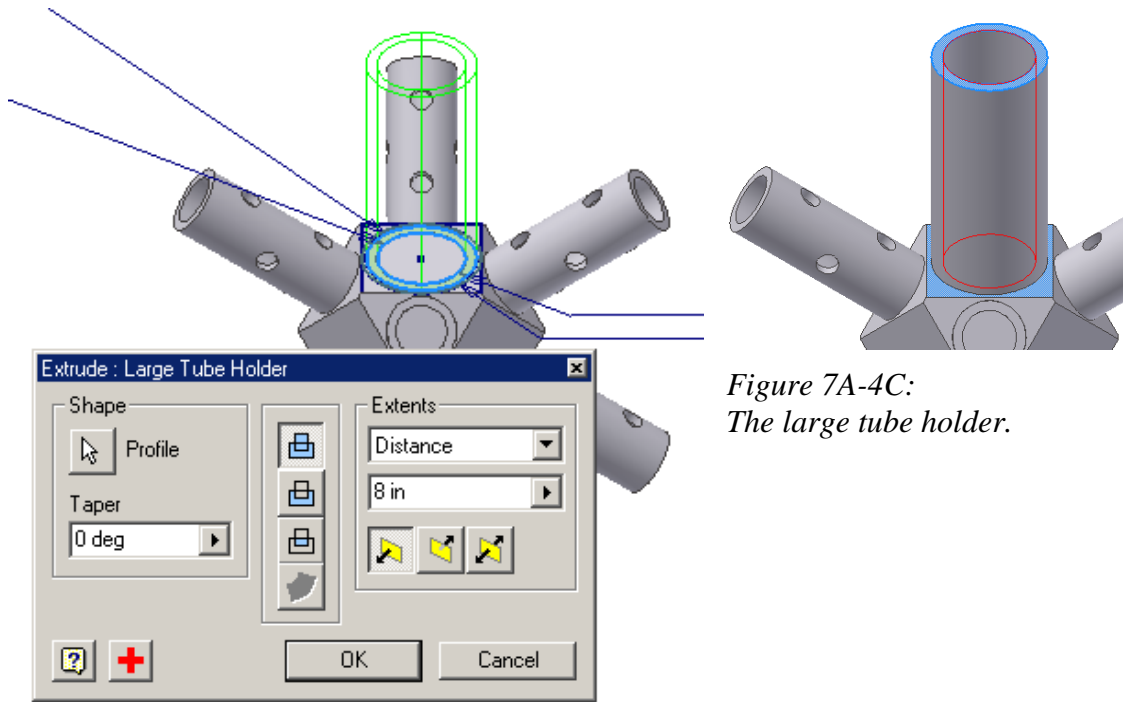


Figure 7A-4C:
The large tube holder.

Figure 7A-4B: Using the Extrude tool.

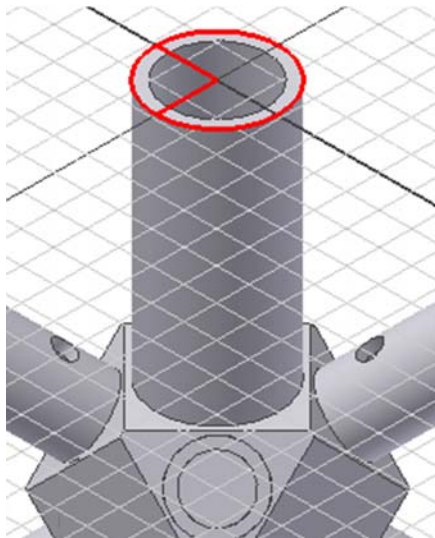


Figure 7A-4D: Creating the
Large Tube Holder Screw Hole
Plane Construction 1 Sketch.

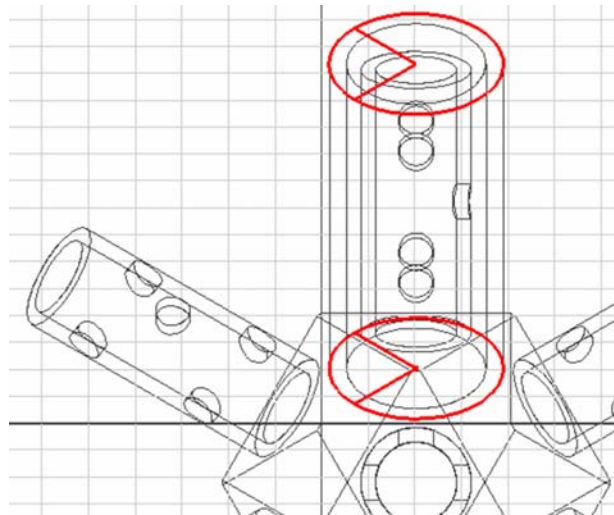


Figure 7A-4E: Creating the Plane
Construction 2 Sketch for the Large Tube
Holder Screw Hole.

