Module 3B:
Intersection of Three Cylinders - Creating a Y-Branch

In this *Module*, we will explore the topic of intersection and development of three cylinders by creating the basic sheet metal part of a Y-branch with three branches at 120°-angle apart in Inventor (*Figure 3B-1A*). In heating, ventilating, and air-conditioning trade, a Y-branch is used as a junction connecting three pipes. The basic methods are:

- Creating one truncated circular cylinder in an Inventor **Sheet Metal (in).ipt** format file;

- Creating an Inventor **Standard (in).iam** assembly file; using the **Place Component** tool to import one occurrence of the truncated cylinder into the assembly file; and using the **Pattern Component** tool with the **Circular** option to create the other two branches.

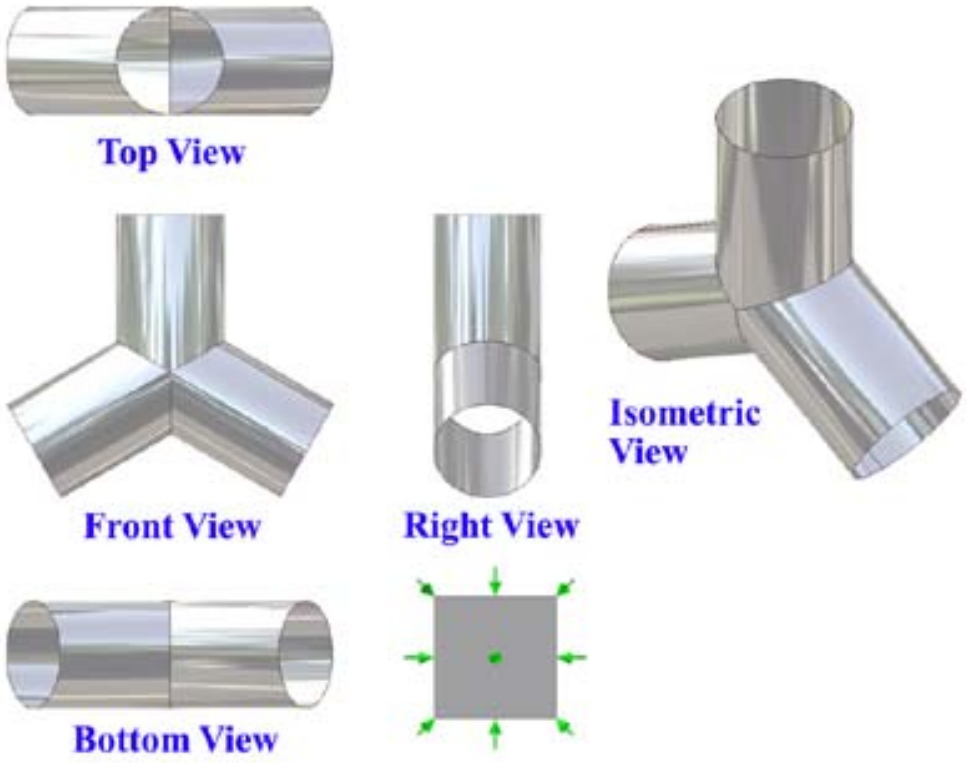

Figure 3B-1A: A Y-Branch with three component at 120° apart.

Step 1: Creating a truncated cylindrical sheet-metal part

Launch Inventor, start a new **Sheet Metal (in).ipt** file under the **English** tab. An Inventor sheet-metal file opens. "*Sketch1*" is created by default in the **Model** panel on the **XY Plane**. Rename the new sketch as *Cylinder Profile* in the **Model** panel; click-select the **Project Geometry** tool button from the **Sketch** tool panel, and then the **Center Point** feature under the **Origin** folder in the **Model** panel; the **Center Point** is projected onto the sketch to provide a snap point; next, select the **Line** tool, move the mouse closer to the projected **Center Point**, and click once when the green dot snap point indicator appears, then move the mouse upward and click one at any convenient location on the screen when the vertical symbol bar appears (this line can be called "the central axis line"); right-click for the shortcut menu and choose **Restart** option; click a point on the right-hand side of the vertical line just drawn and at about the same height as the projected **Center Point**, move the cursor upward to draw another vertical line (this line can be called "the left vertical profile line," *Figure 3B-1B*); next, use the **Offset** tool to offset the "left vertical profile line" and use the **General Dimension** tool to apply a 0.12-inch (the thickness of the intended sheet-metal material) between the "left vertical profile line" and the offset line (*Figure 3B-1C*); next, use the **Line** tool to draw two short line segments connecting the upper end points and the lower endpoints of the "left vertical profile line" and the offset line, respectively, clicking the endpoints at the appearance of the green dot snap symbols (*Figure 1C-1D*); This completes the closed rectangular cross-section profile for the sheet-metal part; next, use the **General Dimension** tool again to apply an 18-inch height dimension to the vertical lines of the rectangular profile, and a 6-inch distance between the "the central axis line" and "left vertical profile line" (the radius of the cylindrical space); click the **Return** button in the **Command Bar** to exit the sketch (*Figure 3B-1E*). Save the file as *Tut-Y Branch.ipt* inside a new folder named *Tut-Y Branch* created in the **Save** dialog window, in an appropriate directory. Save often.

