Module 10A:
Creating a Compound Elbow in Inventor

In this *Module*, we will learn how to create a compound elbow in Autodesk Inventor. A compound elbow is a curved sheet metal transition part used to connect two rectangular or square pipes that are perpendicular to each other and are located at a certain diagonal distance, as shown in *Figure 10A-1A*. The basic procedures of creating a compound elbow are explained below:

- Creating a 3D solid model of the interior space enclosed by the compound elbow in an Autodesk Inventor **Standard (in).ipt** or **Sheet Metal (in).ipt** part file, to be used as **Work Surface** geometry of a **Derived Part** in other **Sheet Metal (in).ipt** files;

- Creating separate **Sheet Metal (in).ipt** files for the top, base, left, and right pieces of the compound elbow;

- Creating an Inventor **Standard (in).iam** assembly file to assemble all left, right, top and base pieces of the compound elbow.

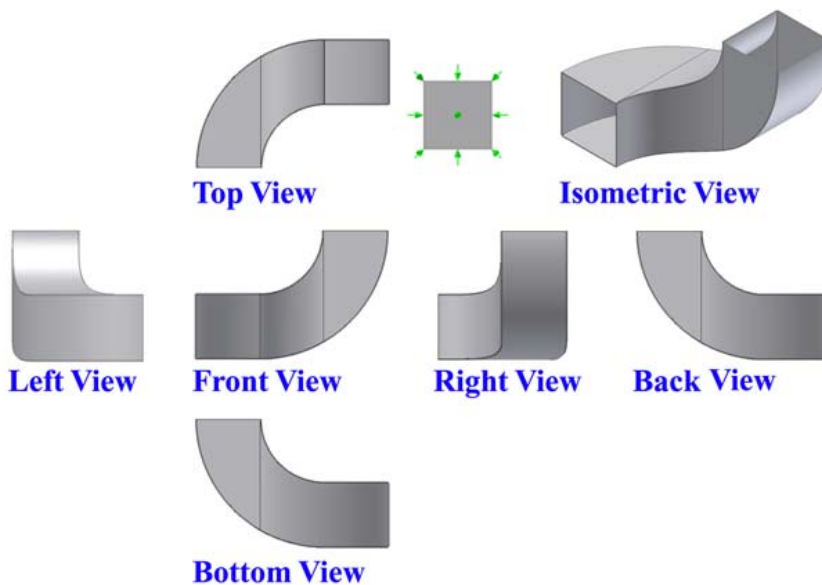These procedures will be illustrated in this *Module*.

Figure 10A-1A: orthographic and isometric views of the assembly of the basic sheet-metal pieces of the compound elbow.

Section 1:
Creating a 3D Solid Model of the Interior Space Enclosed by the
Compound Elbow to Be Used As Work Surface Geometry of a
Derived Part

Step 1: Creating a Extrude prism with a curved base from the XY Plane

Launch Autodesk Inventor; go to the **File→New** to start an Inventor **Sheet Metal (in).ipt** sheet-metal part file; the *Sketch1* feature is automatically created on the **XY Plane** and open on the screen; rename it *XY Plane Curved Body Profile* in the **Model** panel; use the **Project Geometry** tool to project the **Center Point** onto the sketch; use the **Line** tool to create a vertical line and a horizontal line starting at and snapped to the projected **Center Point** upward and rightward respectively, then a horizontal line starting at the top endpoint of the first vertical line and extending rightward, and then another vertical line starting at the right endpoint of the second horizontal line; use the **General Dimension** tool to apply a 8-inch height dimension to the vertical lines and a 8-inch width dimension to the horizontal lines; select the second vertical line and go the **Command Bar** to change its **Style** to **Centerline** (*Figure 10A-1B*); use the **Center Point Circle** tool to create a circle centered at the top endpoint of the second vertical line with the radius pick point at the lower endpoint of the second vertical line, and another concentric circle with the radius pick point at the projected **Center Point** (*Figure 10A-1C*); use the **Trim** tool to trim off the excessive line segments so as to leave a closed profile as shown in *Figure 10A-1D*; click the **Return** button to exit the sketch. Save the file as *Tut-Compound Elbow.ipt* in a new folder to be created and named *Tut-Compound Elbow* in the **Save As** dialog window. Save often upon completion of each feature.
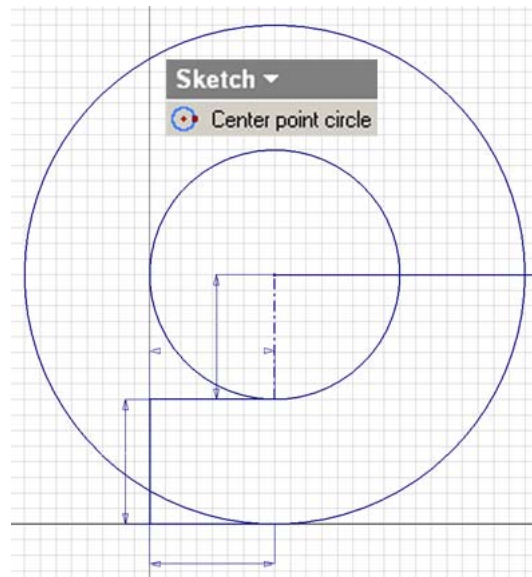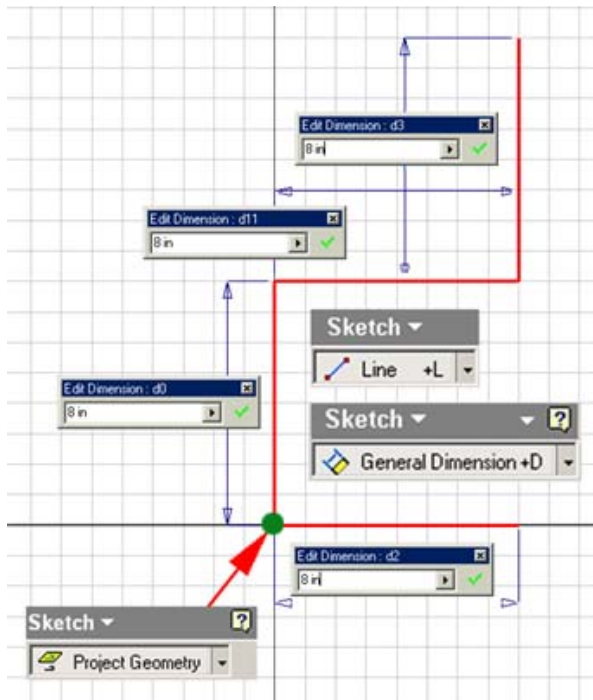
Figure 10A-1C: Creating two
concentric circles with the Center
Point Circle tool.

*Figure 10A-1B: Projecting the Center Point,
creating the horizontal and vertical line, and
applying dimensions.*

        Select the **Extrude** tool; select **Distance** in the **Extents** pull-down menu, type 24
in (inch) in the text field, and select **Midplane** for direction; click the **OK** button to
create the **Extrude** feature (*Figure 10A-1E*); and rename it *XY Plane Curved Body* in the
Model panel.
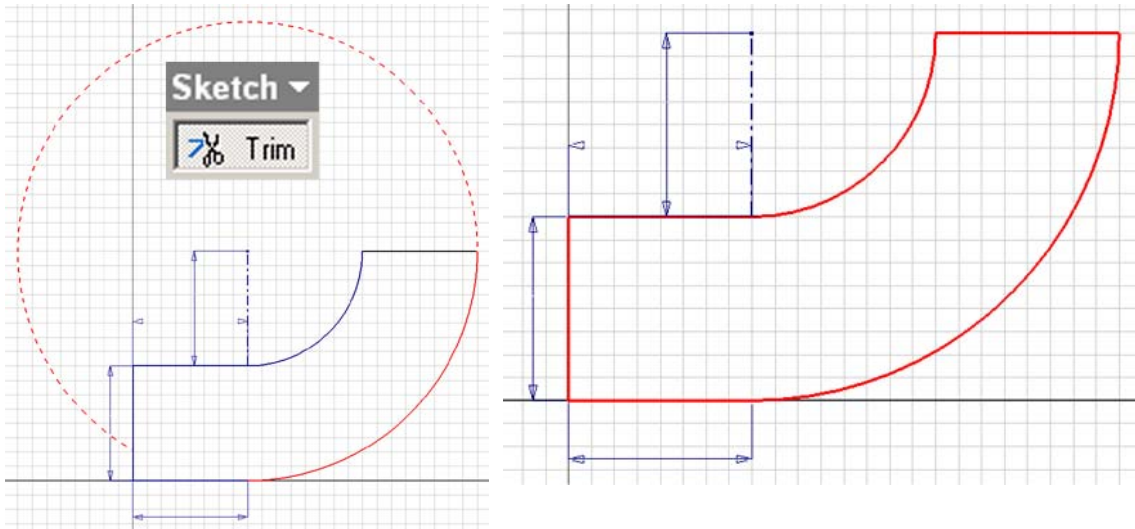
Figure 10A-1D: Trimming of he excessive line segments (left) to leave a closed profile (right).
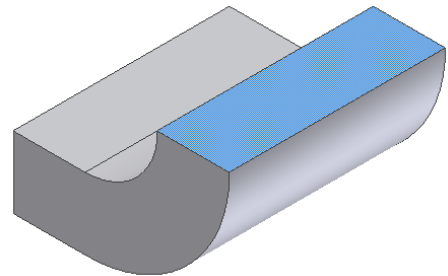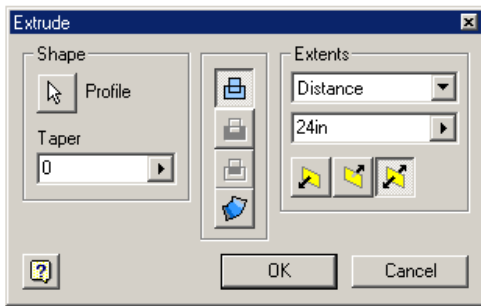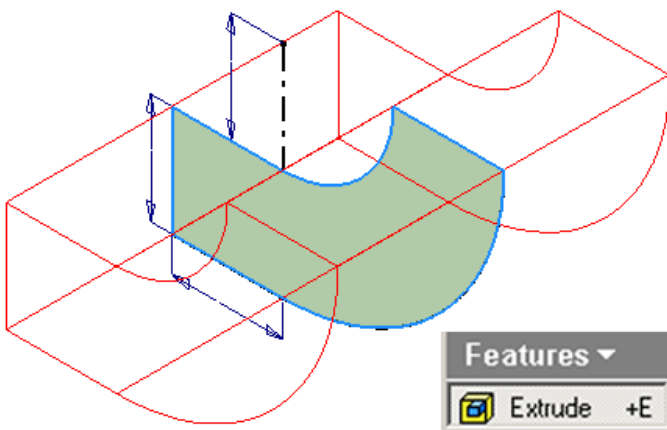

Figure 10A-1E: creating the XY Plane Curved Body Extrude feature.

Step 2: Cutting the Extrude prism with a curved profile from the XZ Plane

Next, select the **XZ Plane** and click the **Sketch** button on the **Command Bar** to start a new sketch; rename it *XZ Plane Curved Cut Profile* in the **Model** panel; use the **Project Geometry** tool to project the right top horizontal edge line and the right vertical edge line onto the sketch (*Figure 10A-1F*); click the **Look At** tool button on the **Command Bar** and then the *XZ Plane Curved Cut Profile* feature in the **Model** panel to switch to an orthographic normal view; select the projected edge lines and go the **Command Bar** tool change their line **Style** to **Normal** (*Figure 10A-1G*); select the **Fix** tool to fix the top right horizontal edge line; then use the **General Dimension** tool to apply a 8 in (inch) height dimension to the right edge line (*Figure 10A-1H*); next, use the **Line** tool to create a horizontal line starting at the lower endpoint of the right edge line and extending leftward, and a vertical line starting at the left endpoint of the top right edge line and extending downward (the two newly created lines should be long enough of cross each other); then use the **General Dimension** tool to apply a 16 in (inch) height dimension to the new vertical line (*Figure 10A-1K*); select the **Trim** tool to trim off the unneeded segments of the new horizontal and vertical lines (*Figure 10A-1L*); select the new vertical line and go to the **Command Bar** to change its **Style** to **Centerline** (the "concentric circle's radius line"); next, select the **Center Point Circle**, click on the lower endpoint of the "concentric circle's radius line" to establish the center and then the upper endpoint to establish the radius; create another concentric circle with the circumference point clicked at the left endpoint of the horizontal top right edge line; next, use the **Line** tool to create a horizontal line starting at the lower endpoint of the vertical "concentric circle's radius line" and extending leftward past the larger concentric circle (*Figure 10A-1M*); next, use the **Center Point Circle** tool again to create another circle center at any convenient point inside the 3D model and large enough to enclose the whole model; next, use the **Trim** tool to trim off all unneeded line segments, so as to completed the closed profile (*Figure 10A-1N*); click the **Return** button to exit the sketch. Save the file.
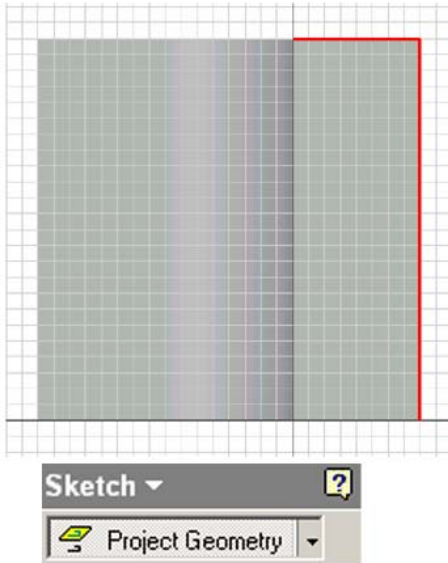
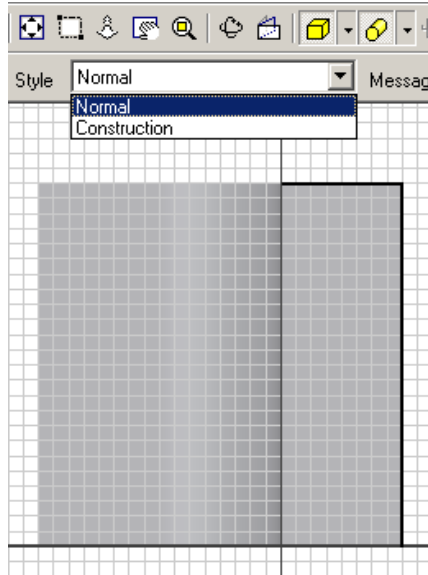*Figure 10A-1F: Projecting the edge lines onto the sketch.*

Figure 10A-1G: Changing the Style of the projected edge lines.
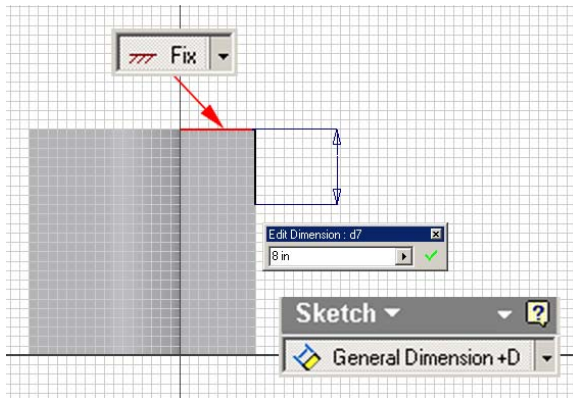

Figure 10A-1H: Fixing the top right edge line and applying a height dimension to the right vertical edge line.
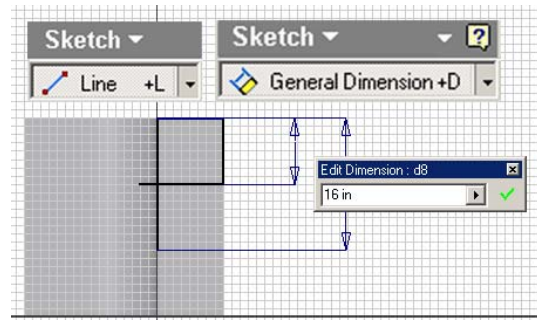

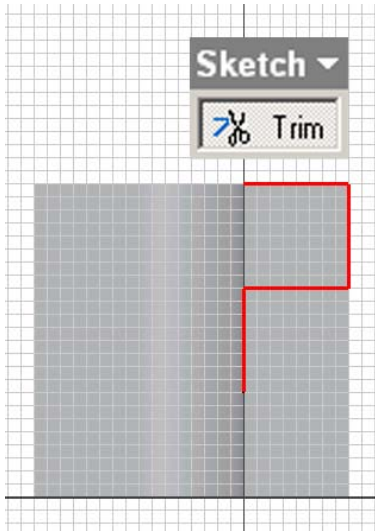*Figure 10A-1K: Creating additional horizontal and vertical lines.*
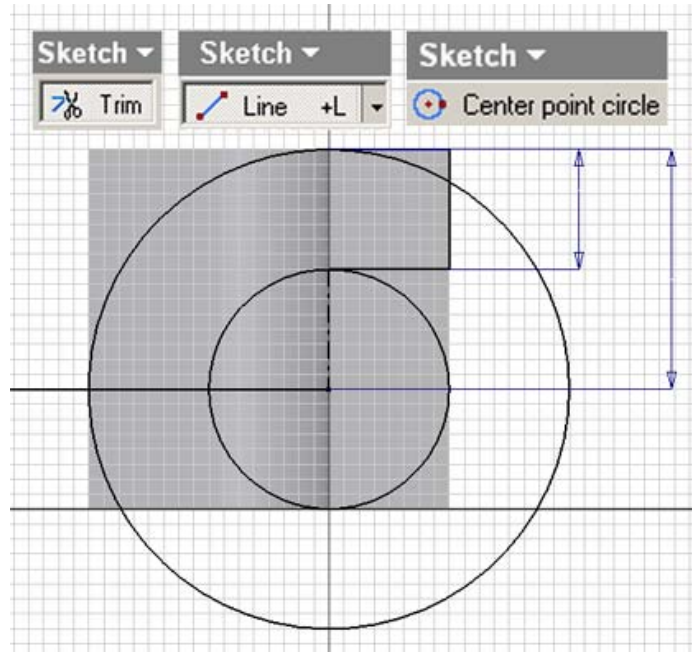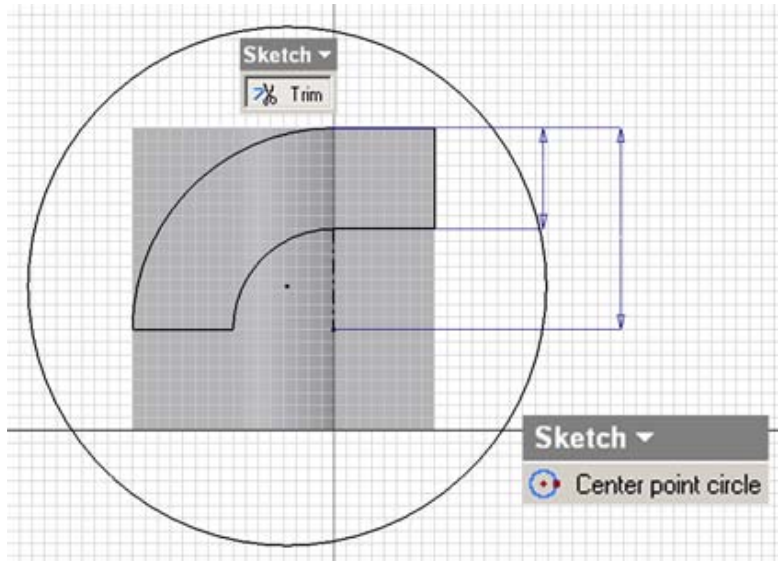
Figure 10A-1L: Trimming off unneeded line segments.