Next, create two construction sketches to provide three points for the definition of work planes. Select the top surface of the large tube holder and click the **Sketch** button to start a new sketch; rename the sketch as *Large Tube Holder Screw Hole Plane Construction 1* in the **Model** panel; use the **Project Geometry** tool to project the circular outer edge of the tube holder onto the sketch; use the **Line** tool to draw two perpendicular lines centered at the center of the projected circle and extending beyond the circle's circumference (the two lines should be parallel to grid lines), with the help of Inventor's **perpendicular** and **Parallel** indicators (an alternative method is to use the **Perpendicular** and **Parallel** tools to constrain the lines drawn); use the **Trim** tool to trim off the excess line segments beyond the circle (*Figure 7A-3E*); click the **Return** button to exit the **Sketch** mode. Next, select the rectangular surface at the base of the tube holder and click the **Sketch** button to start a new sketch; rename the sketch as *Large Tube Holder Screw Hole Plane Construction 2* in the **Model** panel; use the **Project Geometry** tool to project the profile of the *Large Tube Holder Screw Hole Plane Construction 1* onto the sketch (*Figure 7A-4E*); click the **Return** button to exit the **Sketch** mode.

Next, create the work planes for the fastener holes' profiles. For better visualization, first go to the **Command Bar** to switch to **Wireframe Display** mode. Select the **Work Plane** tool; move the mouse closer to each of the three points from both *Large Tube Holder Screw Hole Construction 1* and *Large Tube Holder Screw Hole Construction 2* sketches, as shown in *Figure 7A-4F*, and click once when the red snap point indicator appears; a new **Work Plane** feature is created; rename it *Large Tube Holder Screw Hole Plane 1* in the **Model** panel. Use the **Work Plane** tool again and move the mouse closer to each of the three points from both sketches, as shown in *Figure 7A-4G*, and click once when the red snap point indicator appears; a new **Work Plane** feature is created; rename it *Large Tube Holder Screw Hole Plane 2* in the **Model** panel.

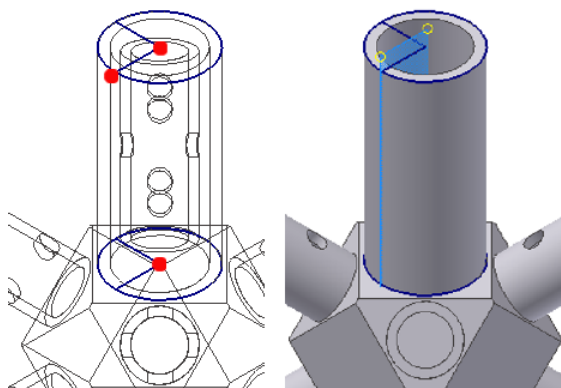


Figure 7A-4F: The Large Tube Holder Screw Hole Plane 1.

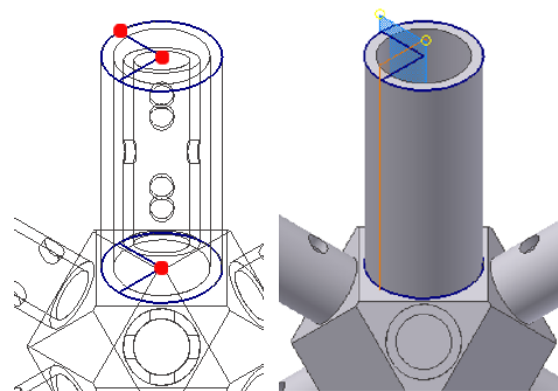


Figure 7A-4G: The Large Tube Holder Screw Hole Plane 2.

Next, select the *Large Tube Holder Screw Hole Plane 1*, click the **Look At** button to switch to a normal orthographic view, and click the **Sketch** button to start a new sketch; rename it *Large tube Screw Hole Sketch 1* in the **Model** panel; use the **Project Geometry** tool to project the top edge of the tube holder onto the sketch; use the **Offset** and **General Dimensions** tool to create an offset line 0.75 inches from the projected top edge line, and another two offset lines at a distance of 3.0 inches each from the previous offset line; next, use the **Circle Center Point** and **General Dimension** tools to create three 0.75-inch diameter circle with the centers snapped to the midpoint of the offset lines, as shown in *Figure 7A-4H*. Next, use the **Extrude** tool with **Cut** for **Type**, **All** for **Extents**, **Midplane** for **Direction**, and the circle for **Profile**, create three fastener holes (*Figure 7A-4J*); rename the new **Extrude** feature *3 Large Tube Screw Holes On Plane 1* in the **Model** panel.