Next, select the **Revolve** tool; the closed rectangular cross-section profile is automatically selected; in the tool's dialog window, select the **Axis** button and then click-select the "central axis line" in the *Cylinder Profile* sketch or the **Y Axis** from the **Model** panel; in the **Extents** section, select the **Angle** option from the drop-down menu and type 359.999 deg in the text field (*Figure 3B-1F*); green outlines for the geometry of the sheet-metal part appear; click the **OK** button to create the **Revolve** feature; and rename it *Cylinder* in the **Model** panel (*Figure 3B-1G*).

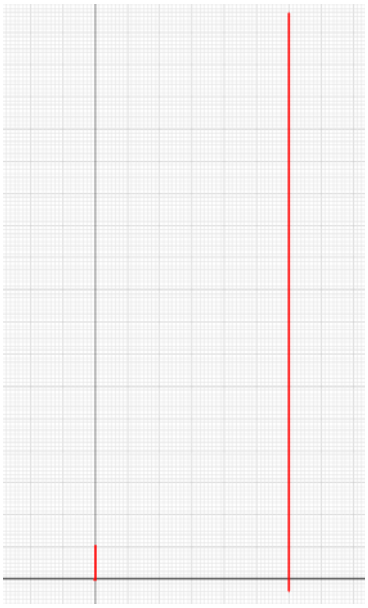

Figure 3B-1B: Projecting the Center Point and drawing two vertical lines.

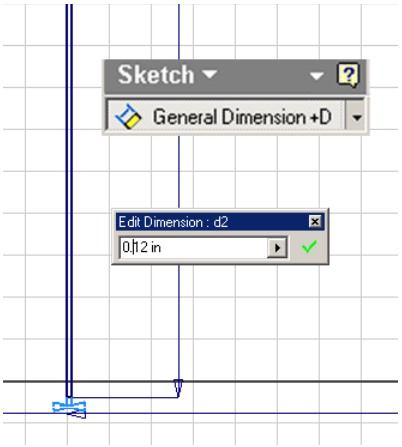

Figure 3B-1C: Applying a 0.12-inch dimension for the thickness of the sheet-metal material.

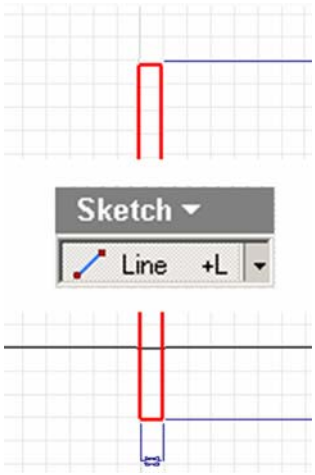

Figure 1C-1D: Completing the rectangular cross-section profile with two short line segments.

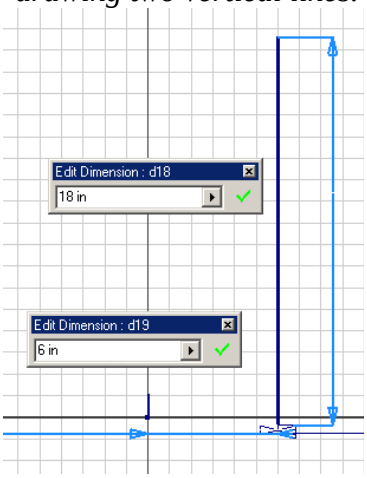

Figure 3B-1E: Applying the height and radius dimensions.