Figure 10A-1M: Creating two concentric circles.

Next, go to the **View→Isometric** menu to switch to an isometric view for better visualization (*Figure 10A-1P*); select the **Extrude** tool; in the tool's dialog window, select **All** for **Extents**, and a downward direction; click the **OK** button to create the **Extrude** feature (*Figure 10A-1Q*), and rename it *XZ Plane Curved Cut* in the **model** panel. The 3D model of the interior space enclosed by the compound elbow is completed. Use the **Rotate** tool with the **Common View [SPACE]** option to view the model from different perspectives (*Figure 10A-1R*). All features are listed in the **Model panel** (*Figure 10A-1S*). Save and close the file.

Figure 10A-1N: Creating a circle center inside the 3D model and large enough to enclose the whole model, and trimming off all unneeded line segments.
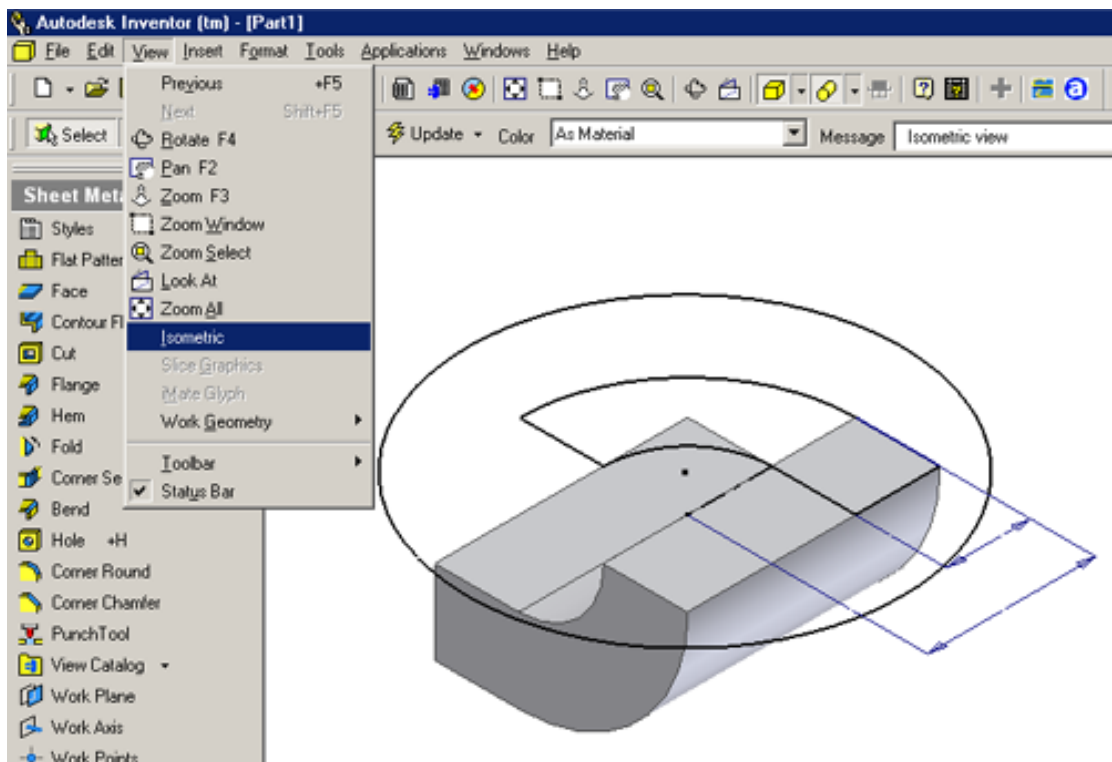

Figure 10A-1P: Switching to an isometric view for better visualization.
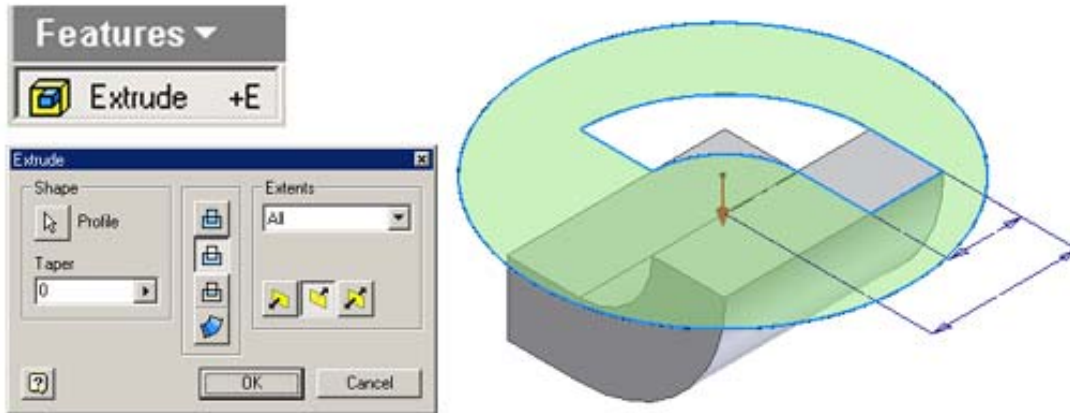
Figure 10A-1Q: Creating the XZ Plane Curved Cut to complete the 3D model of the interior space enclosed by the compound elbow.
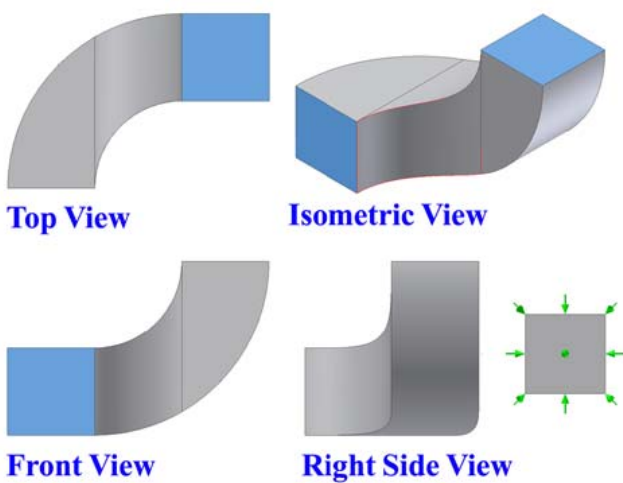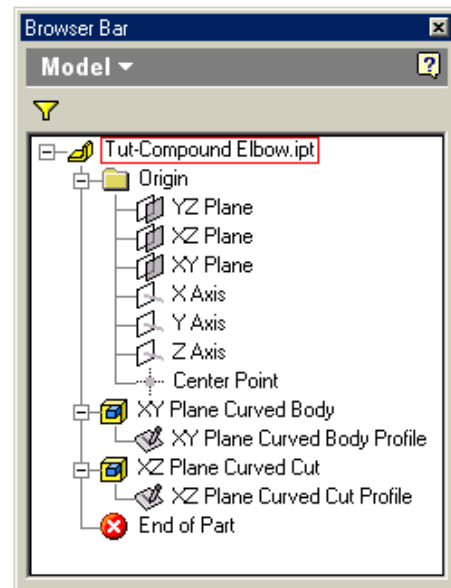

Figure 10A-1R: Using the Rotate tool with the Common View [SPACE] option to view the model from different perspectives.

Figure 10A-1S: All features are listed in the Model panel.

Section 2:
Creating Separate Sheet Metal (In).ipt Files for the Top, Base, Left and Right Pieces of the Compound Elbow

In this Section of the *Module*, we will create four separate **Sheet Metal (in).ipt** files for the top, base left, and right pieces of the compound elbow.

Step 1: Creating a Sheet Metal (in).ipt files for the top piece of the compound elbow

Go to the **File→New** to start an Inventor **Sheet Metal (in).ipt** sheet-metal part file; the *Sketch1* feature is automatically created on the **XY Plane** and open on the screen; click the **Return** button on the **Command Bar** to exit the sketch; click-select the *Sketch1* feature in the **Model** panel, right-click for the shortcut menu and choose **Delete** to delete it (*Figure 10A-2A*). Next, select the **Derived Component** tool; in the **Open** dialog window, select the *Tut-Compound Elbow* file, click the **Open** button (*Figure 10A-2B*); in the **Derived Part** dialog window, select the **Body as Work Surface** option, and click the **OK** button. The light orange **Work Surface** geometry of the 3D model from the *Tut-Compound Elbow.ipt file* appears on the screen (*Figure 10A-2C*). Save the file as *Tut-Top SM Compound Elbow.ipt* in the same *Tut- Compound Elbow* folder. In addition, go to the **File→Save A Copy As** menu to save three copies of the file as *Tut-Base SM Compound Elbow.ipt*, *Tut-Left SM Compound Elbow.ipt*, and *Tut-Right SM Compound Elbow.ipt* in the same folder. Save often upon completion of each feature (*Figure 10A-2D*).
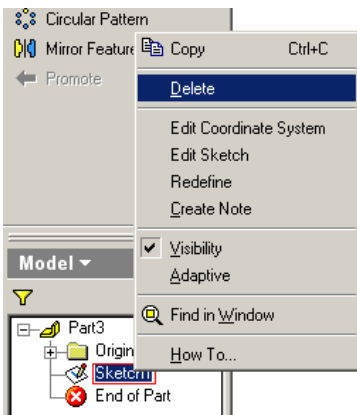

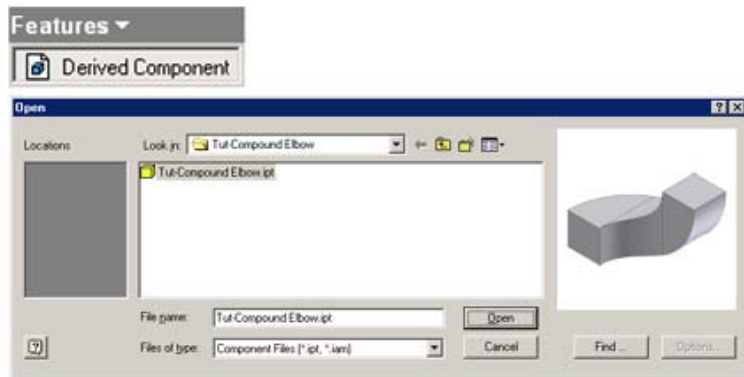*Figure 10A-2A: Deleting the default Sketch1 feature.*

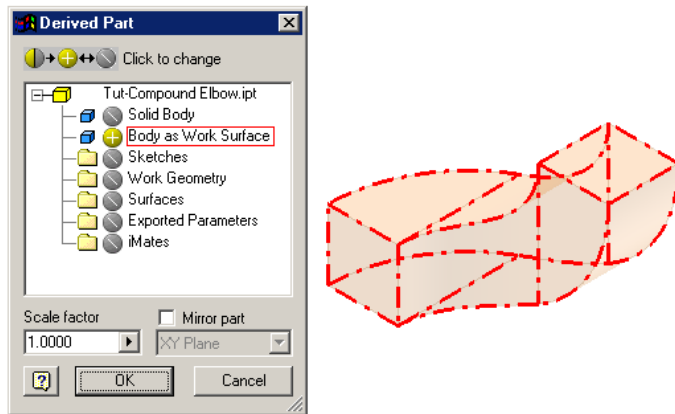*Figure 10A-2B: Selecting the Tut-Compound Elbow.ipt file.*

Figure 10A-2C: Placing the Work Surface of the Derived component.
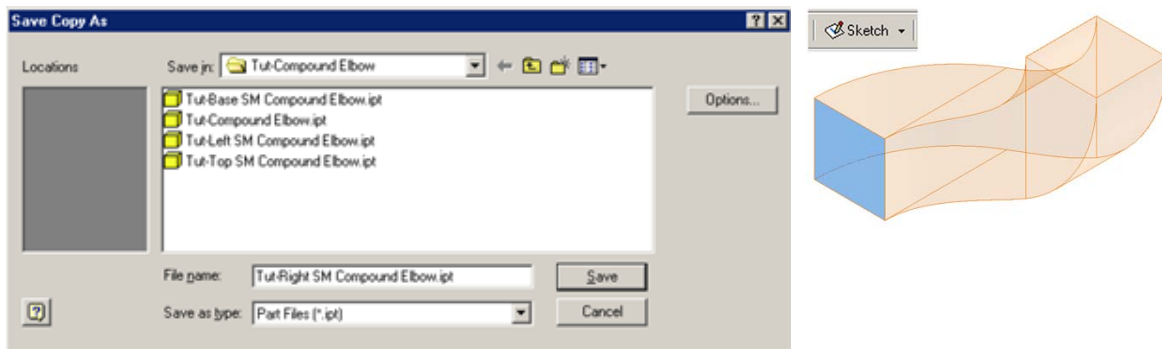

Figure 10A-2D: Save separate copies of the file under different names in the Save Copy As window (left). Selecting the upright square surface at the base of the model as shown to start a new sketch (right).