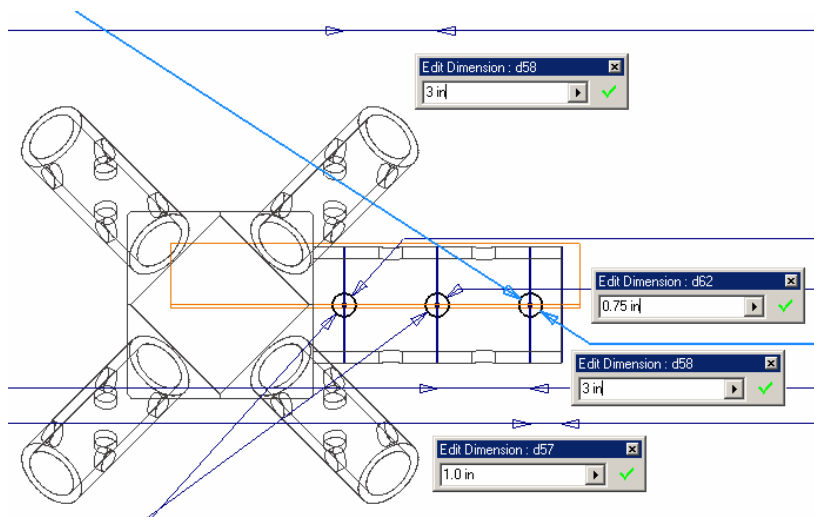


Figure 7A-4H: The Large tube Screw Hole Sketch 1 created on the Large Tube Holder Screw Hole Plane 2.

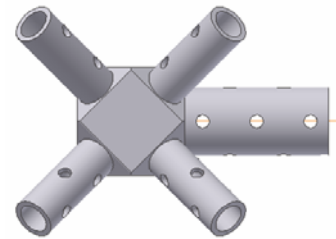


Figure 7A-4J: Three screw holes on the large tube.

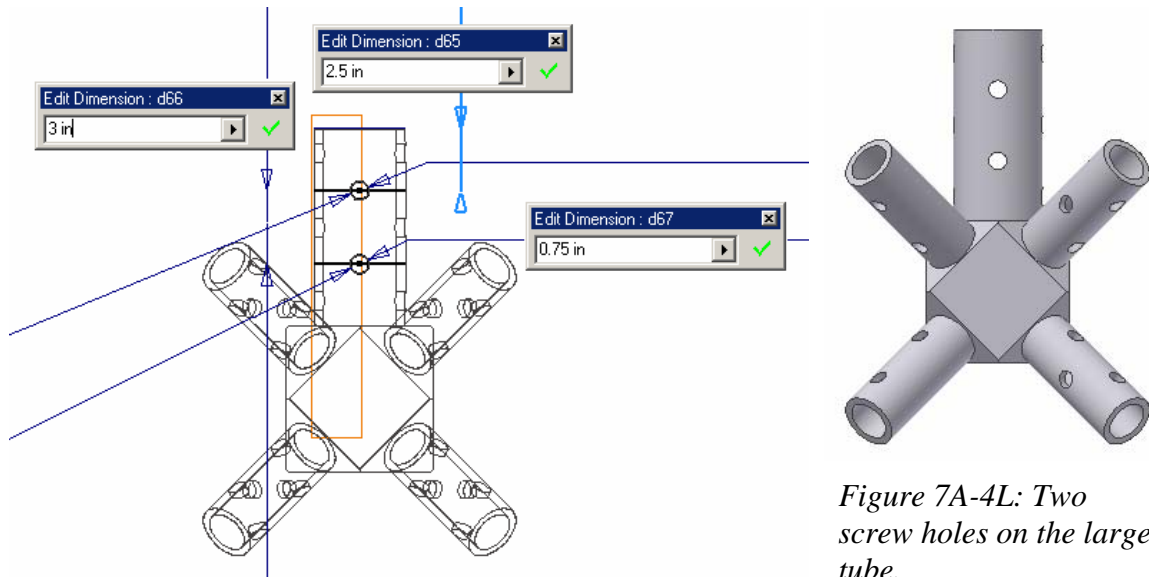


Figure 7A-4K: The Large Tube Screw Hole Sketch 2 created on the Large Tube Holder Screw Hole Plane 2.

Figure 7A-4L: Two screw holes on the large tube.

Next, select the *Large Tube Holder Screw Hole Plane 2*, click the **Look At** button to switch to a normal orthographic view, and click the **Sketch** button to start a new sketch; rename it *Large Tube Screw Hole Sketch 2* in the **Model** panel; use the **Project Geometry** tool to project the top edge of the tube holder onto the sketch; use the **Offset** and **General Dimensions** tool to create an offset line 2.5 inches from the projected top edge line, and another offset line at a distance of 3.0 inches from the first offset line; next, use the **Circle Center Point** and **General Dimension** tools to create two 0.75-inch diameter circle, with the centers snapped to the midpoint of the offset lines, as shown in *Figure 7A-4K*. Next, use the **Extrude** tool with **Cut** for **Type**, **All** for **Extents**, **Midplane** for **Direction**, and the circle for **Profile**, create a fastener hole (*Figure 7A-4L*); rename the new **Extrude** feature *2 Large Tube Screw Holes On Plane 2* in the **Model** panel.