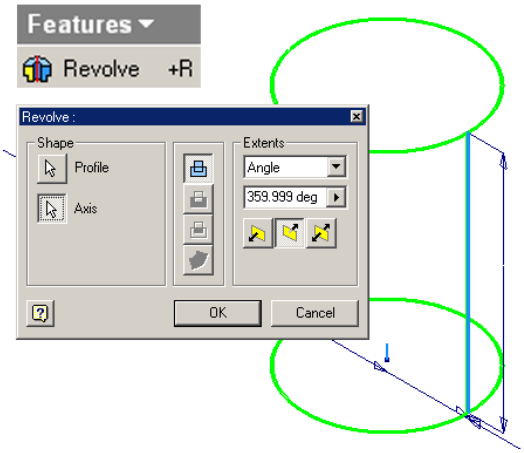

Figure 3B-1F: the Revolve tool's dialog window.

Figure 3B-1F: The Revolve Cylinder feature.

Next, select the **XY Plane** from the **Model** panel and click the **Sketch** button from the **Command Bar** tool start a new sketch; rename it *Truncation Profile* in the **Model** panel; click-select the **Project Geometry** tool and the **Center Point** in the **Model** panel to project the **Center Point** onto the sketch; use the **Line**, and **Center Point Circle** tools to draw horizontal lines, angled lines and a large circle as shown in *Figure 3B-2A*;

use the **Trim** tool to trim off segments of the angled lines beyond the circumference of the large circle as well as the unneeded portion of the circle, so as to leave a basic fan-shaped profile (*Figure 3B-2B*); next, use the **General Dimension** tool to apply 30° **Angular** dimension between the angled lines and the horizontal lines (*Figure 3B-2C*); next, select the two horizontal lines and go to the **Command Bar** to change their **Style** to **Construction** (*Figure 3B-2D*); click the **OK** button to exit the sketch.

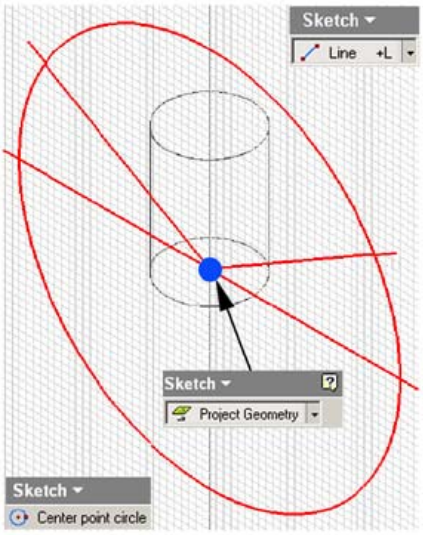

Figure 3B-2A: Starting the Truncation Profile sketcj.

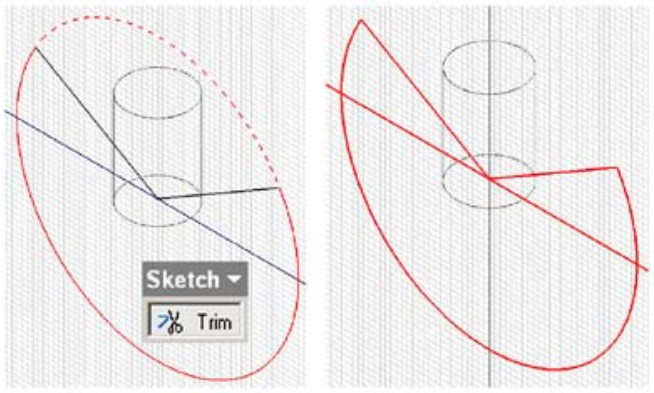

Figure 3B-2B: Trimming off unneeded line and arc segments (left) for the basic profile (right).

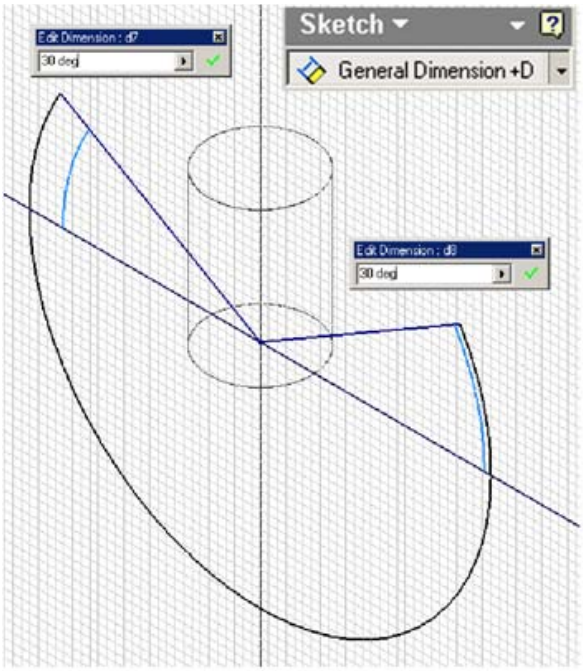

Figure 3B-2C: Applying angular dimensions.