Next, select the upright square surface at the base of the model as shown in *Figure 10A-2D* and click on the **Sketch** tool button on the **Command Bar** to start a new sketch; rename it as *Top Piece Profile* in the **Model** panel; click the **Look At** button from the **Command Bar** and click-select the *Top Piece Profile* from the **Model** panel to switch to an orthographic normal view; select the **Project Geometry** tool to project the two top horizontal edge line (*Figure 10A-2E*); use the **Line** tool and the **General Dimension** tool to create a vertical line starting at the right endpoint of the projected horizontal edge line on the left-hand side and extending upward, and with a height of 8 inches (the "circle radius line," as shown in *Figure 10A-2F*); select the "circle radius line" and go to the **Command Bar** to change its **Style** to Centerline; next, use the **Center Point Circle** tool to create a circle centered at the top endpoint of the "circle radius line" and with the circumference point set at its lower endpoint (*Figure 10A-2G*); next, select the **Line** tool again to create additional horizontal and vertical lines radiating from the center of the circle and extending beyond the circumference for trimming the circle

(*Figure 10A-2H*); next, use the **Trim** tool to trim off unneeded line segments, so as to leave a connected open profile consisting of a horizontal line and a quarter arc, as shown in *Figure 10A-2J*; click the **Return** button to exit the sketch. Save the file.
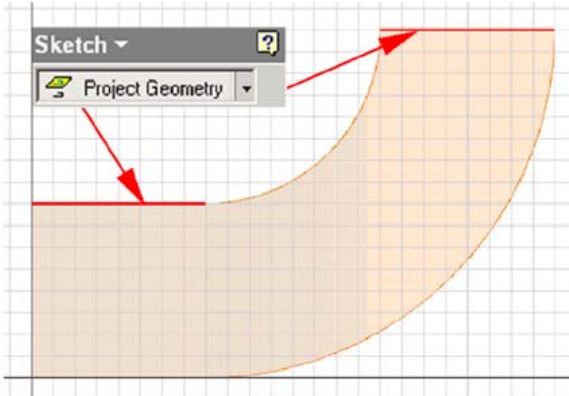

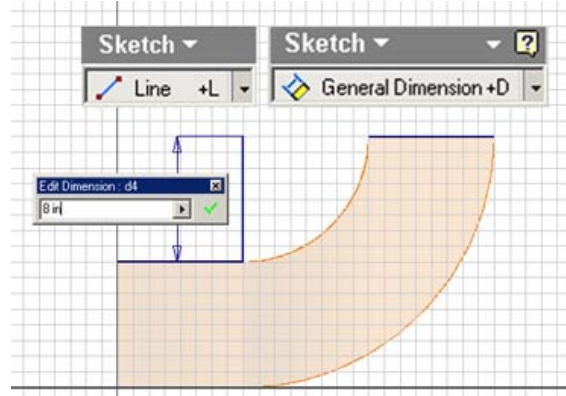*Figure 10A-2E: Projecting the edge lines.*

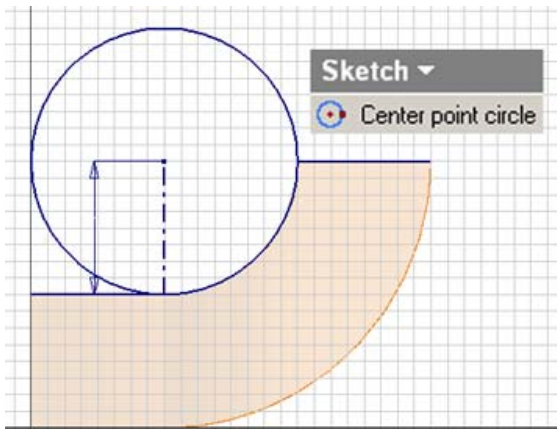*Figure 10A-2F: Creating the "circle radius line" with Line and General Dimension tools.*
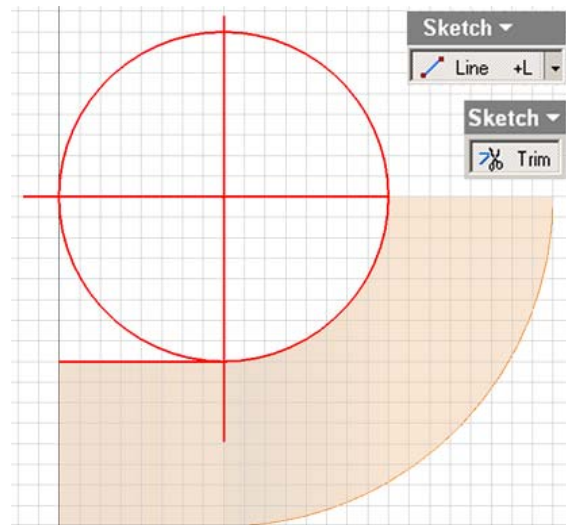

Figure 10A-2G: Creating the circle.

Figure 10A-2H: Creating additional horizontal and vertical lines for trimming the circle.

Next, select the **Contour Flange** tool; in the tool's dialog window, click the **Profile** button and then click-select the open profile; type 24 in (inch) in the **Distance** text field; click the **Flip Direction** button next to the text field to make sure that the

Contour Flange is extruded in the correct direction as shown in *Figure 10A-2K*; use the **Zoom Window** tool to zoom in the left corner area of the green outline geometry; click the **Offset** button in the **Shape** Section to make sure that the **Contour Flange** is extruded outwardly (upward) from the enclosed space of the compound elbow defined by the **Work Surface** geometry (*Figure 10A-2L*); click the **OK** button to create the **Contour Flange** feature (*Figure 10A-2M*); and rename it as *Top Piece* in the **Model** panel.
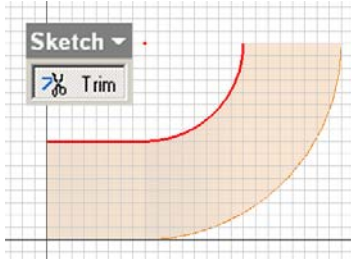
Figure 10A-2J: The open profile consisting of a horizontal line and a quarter arc.
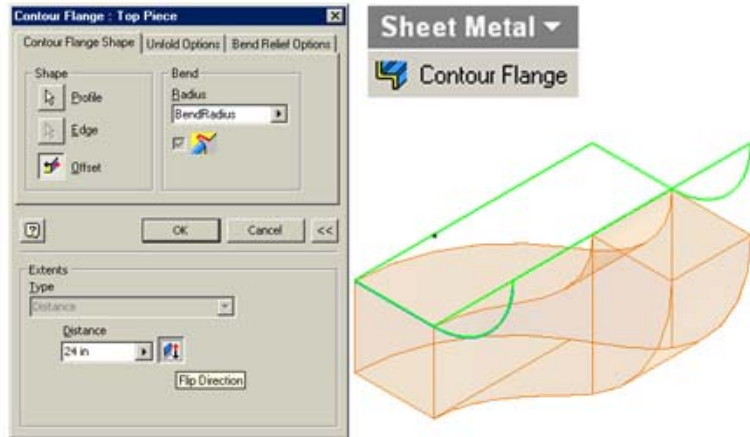
Figure 10A-2K: Using the Contour Flange tool.

Figure 10A-2M: Creating the Top Piece.
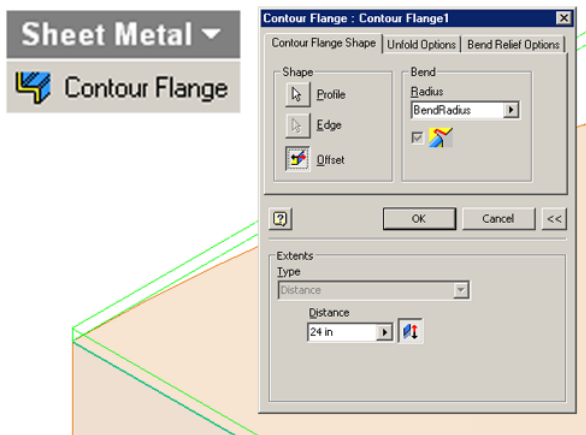
Figure 10A-2L: Clicking the Offset button to make sure that the Contour Flange is extruded outwardly from the enclosed space of the compound elbow.

Next, select the squared top surface of the 3D **Placed Component** model, as shown in *Figure 10A-2N*, and click the **Sketch** button on the **Command Bar** to start a

new sketch; rename it *Compound Curve Cut Profile*; go to the **Command Bar** and select
Hidden Edge Display from the pull-down menu, so that hidden curved edges of the 3D
model of the **Derived Part** can be seen (*Figure 10A-2P*); use the **Project Geometry** tool
to project the two horizontal edge lines of the **Derived Part** and the rightmost vertical
edge of the **Contour Flange** sheet-metal feature onto the sketch (*Figure 10A-2Q*); select
the **Center Point Circle** tool, click the lower endpoint of the projected vertical edge line
to set the center, then click the left endpoint of the lower projected horizontal edge line to
define the radius and create a circle; create another concentric circle with the radius
definition point click-set at the left endpoint of the top projected horizontal edge line;
create another concentric circle large enough to enclose the whole 3D model (*Figure
10A-2R*); click the **Return** button to exit the sketch. Save the file.
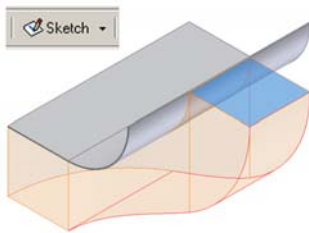
*Figure 10A-2N:
Selecting the top
square surface to start
the Compound Curve
Cut Profile sketch.*
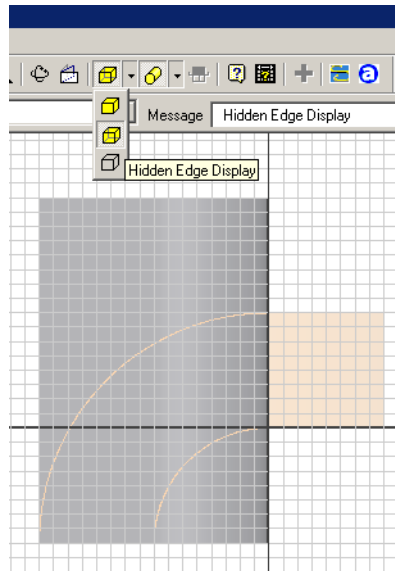
*Figure 10A-2P: Selecting
the Hidden Edge Display
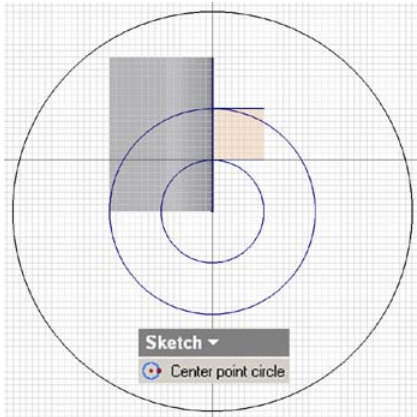mode.*
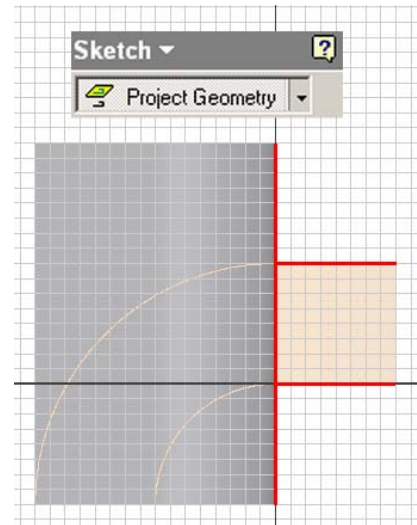
*Figure 10A-2Q: projecting
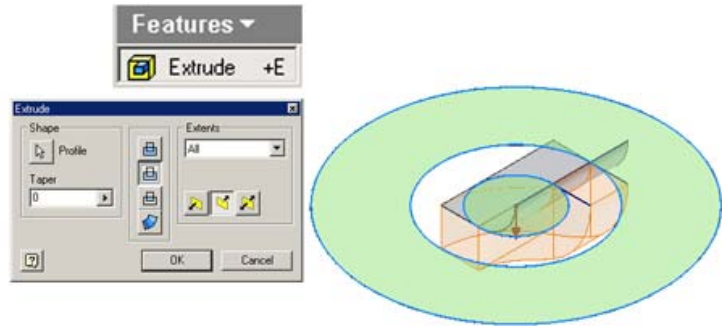the edge lines.*

Figure 10A-2S: Cutting the top compound curved elbow piece with the Cut option of the Extrude tool.

Figure 10A-2R: Creating three concentric circles.

Next, select the **Extrude** tool; in the tool's dialog window, click the **Profile** arrow button and click-select on the screen the are between the largest circle and the medium circle, plus the area of the smallest circle; select the **Cut** as **Type**, **All** for **Extents**, and a downward direction; click the **OK** button to create the **Extrude** feature (*Figure 10A-2S*); and rename it *Compound Curve Cut* in the **Model** panel. The top piece is basically completed. Save the file.

Next, click-select the top surface of the top piece, and click the **Flat Pattern** tool button in the **Sheet Metal** tool bar (Figure 10A-2T); the **Flat Pattern** window opens (*Figure 10A-2U*). All features are listed in the **Model** panel (*Figure 10A-2V*). Save and close the file.
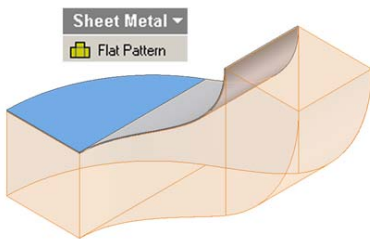

Figure 10A-2T: click-selecting the top surface of the top piece for the creation of a Flat Pattern view.
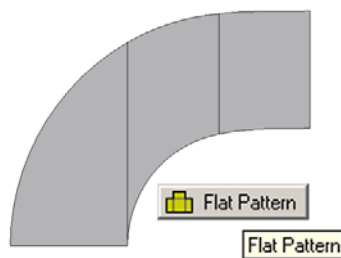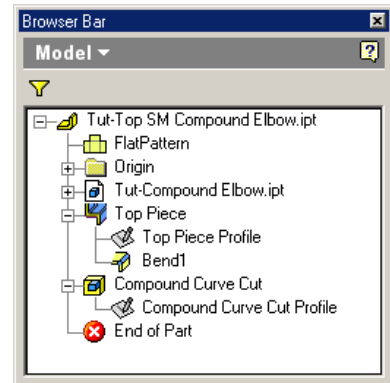

Figure 10A-2U: The Flat Pattern window.

Figure 10A-2V: All features are listed in the Model panel.

Step 2: Creating a Sheet Metal (in).ipt files for the base piece of the compound elbow

The method of creating the base piece of the compound elbow is similar to the method of creating the top piece. Open the *Tut-Base SM Compound Elbow.ipt* created as a **Save A Copy As** file when creating the top piece in the *Tut-Top SM Compound Elbow.ipt* file.

Select the upright square surface at the base of the model as shown in *Figure 11A-3A* and click on the **Sketch** tool button on the **Command Bar** to start a new sketch; rename it as *Base Piece Profile* in the **Model** panel; click the **Look At** button from the **Command Bar** and click-select the *Top Piece Profile* from the **Model** panel to switch to an orthographic normal view; select the **Project Geometry** tool to project the two top horizontal edge line (*Figure 10A-3B*); use the **Line** tool and the **General Dimension** tool to create a vertical line starting at the right endpoint of the projected top horizontal edge line and extending upward, and with a height of 8 inches (the "circle radius line," as shown in *Figure 10A-3C*); select the "circle radius line" and go to the **Command Bar** to change its **Style** to Centerline; next, use the **Center Point Circle** tool to create a circle centered at the top endpoint of the "circle radius line" and with the circumference point set at the right endpoint of the projected lower edge line; next, select the **Line** tool again to create additional horizontal and vertical lines radiating from the center of the circle and extending beyond the circumference for trimming the circle (*Figure 10A-3D*); next, use the **Trim** tool to trim off unneeded line segments (*Figure 10A-3E*), so as to leave a connected open profile consisting of a horizontal line and a quarter arc, as shown in *Figure 10A-3F*; click the **Return** button to exit the sketch. Go to the **View**→**Isometric** menu to switch to isometric view (*Figure 11A-3G*). Save the file.
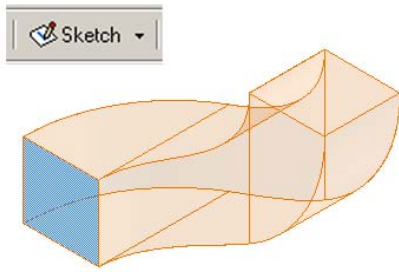
Figure 11A-3A: Selecting the upright square surface at the base of the model as shown to start a new sketch.
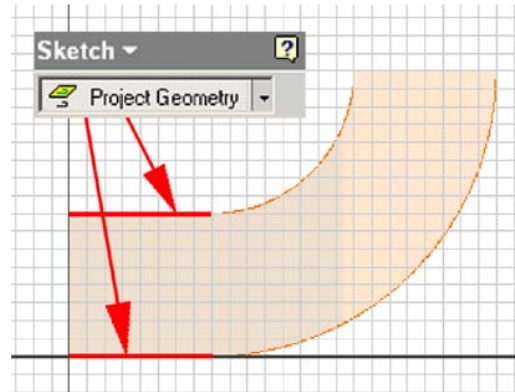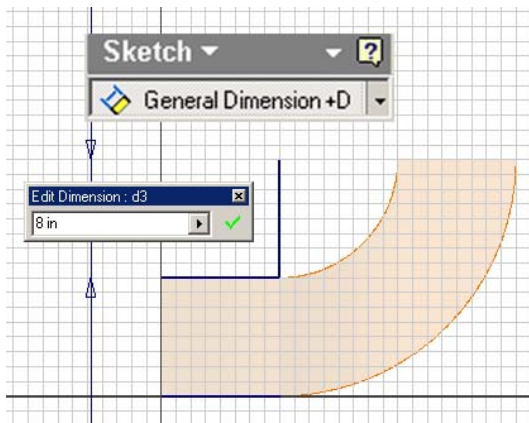

Figure 11A-3B: Projecting the edge lines.