Step 5: Creating additional tube holders with fastener holes to all rectangular surfaces parallel to the principal planes with the Circular Pattern tool

Select the **Circular Pattern** tool; in the **Circular pattern** tool's dialog window, click the **Features** button and click-select the *Large Tube Holder, 3 Large Screw Holes On Plane 1*, and *2 Large Screw Hole On Plane 2* features from the **Model** panel, holding down the **Shift** key on the keyboard for multiple selections (*Figure 7A-4M*); next, click the **Rotation Axis** button and then click-select the **X Axis** from the **Model** panel; type 4 in the **Count** text field; and type 360.00 deg in the **Angle** text field; click the **OK**

button to create additional tube holders with fastener holes, with the **Circular Pattern** feature (*Figure 7A-4N*); and rename it *Large Tube Holder Pattern 1* in the **Model** panel. Three large tube holders with fastener holes are added to three more rectangular surfaces, which are parallel to the principal planes of Inventor. We still need two tube holders on the remaining two rectangular surfaces. Use the **Rotate** tool to figure this situation out.

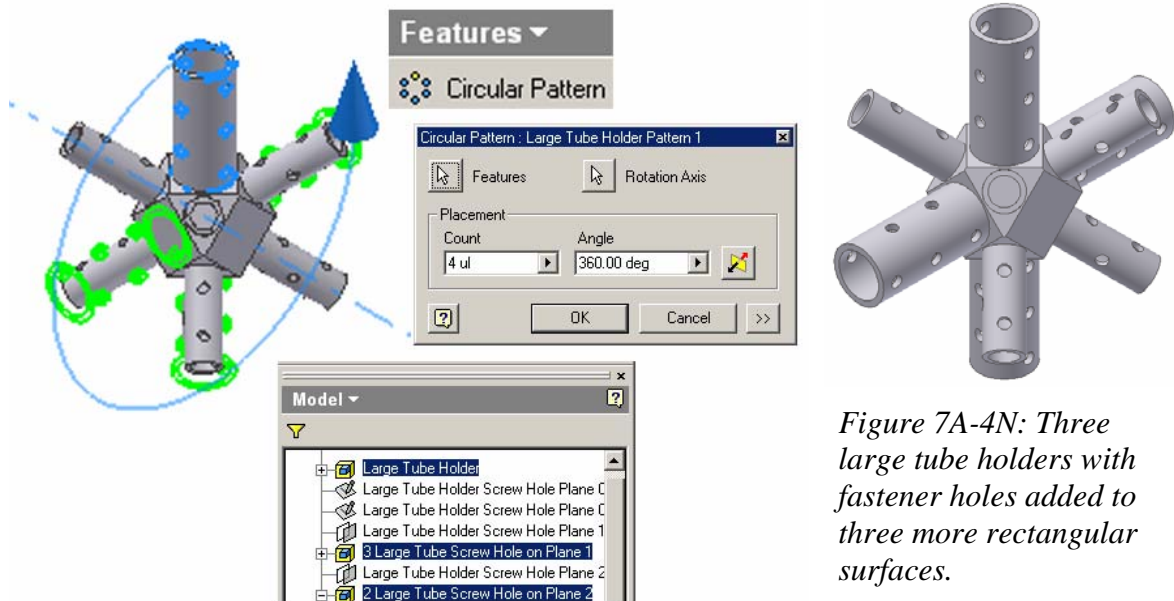


Figure 7A-4M: Using the Circular Pattern tool.

Use the **Circular Pattern** tool again; in the **Circular pattern** tool's dialog window, click the **Features** button and click-select the same *Large Tube Holder*, *3 Large Screw Holes On Plane 1*, and *2 Large Screw Hole On Plane 2* features again from the **Model** panel, holding down the **Shift** key on the keyboard for multiple selections (*Figure 7A-4P*); next, click the **Rotation Axis** button and then click-select the **Z Axis** from the **Model** panel; type 4 ul in the **Count** text field; and type 360.00 deg in the **Angle** text field; click the **OK** button to create additional large tube holders with fastener holes, with the **Circular Pattern** feature (*Figure 7A-4Q*); and rename it *Large Tube Holder Pattern 2* in the **Model** panel. All rectangular surfaces have large tube holders with fastener holes now. Actually, after the second **Circular Pattern** feature is created, the large tube holders with fastener holes on the bottom (in the **Isometric** view) is created twice; and this can be seen when we click the boxed + sign in front of the *Large Tube Holder Pattern 1* and *Large Tube Holder Pattern 2* features to open them in the **Model** panel, and move the mouse over the 3rd **Occurrence** of each **Circular Pattern** feature; the **Occurrence** turns red on the 3D model, as shown in *Figure 7A-4R*. If desired, we can click-select the 3rd **Occurrence** of *Large Tube Holder Pattern 2 Circular Pattern*

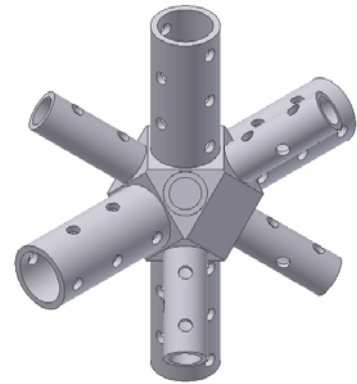


Figure 7A-4N: Three large tube holders with fastener holes added to three more rectangular surfaces.

feature, then right-click for the shortcut menu and choose the **Suppress** option; after an **Occurrence** is suppressed, a horizontal strikethrough is added to the name and the **Occurrence**'s listing is grayed out in the **Model** panel.