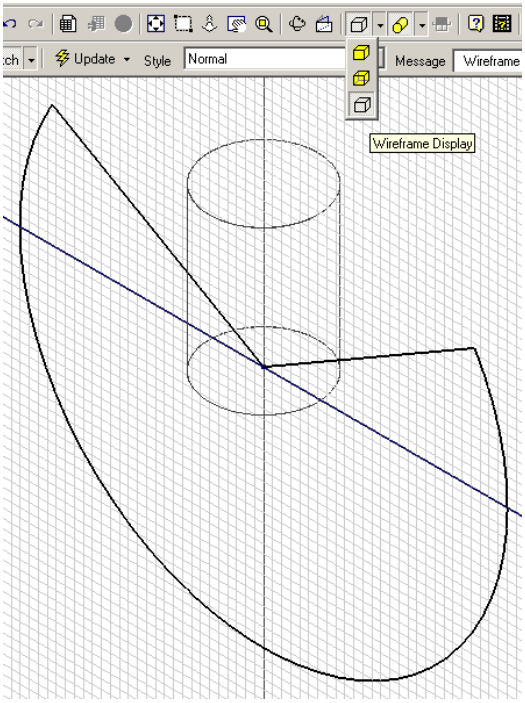

Figure 3B-2D: Changing the Style of the horizontal lines to Construction.

Next, select the **Extrude** tool; the fan-shaped profile is automatically selected, and if not, then click the **Profile** button in the tool's dialog window and click-select it on the screen; choose the **All** option and the **Midplane** option in the **Extents** section; click the **OK** button to cut the part (*Figure 3B-2E*); and rename the **Extrude** feature *Truncation Cut* in the **Model** panel (*Figure 3B-2F*).

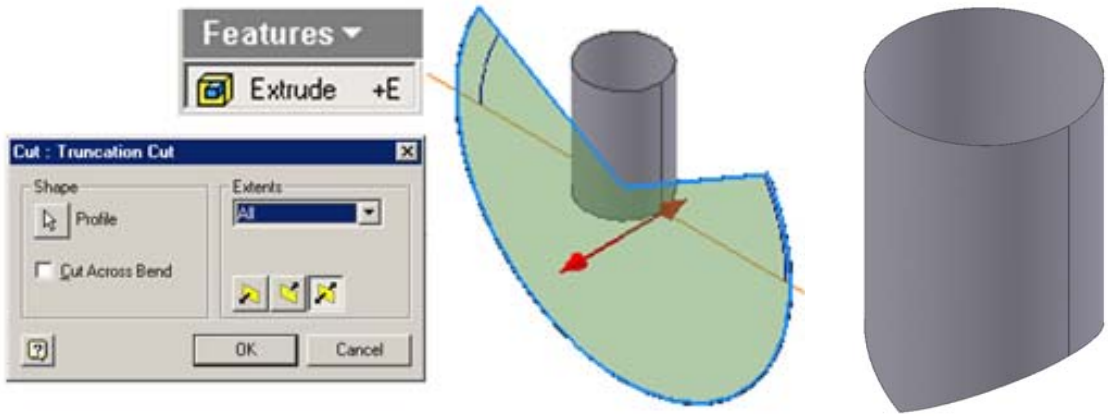

Figure 3B-2E: Truncating the cylindrical sheet-metal part with the Extrude tool (left). The completed part (right).

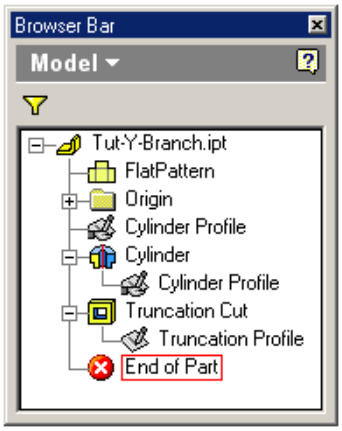

Figure 3B-2F: The features listed in the Model panel.