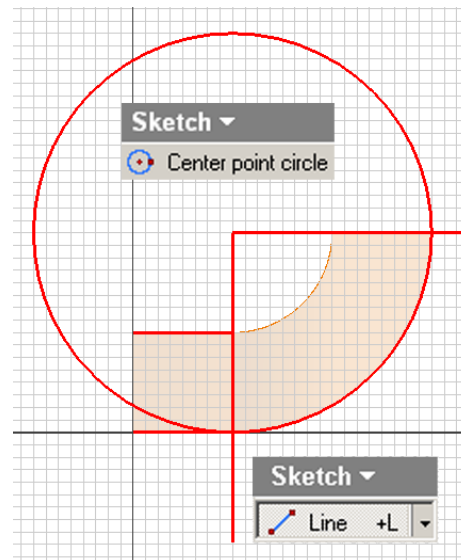*Figure 11A-3C: Creating the "circle radius line."*

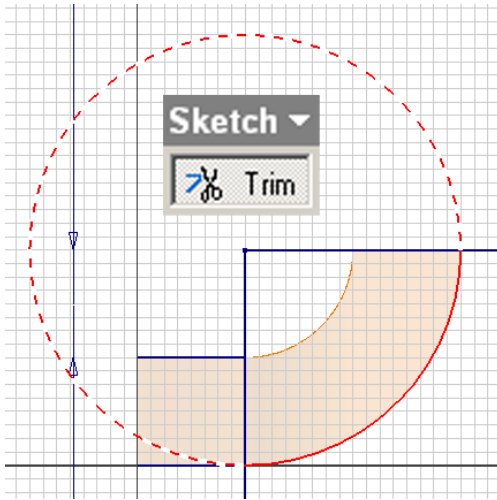*Figure 11A-3D: Creating the circle and additional lines.*

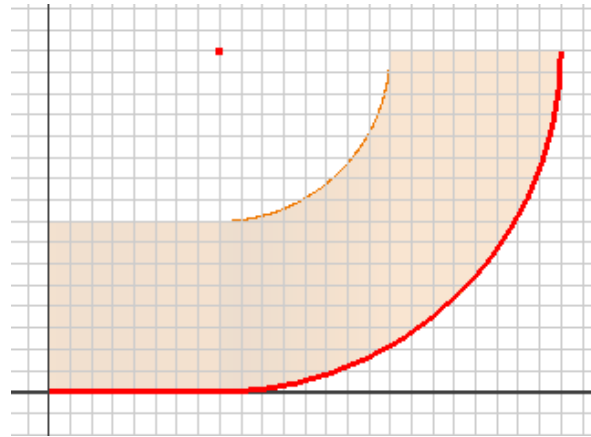*Figure 11A-3E: Trimming off
unneeded line segments.*

*Figure 11A-3F: The completed Top
Piece Profile.*

Next, select the **Contour Flange** tool; in the tool's dialog window, click the
Profile button and then click-select the open profile; type 24 in (inch) in the **Distance**
text field; click the **Flip Direction** button next to the text field to make sure that the
Contour Flange is extruded in the correct direction as shown in *Figure 10A-3H*; use the
Zoom Window tool to zoom in the lower left corner area of the green outline geometry;
click the **Offset** button in the **Shape** Section to make sure that the **Contour Flange** is
extruded outwardly (downward) from the enclosed space of the compound elbow defined
by the **Work Surface** geometry (*Figure 10A-3J*); click the **OK** button to create the
Contour Flange feature; and rename it as *Base Piece* in the **Model** panel.
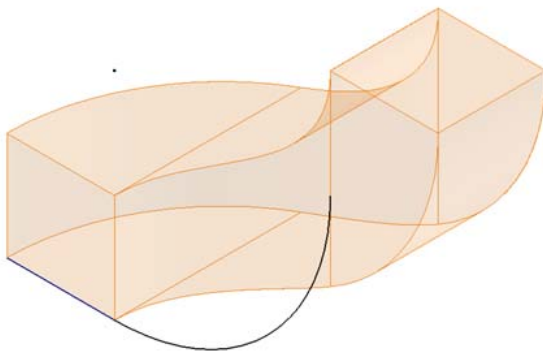
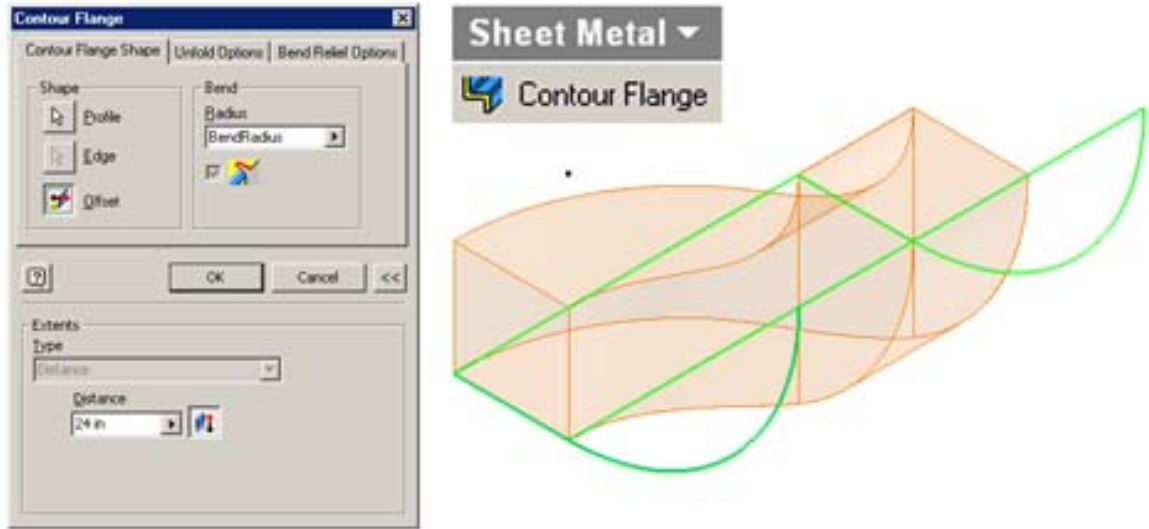*Figure 11A-3G: Switching to the
isometric view.*

Figure 11A-3H: Creating the Contour Flange feature.

Next, select the squared top surface of the 3D **Placed Component** model, as shown in *Figure 10A-3K*, and click the **Sketch** button on the **Command Bar** to start a new sketch; rename it *Compound Curve Cut Profile*; use the **Project Geometry** tool to project the three horizontal edge lines of the **Derived Part** and the rightmost vertical edge of the **Contour Flange** sheet-metal feature onto the sketch (*Figure 10A-3L*); window-select all projected lines and go to the **Command Bar** to change their line **Style** to **Normal**; since the *Base Piece* **Contour Flange** feature has been extruded outward beyond the edge of the **Work Surface** of the **Derived** Part, there are gaps between the two projected horizontal and vertical lines on the right-hand side; to prevent gaps in the profile, select the **Extend** tool, click on the right-hand side of the projected horizontal lines so as to extend them to the projected vertical line (*Figure 11A-3M*); next, select the **Trim** tool to trim off the top and lower portions of the projected vertical line (*Figure 10A-3N*); next, use the **Line** tool and the **General Dimension** tool to create an 8-inch vertical line starting at the left endpoint of the lower horizontal projected line on the right-hand side and extending downward (the "circle radius line", as shown in *Figure 10A-3P*); next, select the **Center Point Circle** tool, click the lower endpoint of the "circle radius line" to set the center, then click the left endpoint of the lower projected horizontal edge line on the right-hand side to define the radius and create a circle; create another concentric circle with the radius definition point click-set at the left endpoint of the top projected horizontal edge line; select the **Line** tool again to create another vertical line starting at the center of the circles and extending upward beyond the circumference of the larger circle (*Figure 10A-3Q*); next, select the **Trim** tool again to trim off all unneeded line segments of the circles plus the last drawn vertical line, as shown in *Figure 10A-3R*, so as to leave a closed profile (the "compound elbow profile," as shown in *Figure 10A-*

3S); next, select the **Two Point Rectangle** tool to create a rectangle large enough to enclose the entire **Contour Flange** feature (*Figure 10A-3T*); The *Compound Curve Cut Profile* is completed; click the **Return** button to exit the sketch. Save the file.
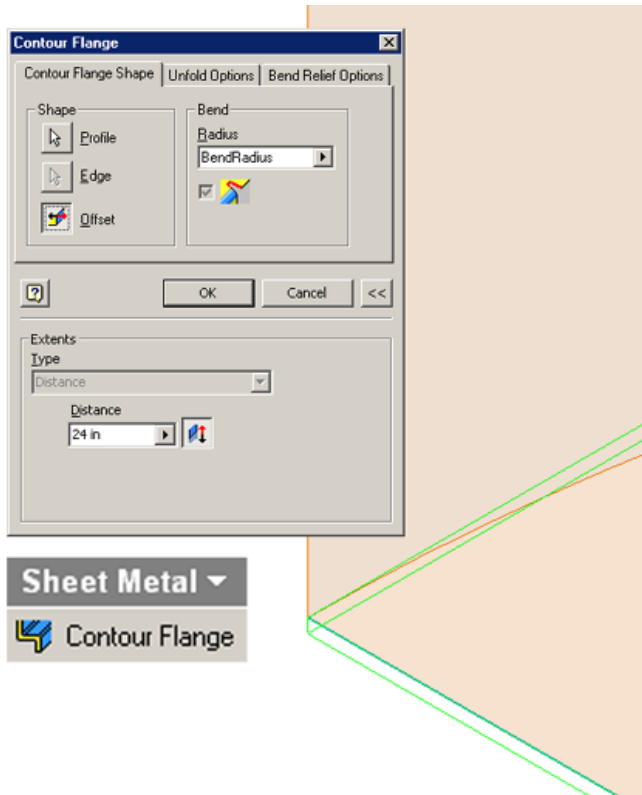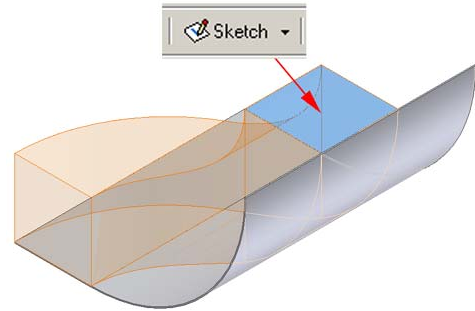

Figure 11A-3K: Selecting the top square surface to start the Compound Curve Cut Profile sketch.

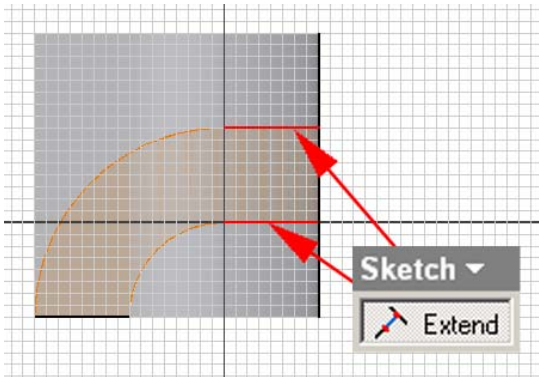*Figure 11A-3J: Clicking the Offset button to make sure that the Contour Flange is extruded outwardly (downward) from the enclosed space of the compound elbow.*
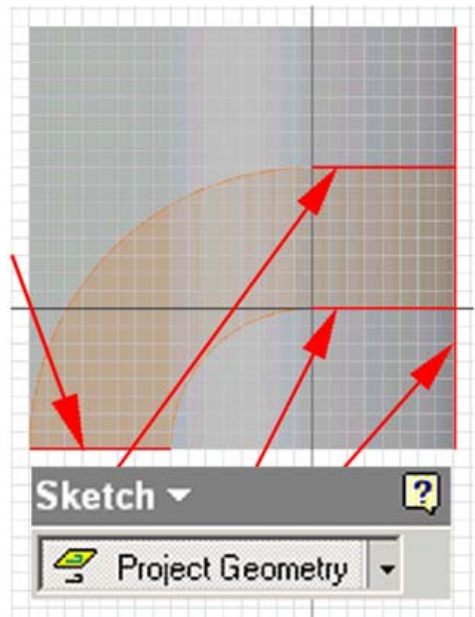
Figure 11A-3L: Projecting the edge lines onto the sketch.

Figure 10A-3M: Extending the endpoints of the two horizontal lines to the edge of the vertical line.
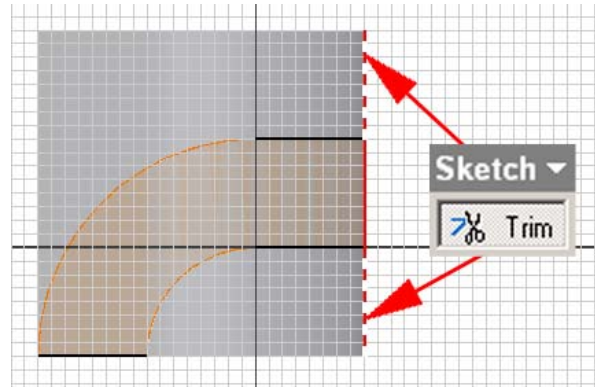
Figure 10A-3N: trimming off the top and lower portions of the projected vertical line.
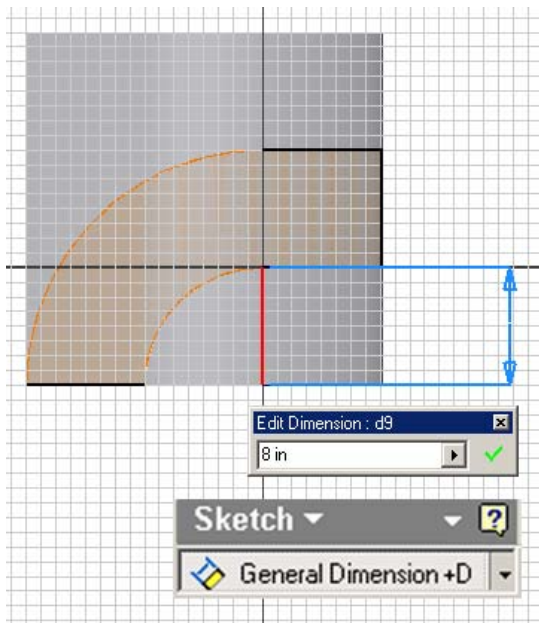
Figure 10A-3P: Creating an 8-inch vertical line.
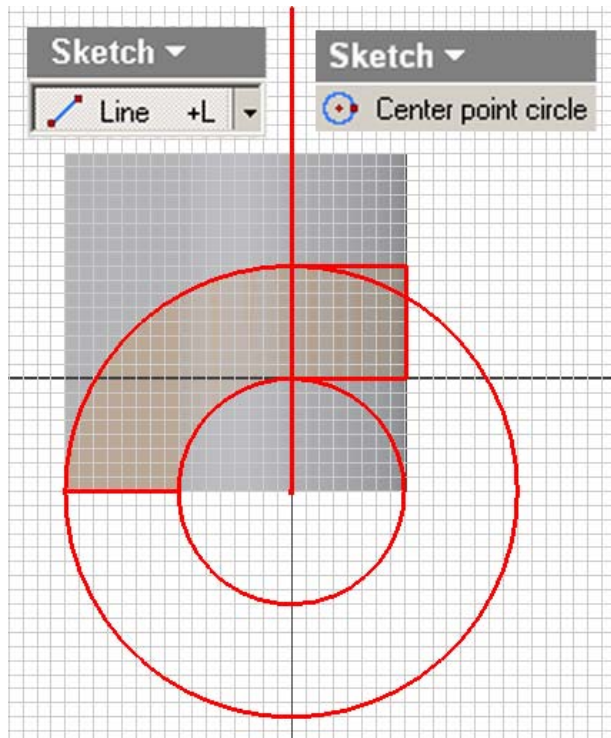
Figure 10A-3Q: Creating two concentric circles.
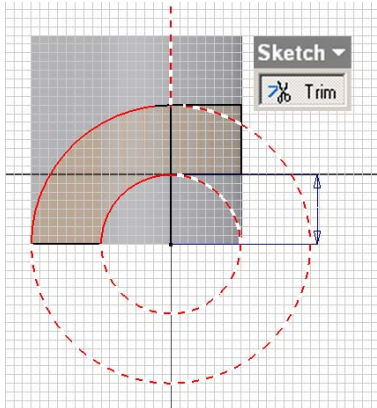
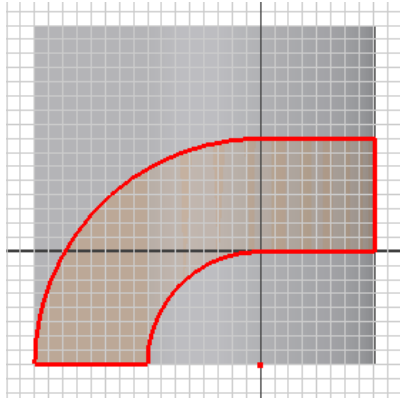*Figure 10A-3R: Trimming off segments of the circles and the last vertical line.*

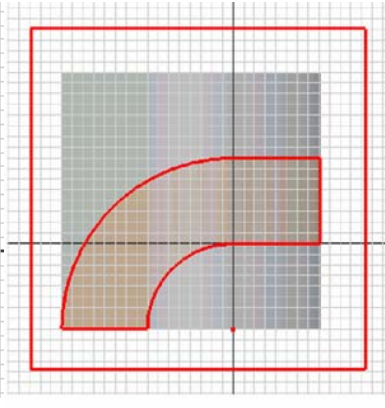*Figure 10A-3S: The closed profile.*
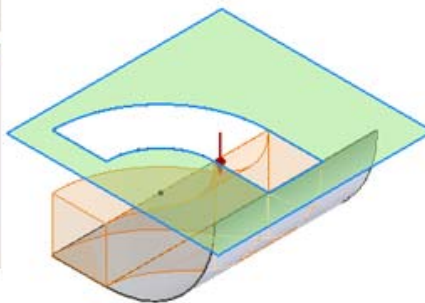
Figure 10A-3T: The completed Compound Curve Cut Profile sketch.

Figure 10A-3V: The completed base piece.

Figure 10A-3U: Cutting the Contour Flange.

Next, go to the **View**→**Isometric** menu to switch to an isometric view; select the **Extrude** tool; in the tool's dialog window, click the **Profile** arrow button and click-select on the screen the area between the "compound elbow profile" and the rectangle; select the **Cut** as **Type**, **All** for **Extents**, and a downward direction; click the **OK** button to create the **Extrude** feature (*Figure 10A-3U*); and rename it *Compound Curve Cut* in the **Model** panel. The base piece is basically completed (*Figure 10A-3V*). Save the file.