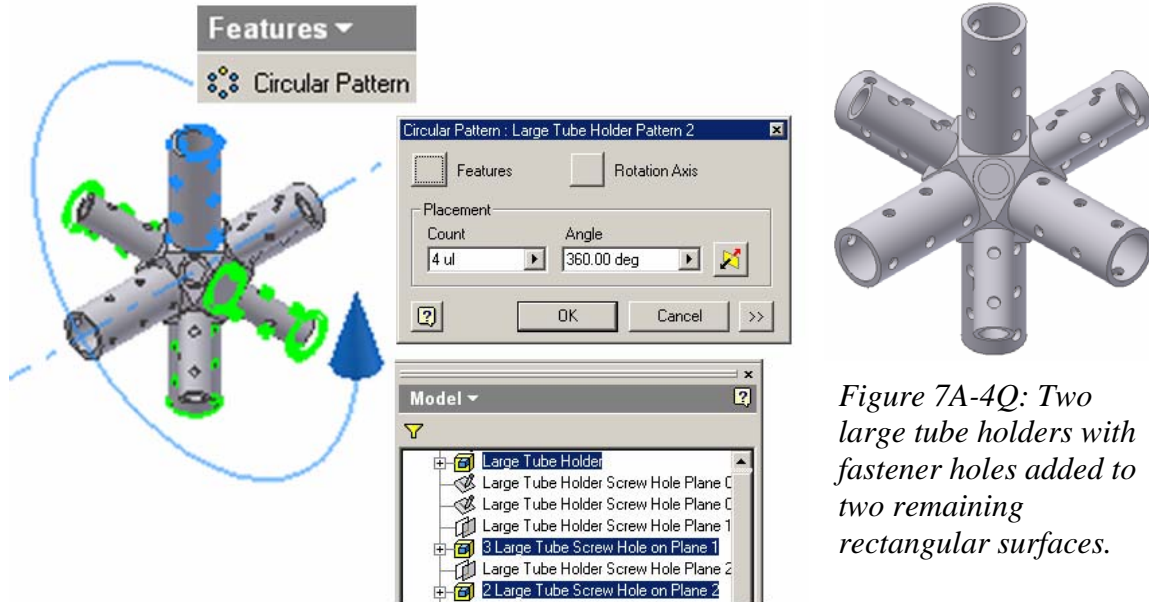


Figure 7A-4Q: Two large tube holders with fastener holes added to two remaining rectangular surfaces.

Figure 7A-4P: Using the Circular Pattern tool again.

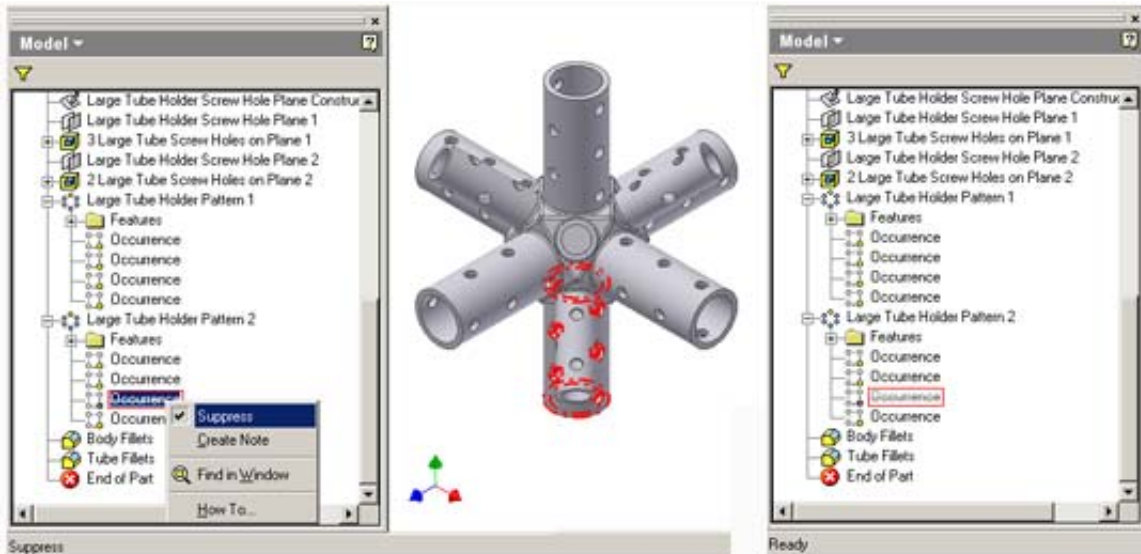


Figure 7A-4R: Suppressing an Occurrence (left); the grayed out listing in the Model panel.