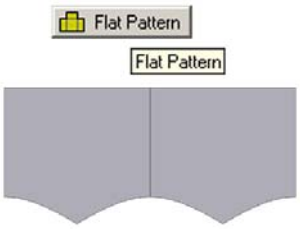

Figure 3B-2G: the Flat Pattern window.

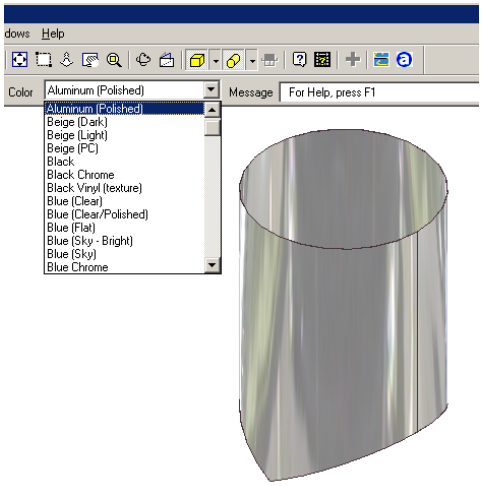

Figure 3B-2H: Applying a Color rendering to the part.

Next, click-select the outer surface of the completed part and the **Flat Pattern** tool icon from the **Sheet Metal** tool panel; the **Flat Pattern** window opens (*Figure 3B-2G*); click the **X** button at the top-right corner, on the second row, to close the **Flat Pattern** window, and to return to the 3D model window. If desired, go to the **Command Bar** to change the **Color** rendering of the part to any metal style from the **Color** drop-down menu (*Figure 3B-2H*); The one shown is **Aluminum (Polished)**. Save and close the file.

Step 2: Creating the Y-branch in an Inventor Standard (in).iam assembly file

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 (*Figure 3B-3A*). A new assembly file opens.