Next, click-select the exterior surface of the base piece, and click the **Flat Pattern** tool button in the **Sheet Metal** tool bar (*Figure 10A-3W*); the **Flat Pattern** window opens

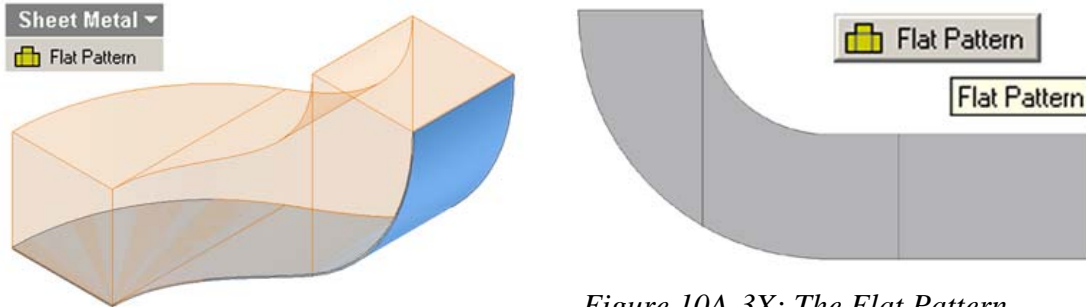(*Figure 10A-3X*). All features are listed in the **Model** panel (*Figure 10A-3Y*). Save and close the file.

Figure 10A-3W: Selecting the exterior surface for the generation of the Flat Pattern view.
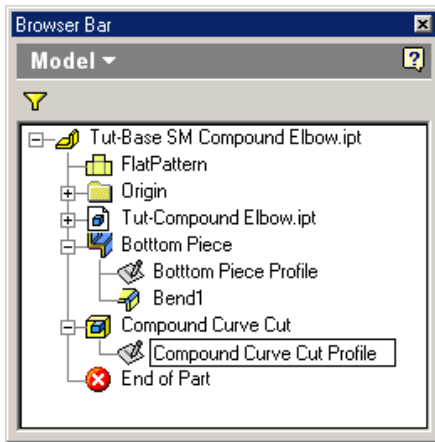

Figure 10A-3X: The Flat Pattern.

Figure 10A-3Y: All features are listed in the Model panel.

Step 3: Creating a Sheet Metal (in).ipt files for the left piece of the compound elbow

The method of creating the left piece of the compound elbow is similar to the method of creating the top and base pieces. Open the *Tut-Left SM Compound Elbow.ipt* created as a **Save A Copy As** file when creating the left piece in the *Tut-Left SM Compound Elbow.ipt* file.

Select the leveled top square surface at the top-right corner of the model as shown in *Figure 11A-4A* and click on the **Sketch** tool button on the **Command Bar** to start a new sketch; rename it as *Left Piece Profile* in the **Model** panel; click the **Look At** button from the **Command Bar** and click-select the *Left Piece Profile* from the **Model** panel to

switch to an orthographic normal view; select the **Project Geometry** tool to project the
two top horizontal edge line (*Figure 10A-4B*); use the **Line** tool and the **General
Dimension** tool to create a vertical line starting at the left endpoint of the projected lower
horizontal edge line on the right-hand side and extending downward, and with a height of
8 inches (the "circle radius line," as shown in *Figure 10A-4C*); select the "circle radius
line" and go to the **Command Bar** to change its **Style** to Centerline; next, use the **Center
Point Circle** tool to create a circle centered at the lower endpoint of the "circle radius
line" and with the circumference point set at the left endpoint of the projected upper
horizontal edge line; next, select the **Line** tool again to create additional horizontal and
vertical lines radiating from the center of the circle and extending leftward and upward
beyond the circumference for trimming the unneeded portions of the circle of the circle
(*Figure 10A-4D*); next, use the **Trim** tool to trim off unneeded horizontal and vertical
lines, so as to leave a connected open profile consisting of a horizontal line and a quarter
arc, as shown in *Figure 10A-4E*; click the **Return** button to exit the sketch. Go to the
View➔**Isometric** menu to switch to isometric view. Save the file.
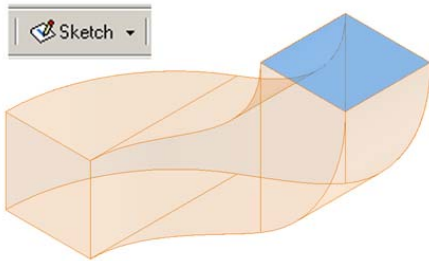
*Figure 10A-4A: Selecting the
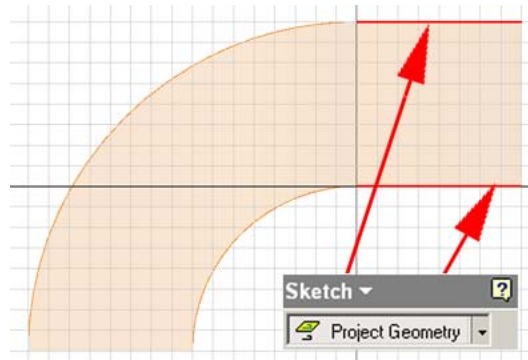top-right square surface to start
a new sketch.*
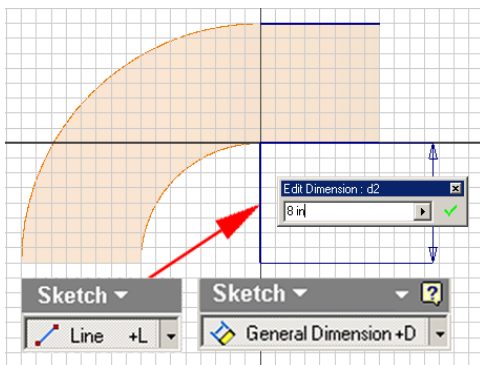

Figure 10A-4B: Projecting horizontal edge lines.

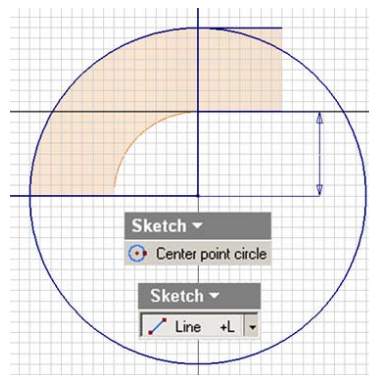

*Figure 10A-4C: Creating the
"circle radius line."*

*Figure 10A-4D:
Creating the
circle and
additional
horizontal and
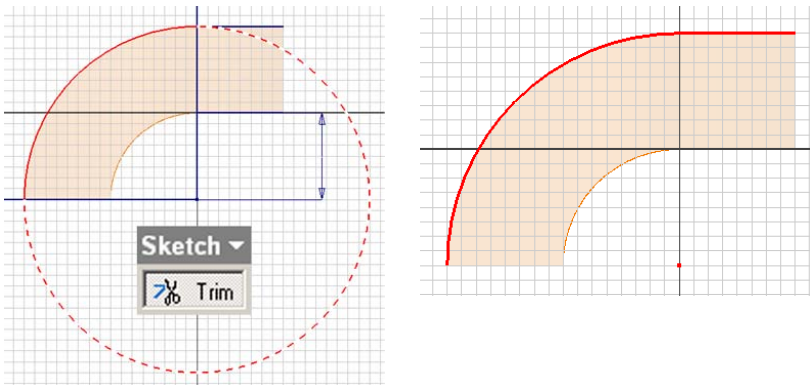vertical lines
for trimming the
unneeded
portions of the
circle.*

Figure 10A-4E: Trimming off the unneeded portion of the circle plus all unneeded horizontal and vertical lines (left); the competed profile (right).
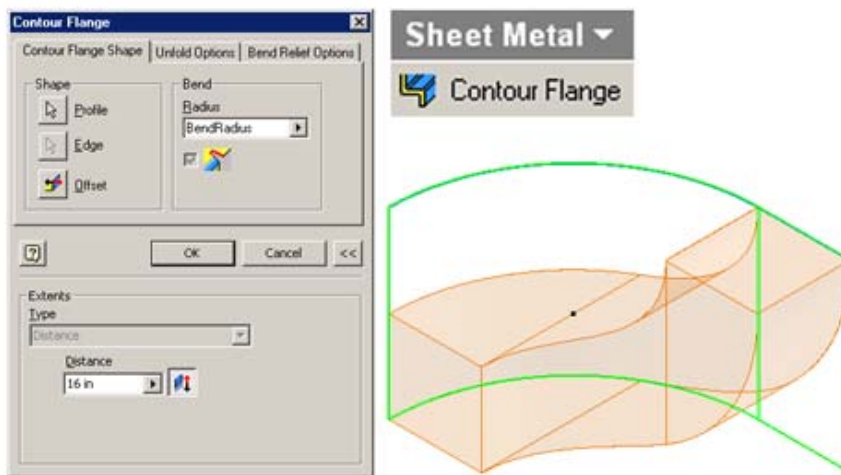

Figure 10A-4F: Creating the Left Piece Contour Flange.

Next, select the **Contour Flange** tool; in the tool's dialog window, click the **Profile** button and then click-select the open profile; type 16 in (inch) in the **Distance** text field; click the **Flip Direction** button next to the text field to make sure that the **Contour Flange** is extruded in the correct direction as shown in *Figure 10A-4F*; use the **Zoom Window** tool to zoom in the lower left corner area of the green outline geometry; click the **Offset** button in the **Shape** Section to make sure that the **Contour Flange** is extruded outwardly (leftward) from the enclosed space of the compound elbow defined by the **Work Surface** geometry (*Figure 10A-4G*); click the **OK** button to create the **Contour Flange** feature (*Figure 10A-4H*); and rename it as *Left Piece* in the **Model** panel.
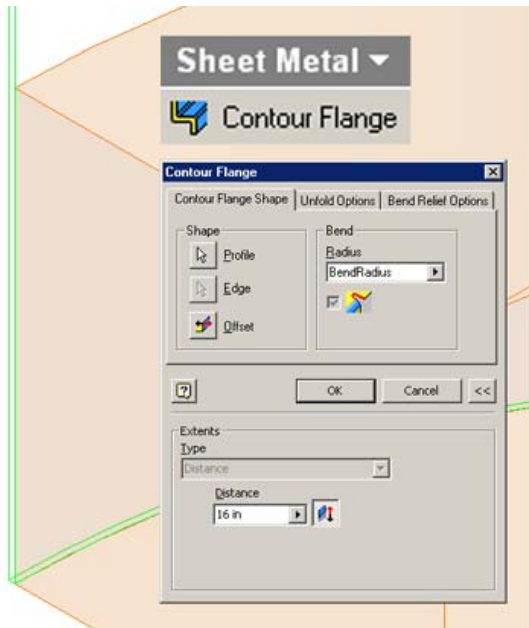
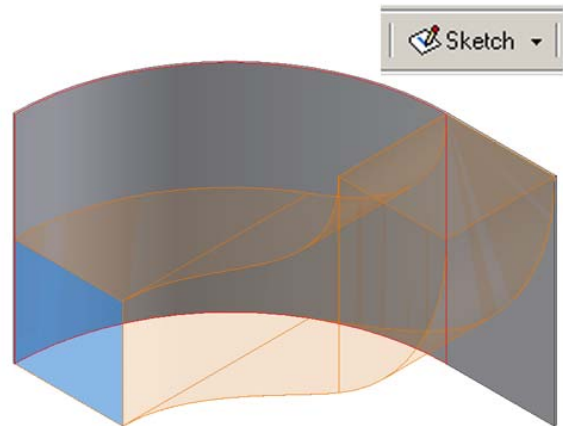*Figure 10A-4G:* The *Contour Flange extruded outwardly* (leftward).

Figure 10A-4H: The completed Left Piece.

Next, select the upright squared surface on the lower-left corner of the 3D **Placed Component** model, as shown in *Figure 10A-4H*, and click the **Sketch** button on the **Command Bar** to start a new sketch; rename it *Compound Curve Cut Profile*; use the **Project Geometry** tool to project the two horizontal edge lines of the **Derived Part** on the left-hand side and the leftmost vertical edge of the **Contour Flange** sheet-metal feature onto the sketch (*Figure 10A-2Q*), select the projected lines and go to the **Command Bar** to change their **Style** to **Normal**; since the *Left Piece* **Contour Flange** feature has been extruded outward beyond the edge of the **Work Surface** of the **Derived** Part, there are gaps between the two projected horizontal and vertical lines on the right-hand side; to prevent gaps in the profile, select the **Extend** tool, click on the left-hand side of the projected horizontal lines so as to extend them to the projected vertical line (*Figure 10A-4L*); next, select the **Trim** tool to trim off the top portion of the projected vertical line (*Figure 10A-4M*); next, select the **Line** and **General Dimension** tools to create a vertical line starting at the right endpoint of the upper projected horizontal line and extending upward with a height of 8 inches (*Figure 10A-4N*); next, select the **Center Point Circle** tool, click the upper endpoint of the "circle radius line" to set the center, then click the right endpoint of the upper projected horizontal edge line to define the radius and create a circle; create another concentric circle with the radius definition point click-set at the right endpoint of the lower projected horizontal edge line (*Figure 10A-4P*); next, select the **Trim** tool to trim off the unneeded portions of the concentric circles,

so as to leave a closed profile (the "compound elbow profile," as shown in (*Figure 10A-4Q*); next, select the **Two Point Rectangle** tool to create a rectangle large enough to enclose the entire **Contour Flange** feature (*Figure 10A-4R*); the *Compound Curve Cut Profile* is completed; click the **Return** button to exit the sketch. Save the file.
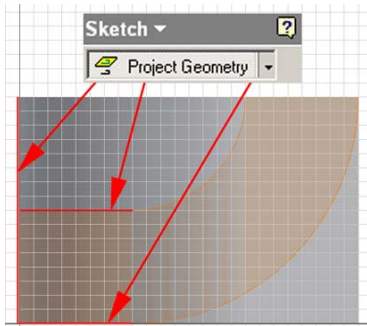

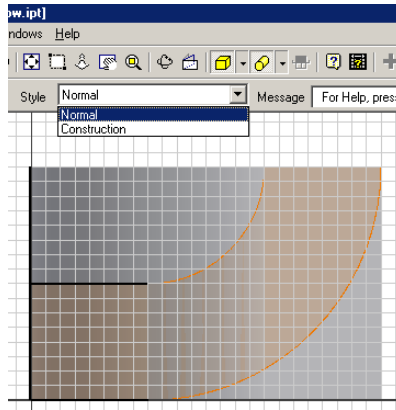*Figure 10A-4J: Projecting edge lines.*
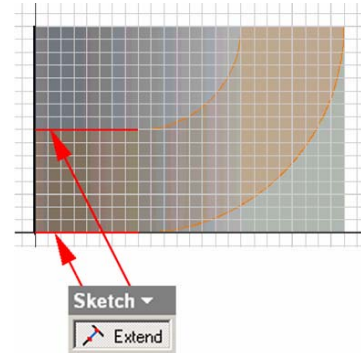

Figure 10A-K: Changing the Style of the projected lines.

Figure 10A-4L: Extending the two projected horizontal lines to the projected vertical line.
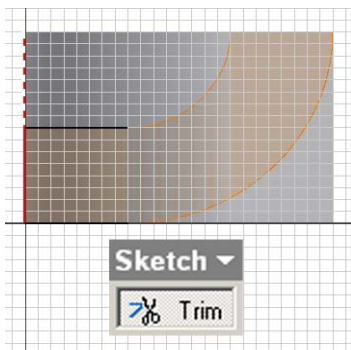
Figure 10A-4M: Trimming off the top portion of the projected vertical line.
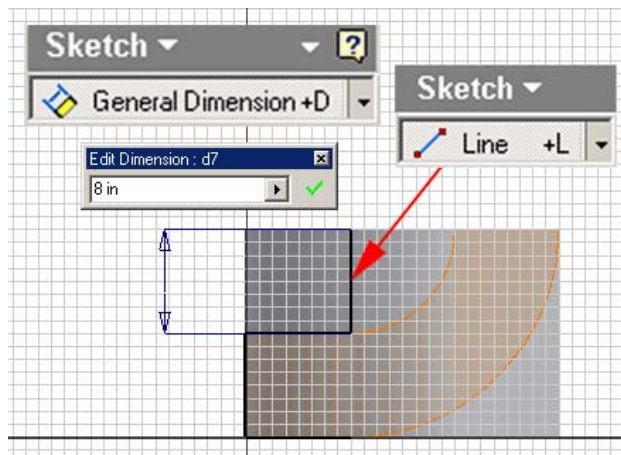
Figure 10A-4N: Creating the 8-inch "circle radius line."
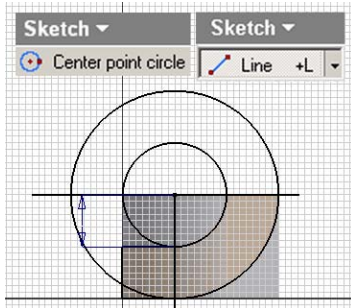
Figure 10A-4P: Creating two concentric circles.
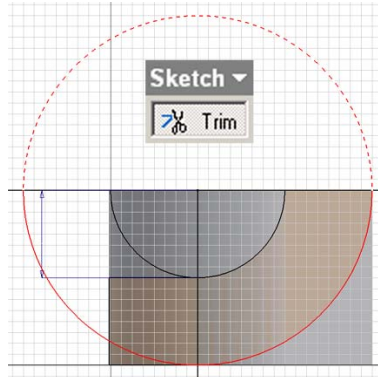
Figure 10A-4Q: Trimming off unneeded portions of the concentric circles.
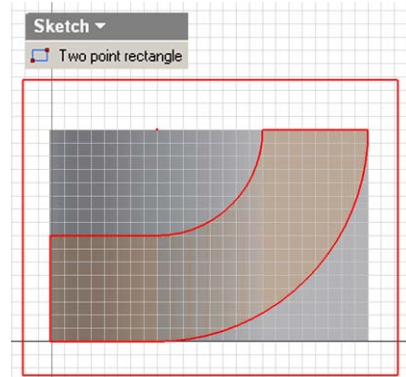
Figure 10A-4R: Creating a rectangle large enough to enclose the entire Contour Flange feature.