Step 6: Adding fillets to the edges of the original polyhedron body and to the intersection between the tube holders and the surfaces of the polyhedron body

Select the **Fillet** tool; in the tool's dialog window, click the default text field under **Radius** heading, the "pencil" symbol appears in front of "0 Selected" text field; type 0.25 in (inch) over the default text (*Figure 7A-5A*); click the white space beneath the text fields; the "pencil" symbol changes to an arrow symbol; click-select the **Loop** option in the **Select mode** section; next, click-select all edges of the original polyhedron body, with the help of the **Rotate** tool; notice that the number in the "...Selected" text field increases when additional edges are added to the selection (the total number is 36), as shown in *Figure 7A-5B*; click the **OK** button to create the fillets, and rename the **Fillet** feature *Body Fillets* in the **Model** panel (*Figure 7A-5C*).

Use the **Fillet** tool again; in the tool's dialog window, type 0.125 in (inch) over the default text (*Figure 7A-5A*); select the **Loop** option in the **Select mode** section; click-select all edges of intersection between the tube holders and the rectangular and triangular surfaces of the original polyhedron body, with the help of the **Rotate** tool (the total number is 62), as shown in *Figure 7A-5D*; click the **OK** button to create the fillets, and rename the **Fillet** feature *Body Fillets* in the **Model** panel.

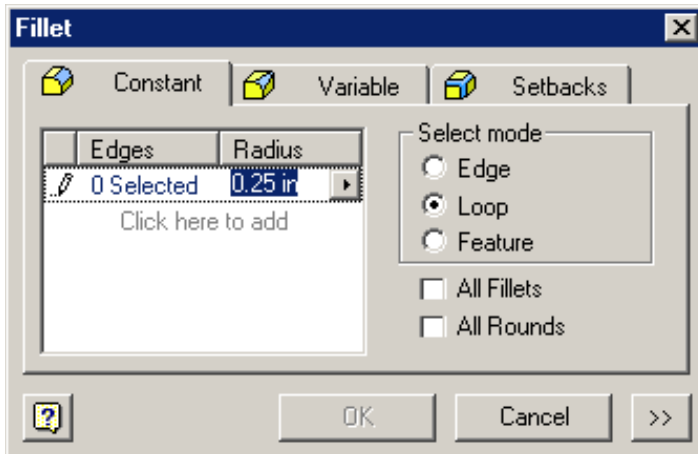


Figure 7A-5A: Typing the Radius value and choosing the Loop option.

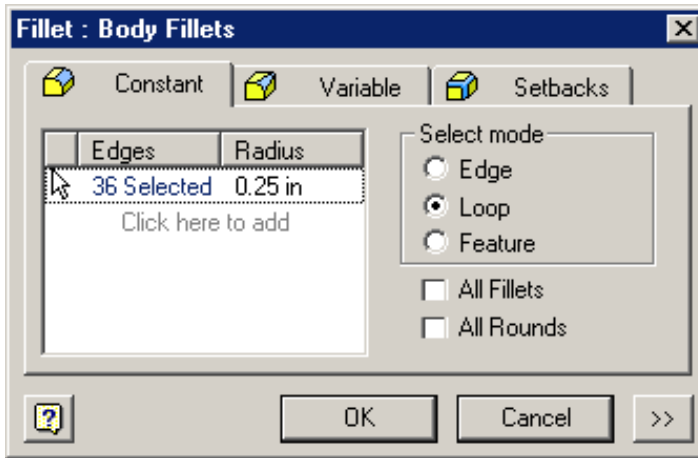


Figure 7A-5B: Selecting the edges.