Next, select the **Extrude** tool; in the tool's dialog window, click the **Profile** arrow button and click-select on the screen the area between the rectangle and the "compound elbow profile;" select the **Cut** as **Type**, **All** for **Extents**, and a downward direction; click the **OK** button to create the **Extrude** feature (*Figure 10A-4S*); and rename it *Compound Curve Cut* in the **Model** panel. The top piece is basically completed (*Figure 10A-4T*). Save the file.

Next, select the **Rotate** tool to rotate the model to a convenient position where the exterior surface of the *Left Piece* **Contour Flange** feature can be click-selected (*Figure 10A-4U*); click-select the exterior surface of the left piece, and click the **Flat Pattern** tool button in the **Sheet Metal** tool bar (Figure 10A-4V); the **Flat Pattern** window opens (*Figure 10A-4W*). All features are listed in the **Model** panel (*Figure 10A-4X*). Save and close the file.
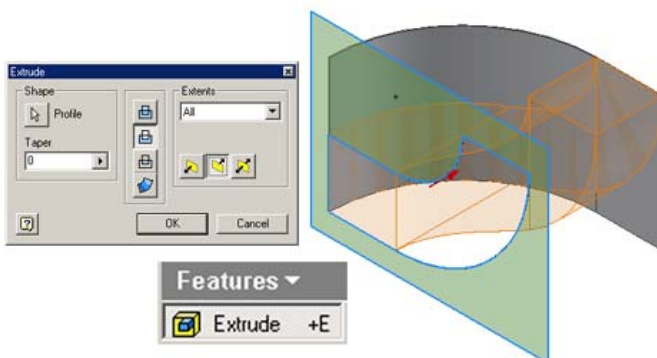
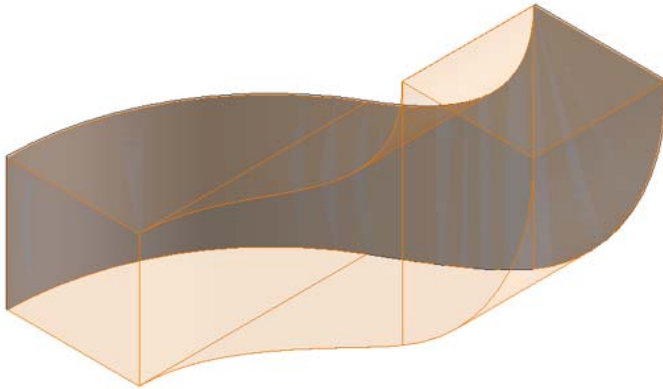*Figure 10A-4S: Cutting the Left Piece Contour Flange feature.*

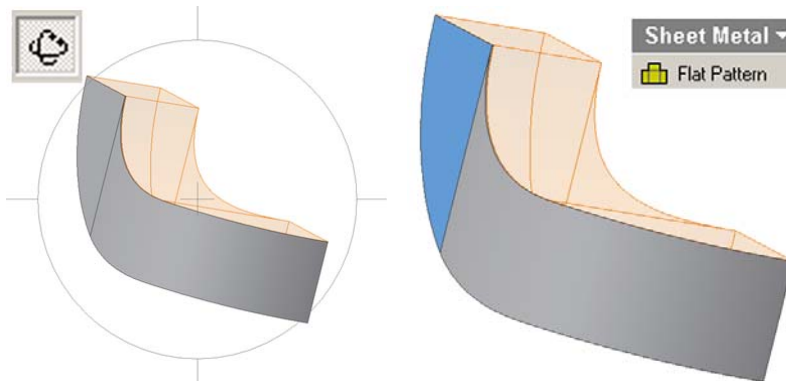*Figure 10A-4T: The completed left Piece sheet-metal part.*

Figure 10A-4V: Selecting the exterior surface for Flat Pattern projection.
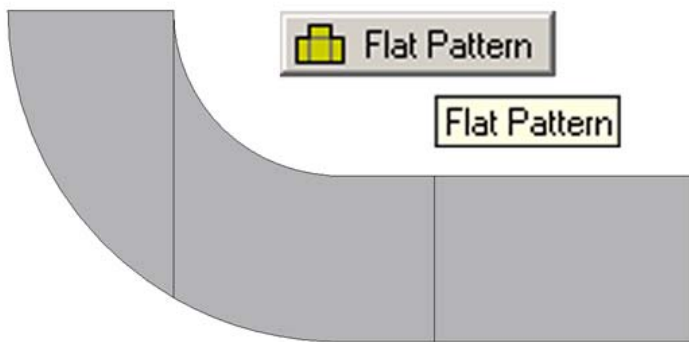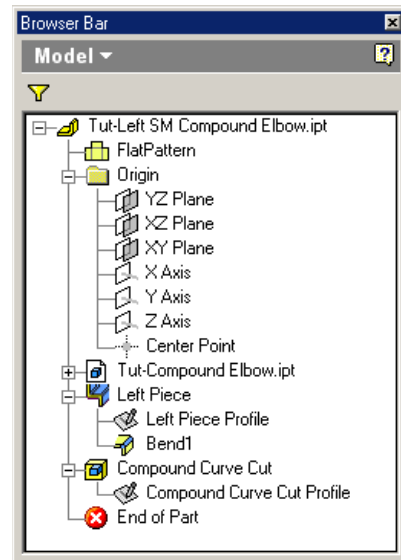
Figure 10A-4U: Rotating the model.

Figure 10A-4W: The Flat Pattern window.

Figure 10A-4X: All features are listed in the Model panel

Step 4: Creating a Sheet Metal (in).ipt files for the right piece of the compound elbow

The method of creating the right piece of the compound elbow is similar to the method of creating the left piece. Open the *Tut-Right SM Compound Elbow.ipt* created as a **Save A Copy As** file when creating the top piece in the *Tut-Top SM Compound Elbow.ipt* file.
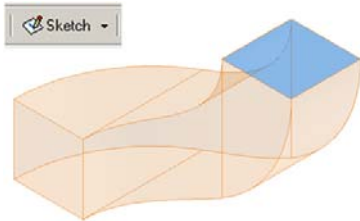
Figure 10A-5A: Selecting the top square surface to start the Right Piece Profile sketch.
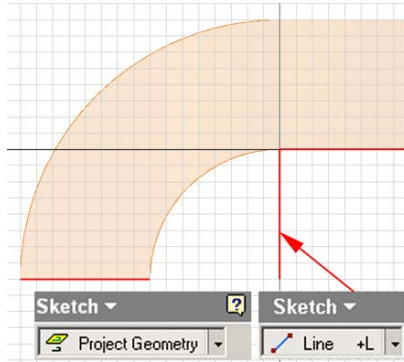
Figure 10A-5B: Projecting the horizontal edge lines and creating the vertical "circle radius line."
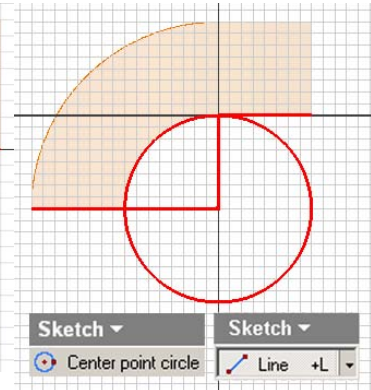
Figure 10A-5C: Creating the circle and an additional horizontal line for trimming the circle.
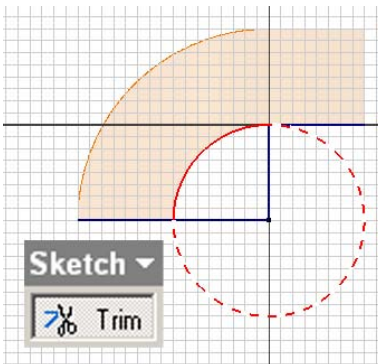
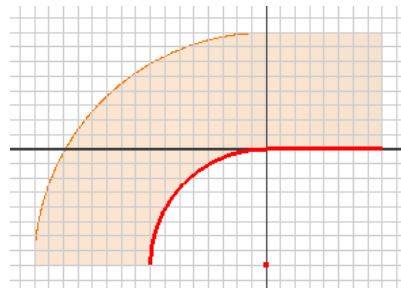*Figure 10A-5D: Trimming off the unneeded portion of the circle.*

Figure 10A-5E: The completed Right Piece profile sketch.

Select the leveled square surface at the top of the model as shown in *Figure 11A-5A* and click on the **Sketch** tool button on the **Command Bar** to start a new sketch; rename it as *Right Piece Profile* in the **Model** panel; click the **Look At** button from the **Command Bar** and click-select the *Right Piece Profile* from the **Model** panel to switch

to an orthographic normal view; select the **Project Geometry** tool to project the two lower horizontal edge lines; use the **Line** tool and the **General Dimension** tool to create a vertical line starting at the left endpoint of the projected lower horizontal edge line on the right-hand side and extending downward, and with a height of 8 inches (the "circle radius line," as shown in *Figure 10A-5B*); next, use the **Center Point Circle** tool to create a circle centered at the lower endpoint of the "circle radius line" and with the circumference point set at the left endpoint of the projected lower edge line on the right-hand side; next, select the **Line** tool again to create another horizontal line radiating from the center of the circle and ending at the right endpoint of the projected lower horizontal edge line for trimming the circle (*Figure 10A-5C*); next, use the **Trim** tool to trim off unneeded line segments (*Figure 10A-5D*), so as to leave a connected open profile consisting of a horizontal line and a quarter arc, as shown in *Figure 10A-5E*; click the **Return** button to exit the sketch. Go to the **View→Isometric** menu to switch to isometric view. Save the file.
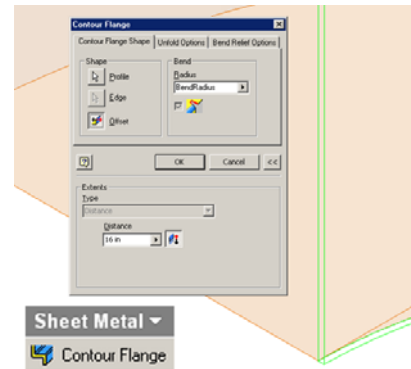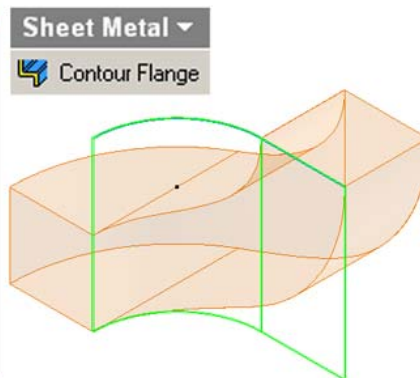

Figure 10A-5F: Creating the Contour Flange feature.

Figure 10A-5G: Ensuring that the Contour Flange is extruded outward from the Work Surface of the Derived Part of the compound elbow' enclosed space.

Next, select the **Contour Flange** tool; in the tool's dialog window, click the **Profile** button and then click-select the open profile; type 16 in (inch) in the **Distance** text field; click the **Flip Direction** button next to the text field to make sure that the **Contour Flange** is extruded in the correct direction as shown in *Figure 10A-5F*; use the **Zoom Window** tool to zoom in the lower left corner area of the green outline geometry, which is adjacent to the lower portion of the **Derived Part** model; click the **Offset** button in the **Shape** Section to make sure that the **Contour Flange** is extruded outwardly (rightward) from the enclosed space of the compound elbow defined by the **Work**

Surface geometry (*Figure 10A-5G*); click the **OK** button to create the **Contour Flange** feature; and rename it as *Right Piece* in the **Model** panel. Save the file.

Next, select the upright squared surface at the lower left portion of the 3D **Placed Component** model, as shown in *Figure 10A-5H*, and click the **Sketch** button on the **Command Bar** to start a new sketch; rename it *Compound Curve Cut Profile*; go to the **Command Bar** and select **Hidden Edge Display** from the pull-down menu, so that hidden curved edges of the 3D model of the **Derived Part** can be seen (*Figure 10A-5J*); use the **Project Geometry** tool to project the leftmost vertical edge line of the *Left Piece* **Contour Flange sheet-metal** feature and the top horizontal edge line of the **Derived Part** onto the sketch (*Figure 10A-5K*); next, select the **Center Point Circle** tool, click the top endpoint of the projected vertical edge line to set the center, then click the left endpoint of projected horizontal edge line to define the radius and create a circle; create another concentric circle with the radius definition point click-set at the right endpoint of the projected horizontal edge line; select the **Line** tool to create a horizontal line starting at the center of the circle and extending leftward beyond the circumference of the circle (*Figure 10A-5L*); next, select the **Trim** tool to trim off all unneeded portions of the concentric circles and of horizontal and vertical lines (*Figure 10A-5M* through *Figure 10A-5R*), so as to leave a closed "compound elbow profile" (*Figure 10A-5S*); next, select the **Two Point Rectangle** tool to create a rectangle large enough to enclose the whole 3D model (*Figure 10A-5T*); the *Compound Curve Cut Profile* is completed; click the **Return** button to exit the sketch. Save the file.
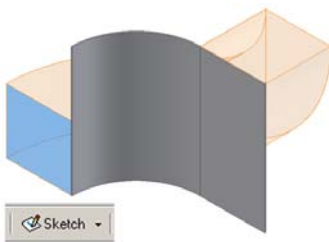

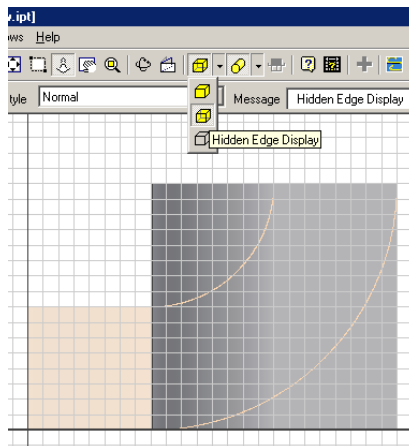*Figure 10A-5H: Selecting the upright squared surface to start the Compound Curve Cut Profile sketch.*

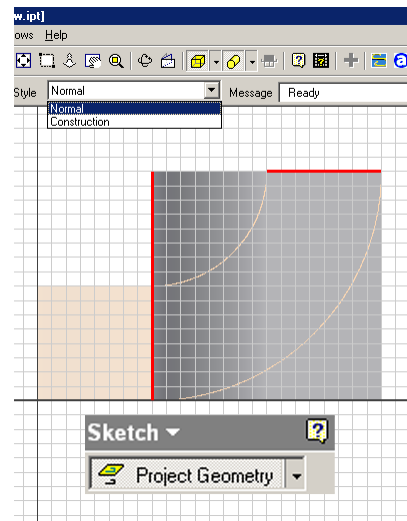*Figure 10A-5J: Switching to the Hidden Edge Display mode in the Command Bar.*

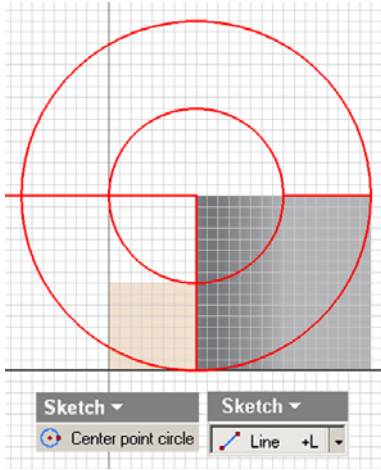*Figure 10A-5K: Projecting the edge lines.*

Figure 10A-5L: Creating concentric circles and the horizontal line for trimming the circle.
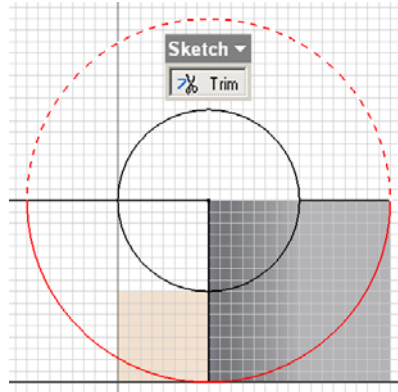

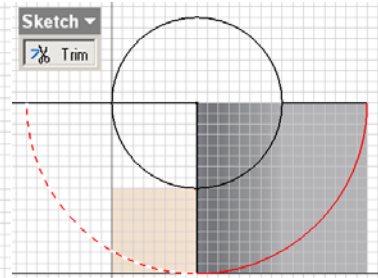*Figure 10A-5M: Trimming off the unneeded portions of the circles.*

Figure 10A-5N: Trimming off the unneeded portions of the circles.
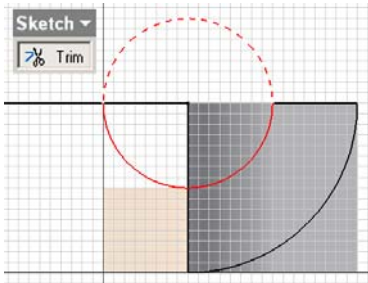

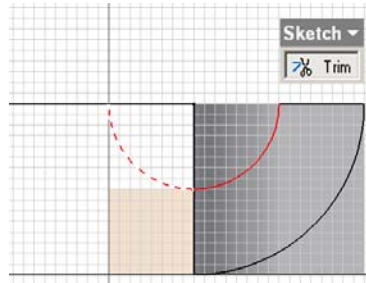*Figure 10A-5P: Trimming off the unneeded portions of the circles.*

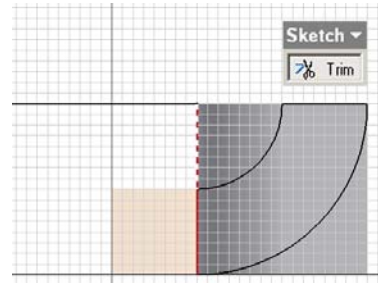*Figure 10A-5Q: Trimming off the unneeded portions of the circles.*

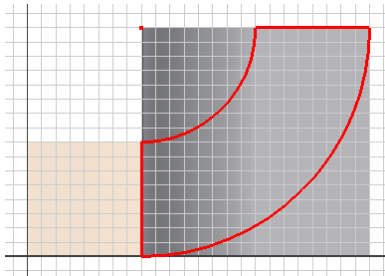*Figure 10A-5R Trimming off the unneeded line segment.*
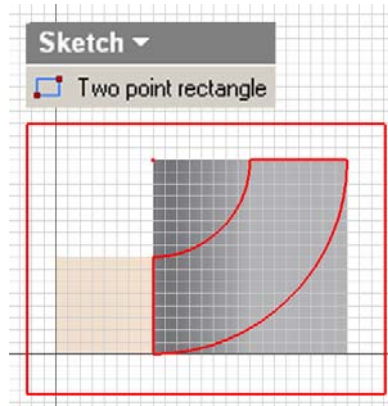
*Figure 10A-5S: The closed
"compound elbow profile."*

*Figure 10A-5T: The
completed Compound
Curve Cut Profile sketch.*

Next, select the **Extrude** tool; in the tool's dialog window, click the **Profile** arrow button and click-select on the screen the area between the rectangle and the "compound elbow profile;" select the **Cut** as **Type**, **All** for **Extents**, and an inward direction; click the **OK** button to create the **Extrude** feature (*Figure 10A-5U*); and rename it *Compound Curve Cut* in the **Model** panel. The right piece is basically completed (*Figure 10A-5V*). Save the file.