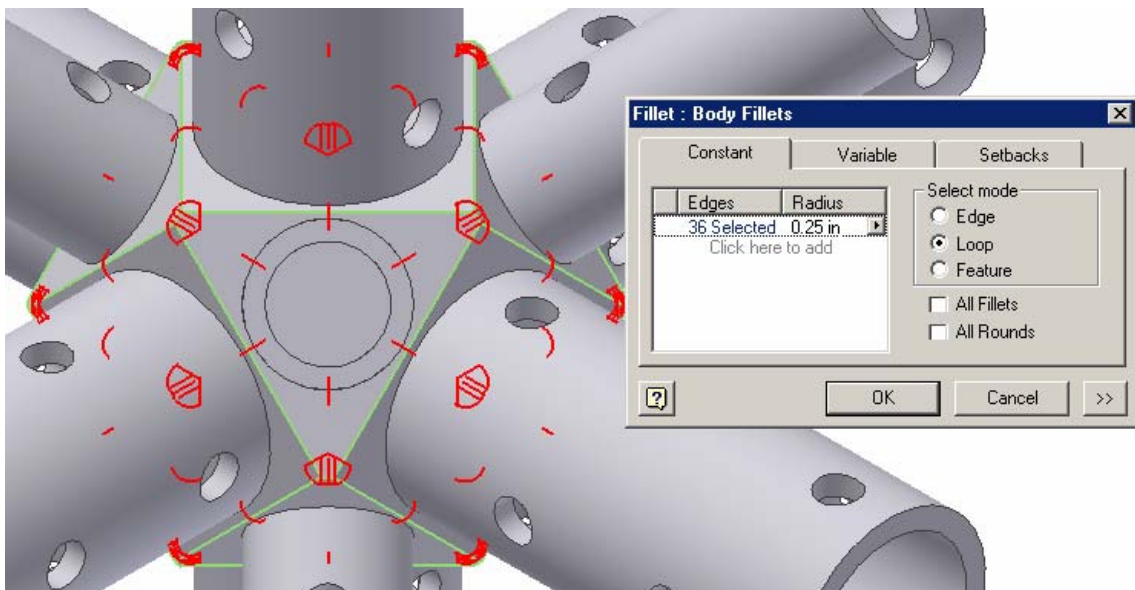


Figure 7A-5C: Creating the Body Fillets.

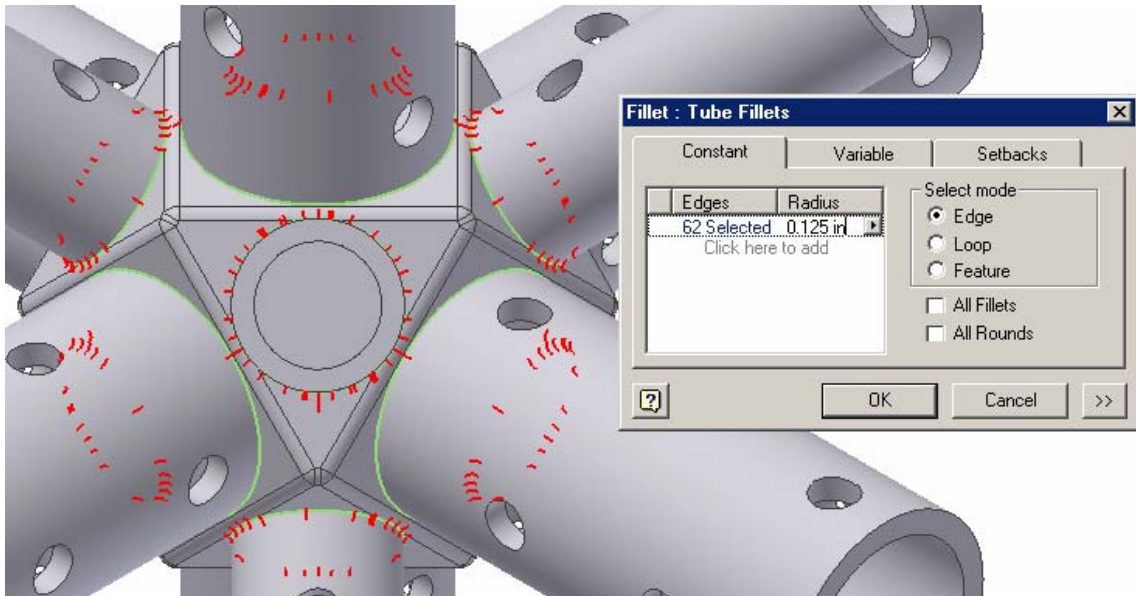


Figure 7A-5D: Creating the Tube Fillets.

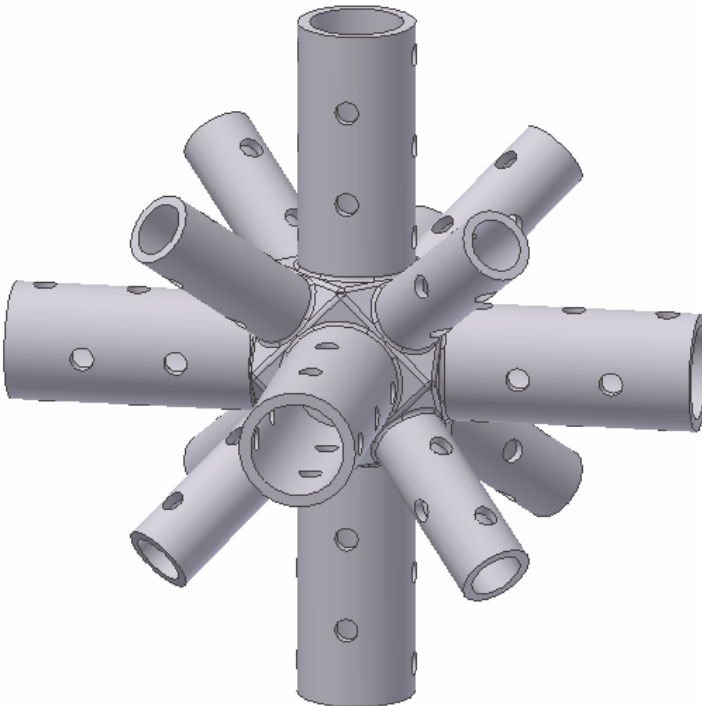


Figure 7A-5E: The completed Multi-Angle Connector part.