Next, click-select the exterior surface of the right piece, and click the **Flat Pattern** tool button in the **Sheet Metal** tool bar (*Figure 10A-5W*); the **Flat Pattern** window opens (*Figure 10A-5X*). All features are listed in the **Model** panel (*Figure 10A-5Y*). Save and close the file.
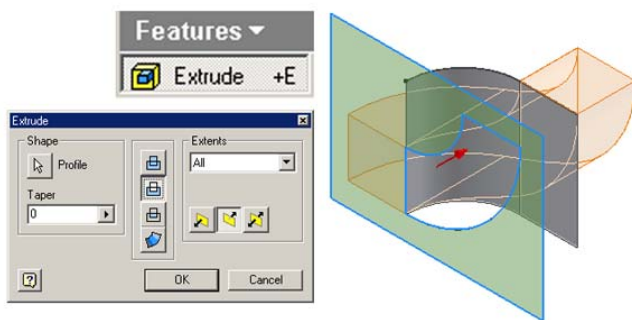
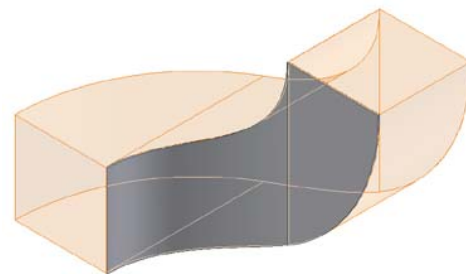*Figure 10A-5U: Cutting the Right Piece Contour
Flange feature.*

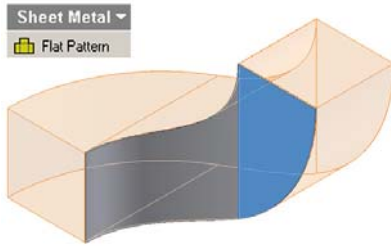*Figure 10A-5V: The completed
Right Piece.*

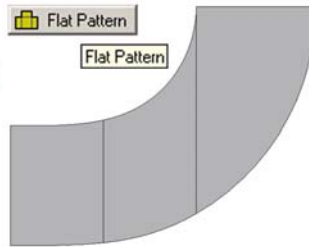*Figure 10A-5W: Selecting the exterior surface for the Flat pattern projection.*
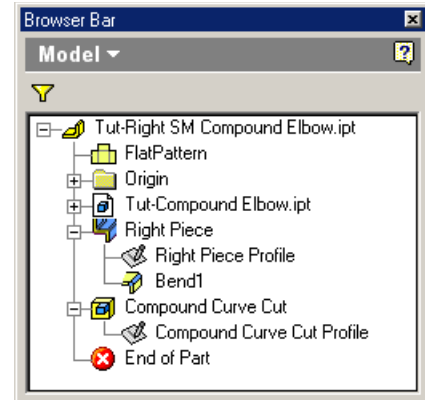

Figure 10A-5X: The Flat Pattern window.

Figure 10A-5Y: All features are listed in the Model panel.

Section 3:
Assembling The Top, Base, Left and Right Pieces Into The Compound Elbow

Go to **File→New**, or click on the **New** icon on **Standard Tool Bar** to start a new Inventor assembly file; the **Open** window appears. Select the **English** tab; then select the **Standard (in).iam** template, click **OK** button. A new assembly file opens.

Select the **Place Component** tool; in the **Open** dialog window, navigate to the *Tut-Compound Elbow* folder and select the *Tut-Base SM Compound Elbow.ipt* file; and click the **OK** button (*Figure 3B-6A*); the 3D folded model of the *Tut-Base SM Compound Elbow.ipt* file appears on the screen; click once on the screen to place one occurrence of the component; right-click for the shortcut menu and choose the **Done** option to exit the tool (*Figure 3B-6B*). The *Tut-Base SM Compound Elbow.ipt1* **Place Component** feature appears in the **Model** panel. Save the file as *Tut-Compound Elbow.iam* in the same *Tut-Compound Elbow* folder. Save often.
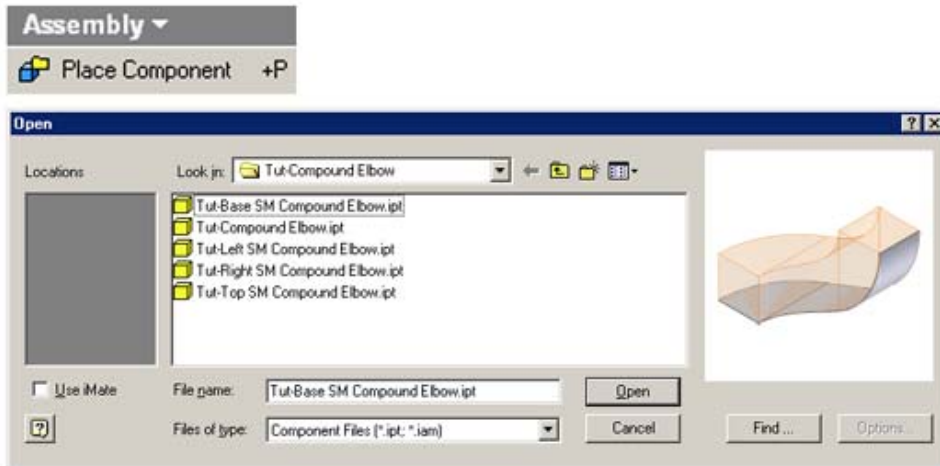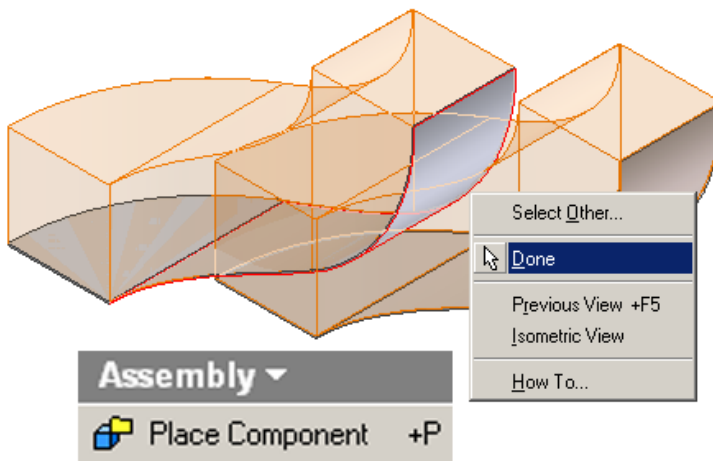
Figure 10A-6A: Selecting and opening the Tut-Base SM Compound Elbow.ipt file.

*Figure 10A-6B: Placing one occurrence of the component; right-click for the shortcut menu and choosing the **Done** option to exit the tool.*
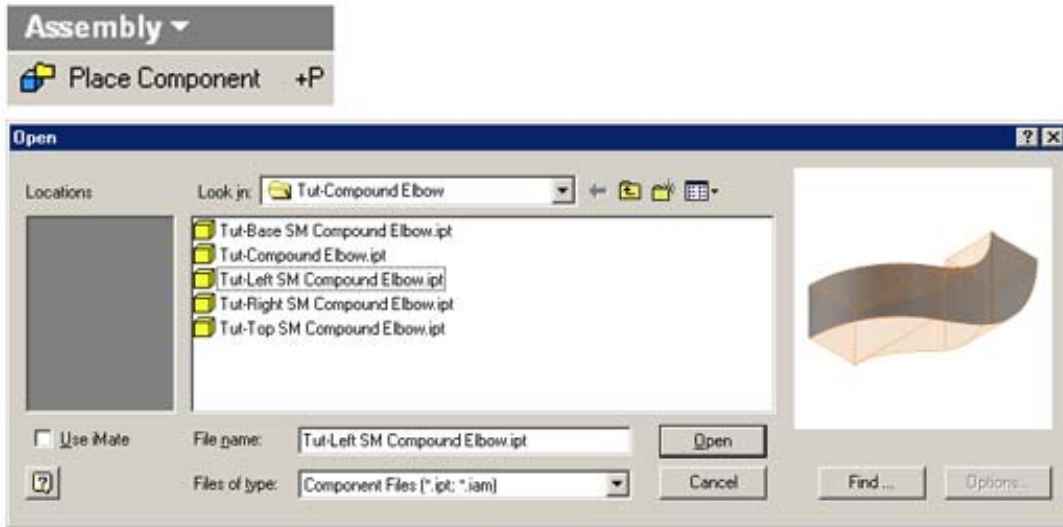
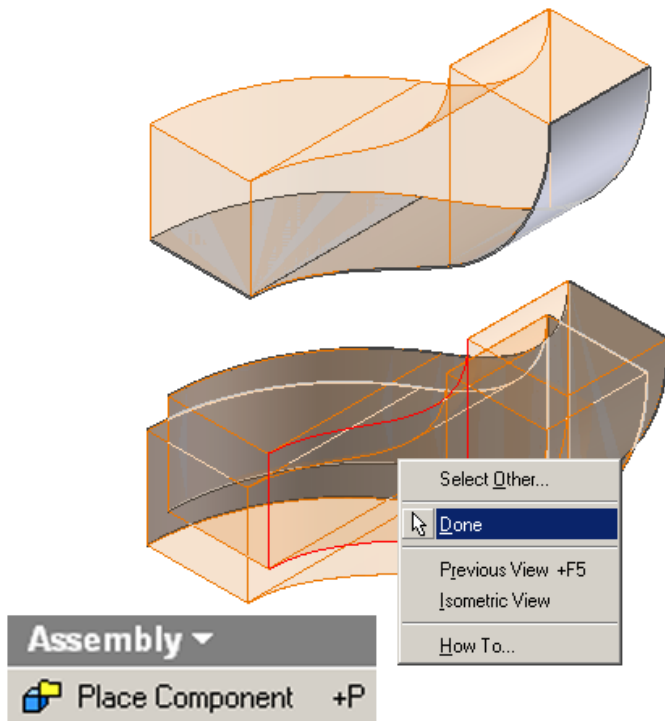*Figure 10A-6C: Opening the Tut-Left SM Compound Elbow.ipt file.*

Figure 10A-6D: Placing one copy of the model.

Next, repeat the same steps to place one copy of the 3D sheet-metal part from the *Tut-Left SM Compound Elbow.ipt*, *Tut-Right SM Compound Elbow.ipt*, and *Tut-Top SM Compound Elbow.ipt* files into the assembly file (*Figure 10A-6C* through *Figure 10A-6H*). Save the file often.
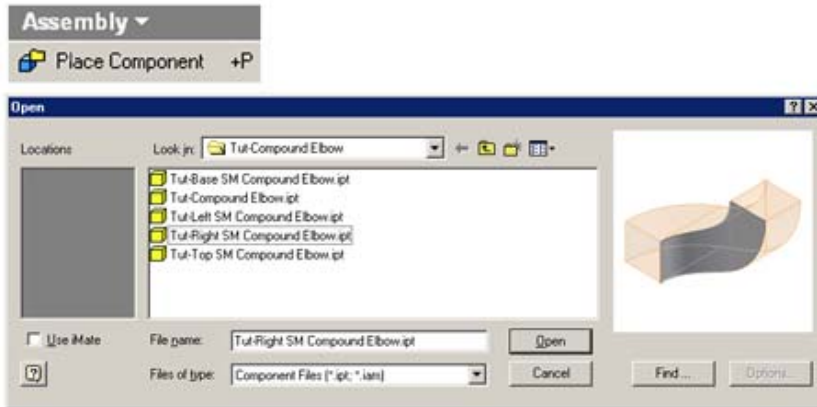

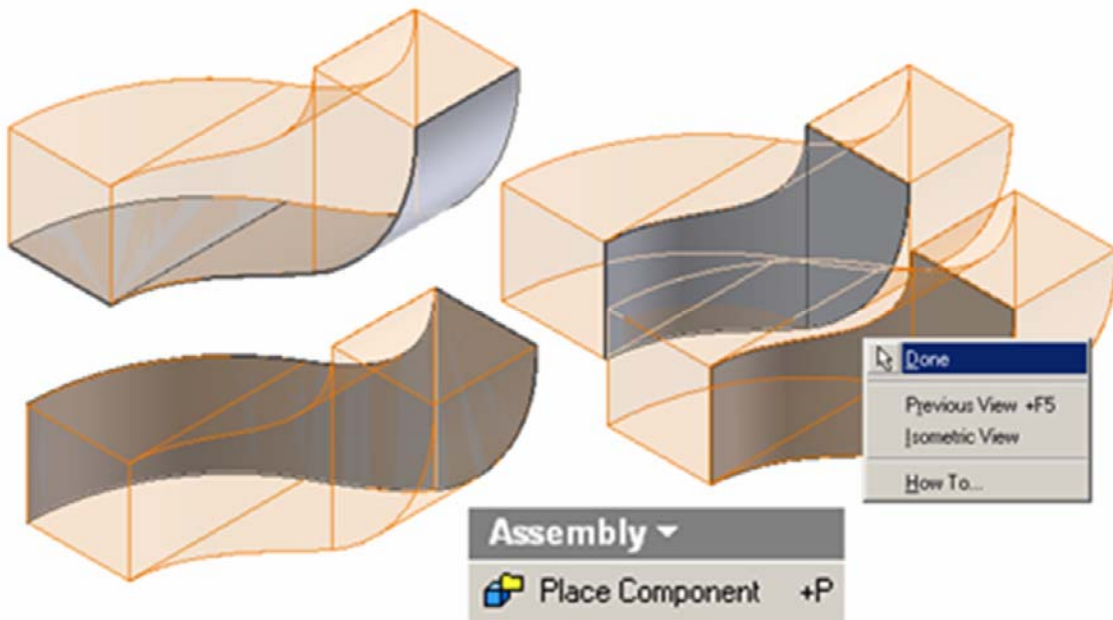*Figure 10A-6E: Opening the Tut-Right SM Compound Elbow.ipt file.*