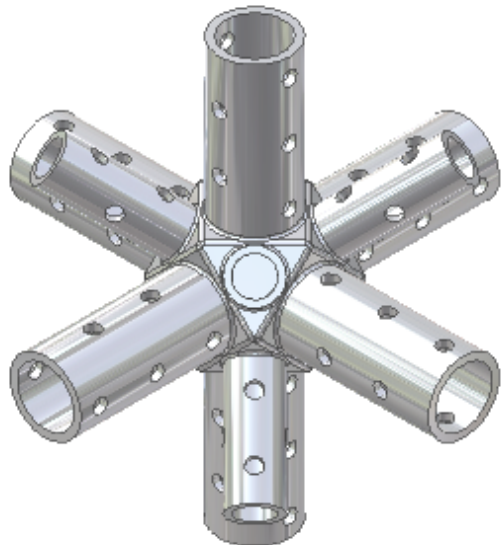
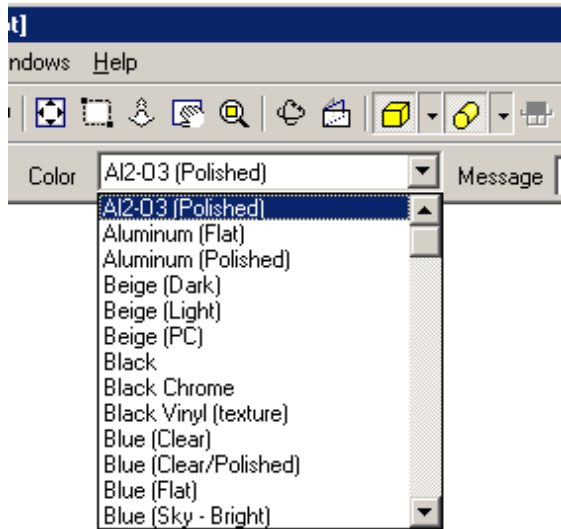


Figure 7A-5F: Changing material appearance of the part.

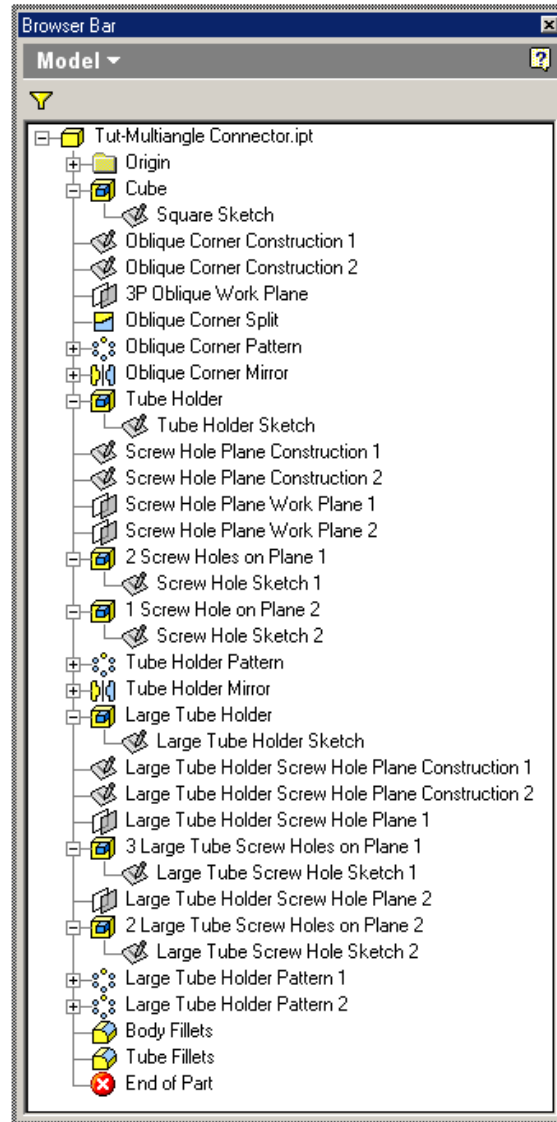


Figure 7A-5G: All features of the part listed in the Model panel.

The Multi-Angle Connector part is completed (Figure 7A-5E). If desired, go the **Color** pull-down menu in the **Command Bar** to change material appearance of the part (Figure 7A-5F). All features of the part are listed in the **Model** panel (Figure 7A-5G).

Congratulations! In this Module, you have learned how to use

- The **Work Plane** tool to create an oblique work plane;

- The **Split** tool to cut a 3D model with an oblique surface;
- The **Circular Pattern** and **Mirror Feature** tools to duplicate existing features on a 3D model, such as a cut oblique corner in this *Module*.

This completes our exploration of the **Work Plane** tool in Inventor.