Figure 10A-6F: Placing one copy of the model from the Tut-Right SM Compound Elbow.ipt file.
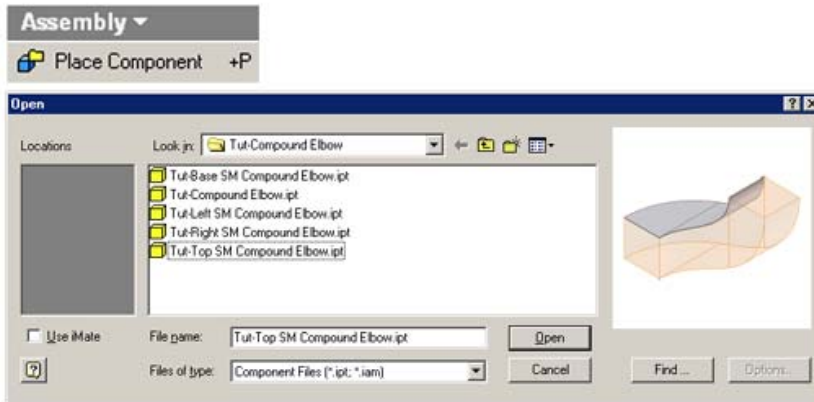
*Figure 10A-6G:
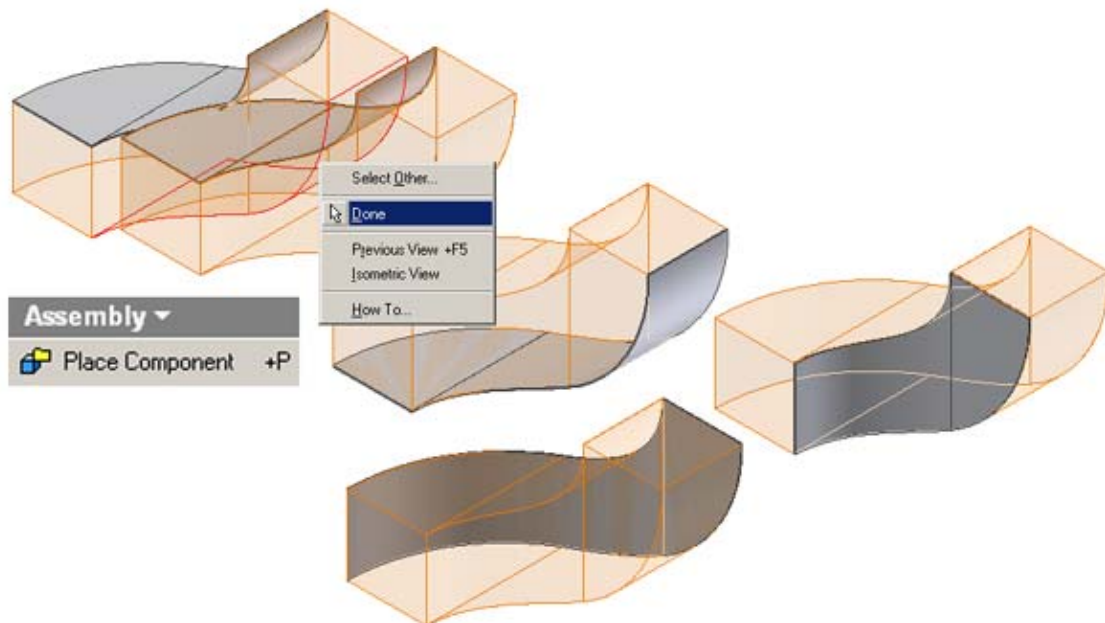Opening the Tut-Top
SM Compound
Elbow.ipt file.*

*Figure 10A-6H: Placing one copy of the model from the Tut-Right SM
Compound Elbow.ipt file.*

Next, go to the **Model** panel; click on the + buttons to open up the feature trees.
Select the **Place Constraint** tool (*Figure 10A-6J*); in the tool's dialog window, under the
Assembly tab, click the **Mate** button (the leftmost one) in the **Type** section, and the
Flush button in the **Solution** section; next, go to the **Model** panel, click on pairs of **YZ
Planes**, then **XZ Planes**, and then **XY Planes** of the assembly file's own **Origin** folder
and the *Tut-Base SM Compound Elbow.ipt1*, *Tut-Base SM Compound Elbow.ipt1*, *Tut-
Base SM Compound Elbow.ipt1*, and *Tut-Base SM Compound Elbow.ipt1* **Place
Component**'s **Origin** folder, one pair at a time, and click the **Apply** button after hearing

a loud sound (*Figure 10A-6K*). **Place Constraint** symbols appear in the **Model** panel. The compound elbow is completed; and all features are listed in the **Model** panel (*Figure 10A-6M*). Save the file.
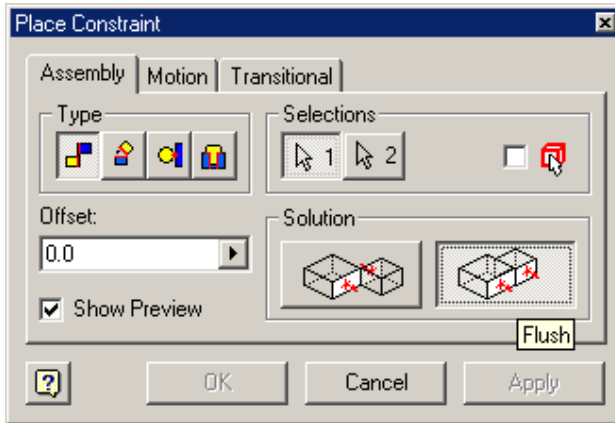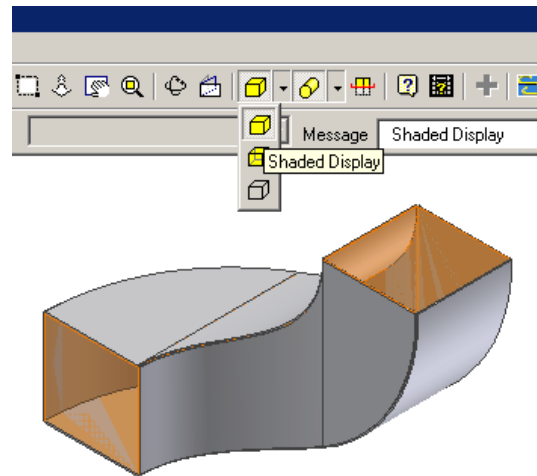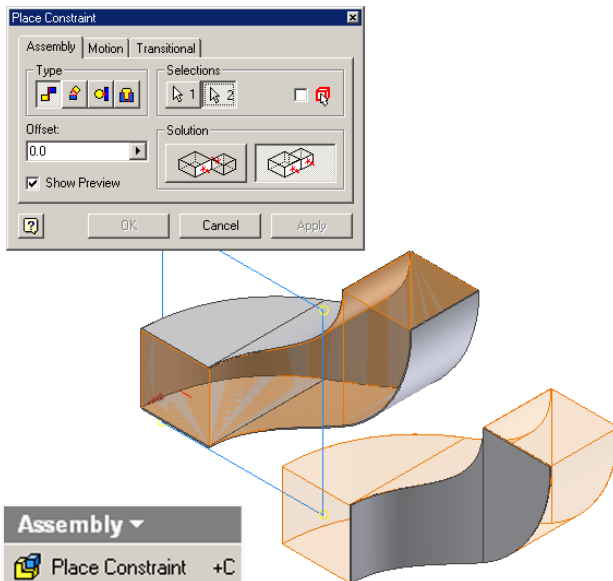

Figure 10A-6J: The Place Constraint tool.

Figure 10A-6K: Applying Mate and Flush constraints.

Figure 10A-6L: The completed compound elbow's basic parts.

Notice that deep orange display of the **Derive Part** can be seen in the assembled model; to remove this distraction, open the *Tut-Base SM Compound Elbow.ipt*, *Tut-Base SM Compound Elbow.ipt*, *Tut-Base SM Compound Elbow.ipt*, and *Tut-Base SM Compound Elbow.ipt* files; go to the **Model** panel; click on the + button on the left of the *Tut-compound Elbow.ipt* **Derived Part** feature to open up its **Work Surface** feature;

click-select the **Work Surface** feature, right-click for the shortcut mane and turn off the **Visibility** option; the orange **Work Surface** display disappears immediately (*Figure 10A-6N* through *Figure 10A-6R*). Save the files. Now, return to the assembly file, the orange **Work Surface** display disappears as well (*Figure 10A-6S*).
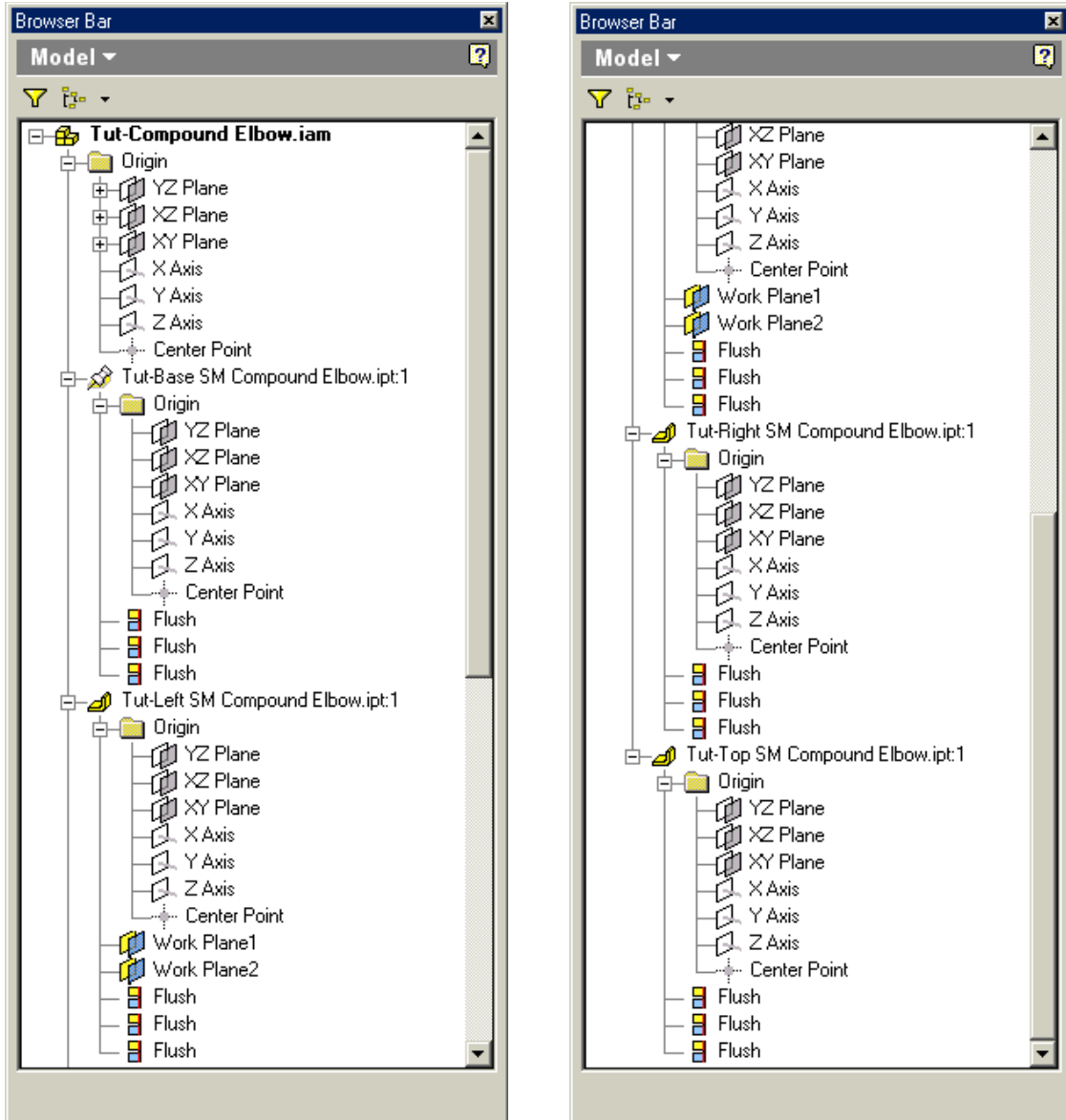

*Figure 10A-6M: All features are listed in the **Model** panel.*
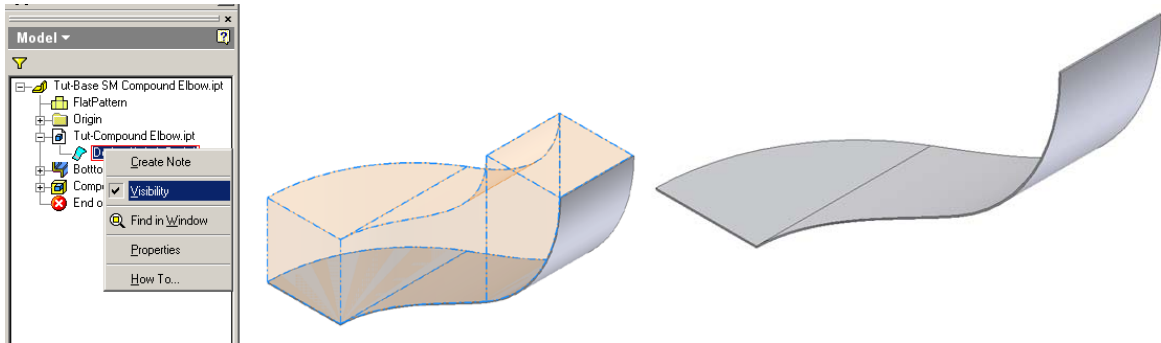
Figure 10A-6N: Turning off the Visibility of the Work Surface model in the Tut-Base SM Compound Elbow.ipt file.
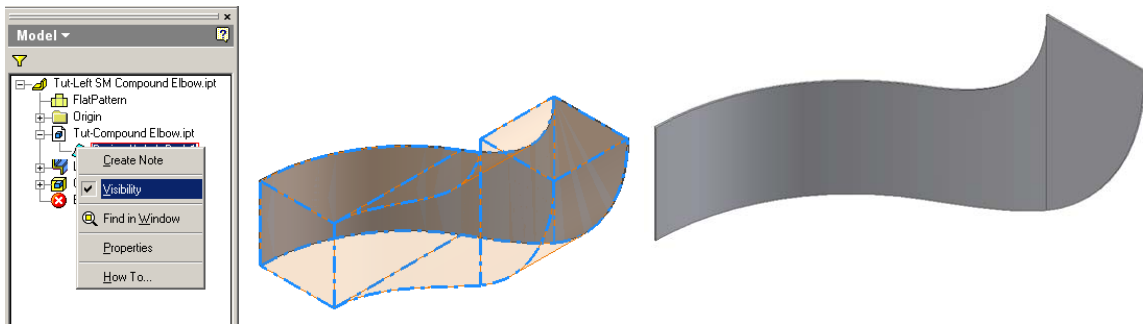

Figure 10A-6P: Turning off the Visibility of the Work Surface model in the Tut-Left SM Compound Elbow.ipt file.
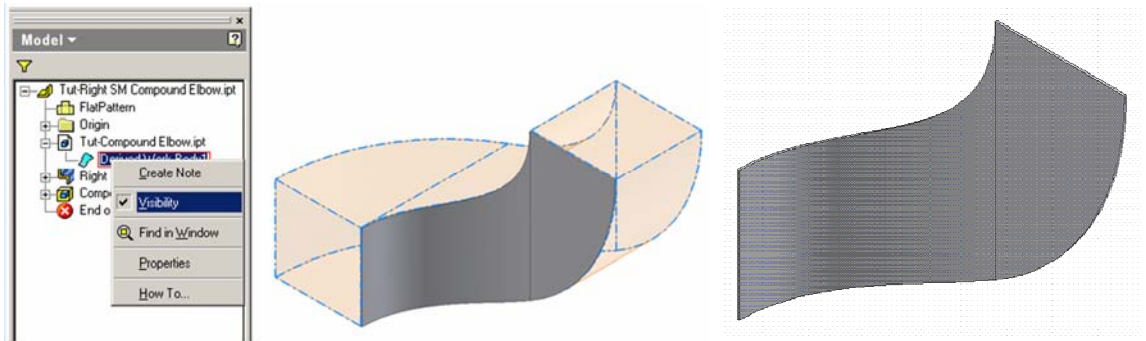

Figure 10A-6Q: Turning off the Visibility of the Work Surface model in the Tut-Right SM Compound Elbow.ipt file.
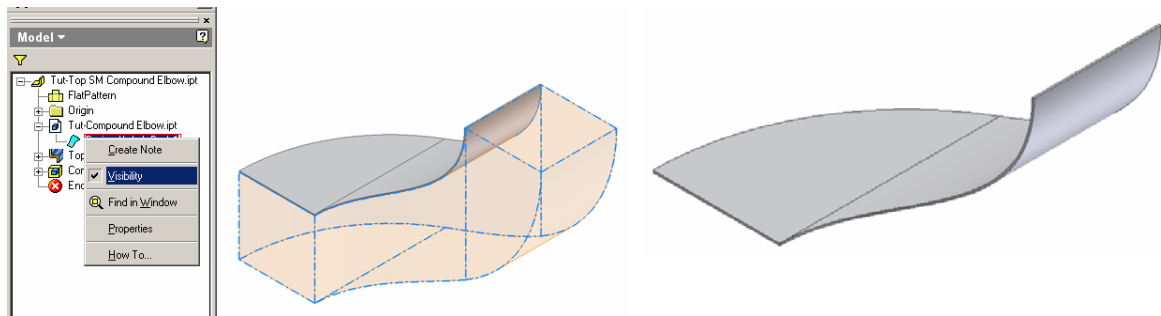
Figure 10A-6R: Turning off the Visibility of the Work Surface model in the Tut-Top SM Compound Elbow.ipt file.
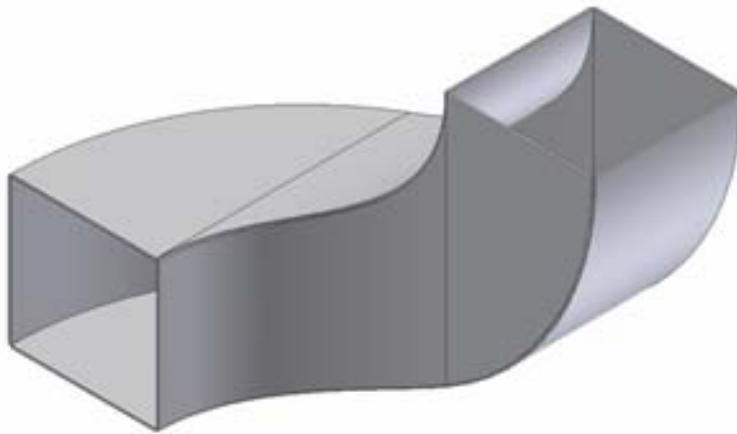

Figure 10A-6S: The orange Work Surface model disappears from the assembly file.

Congratulations! In this Module, you have leaned how to create the basic sheet-metal parts that form a compound elbow.

THE END