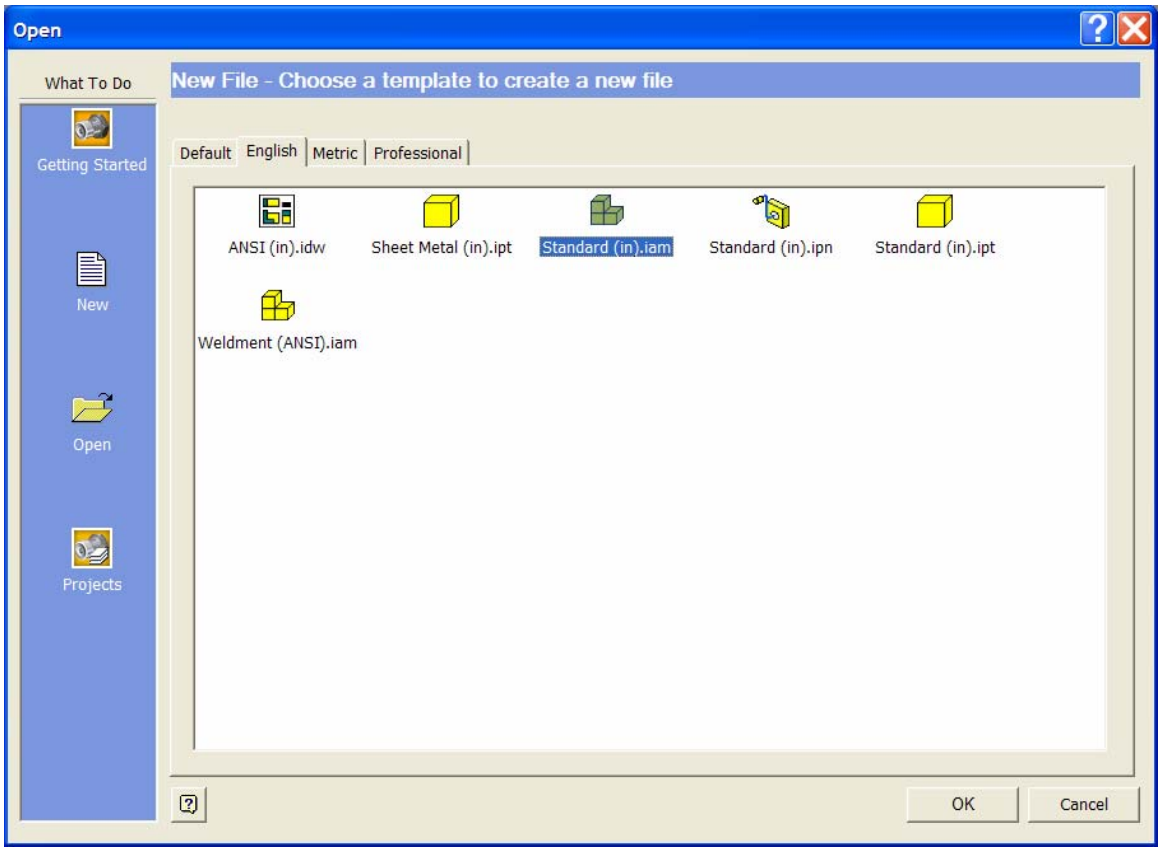

Figure 3B-3A: Starting a new Inventor Standard (in).iam assembly file.

Select the **Place Component** tool; in the **Open** dialog window, navigate to the *Tut-Y Branch* folder and select the *Tut-Y Branch.ipt* file; and click the **OK** button (*Figure 3B-3B*); the 3D folded model of the *Tut-Y Branch.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-3C* and *Figure 3B-3D*). The *Tut-Y Branch.ipt.1* **Place Component** feature appears in the **Model** panel. Save the file as *Tut-Y Branch.iam* in the same *Tut-Y Branch* folder. Save often.

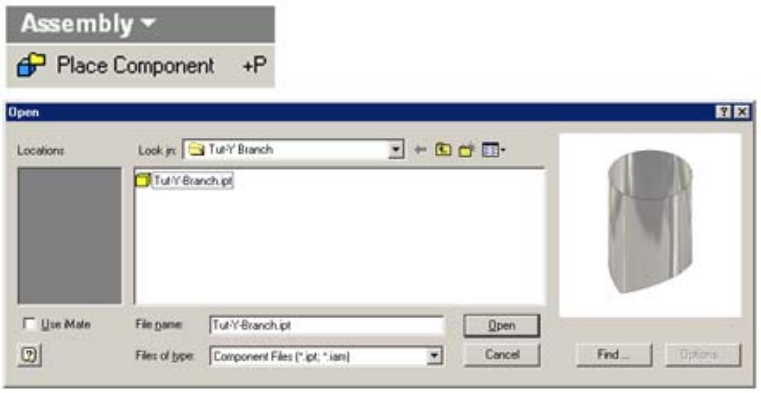

Figure 3B-3B: Importing the 3D model of the part file with the Place Component tool and selecting the part file in the Open window.

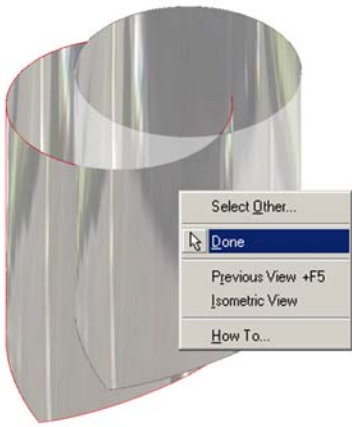

Figure 3B-3C: Placing the part in the Inventor Standard (in).iam assembly file.

Next, go to the **Model** panel; click on the + buttons to open up the feature trees. Select the **Place Constraint** tool; 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-Y Branch.ipt.1* **Place Component**'s **Origin** folder, one pair at a time, and click the **Apply** button after hearing a loud sound (*Figure 3B-3E*). **Place Constraint** symbols appear in the **Model** panel (*Figure 3B-3F*).

Figure 3B-3D: The placed 3D sheet-metal component.

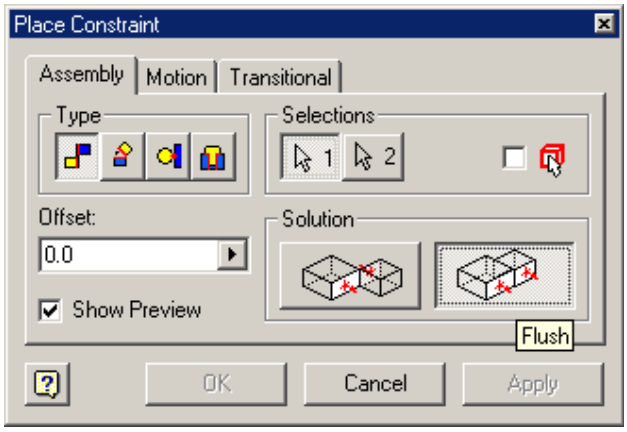

Figure 3B-3E: The Place Constarint tool's dialog window.

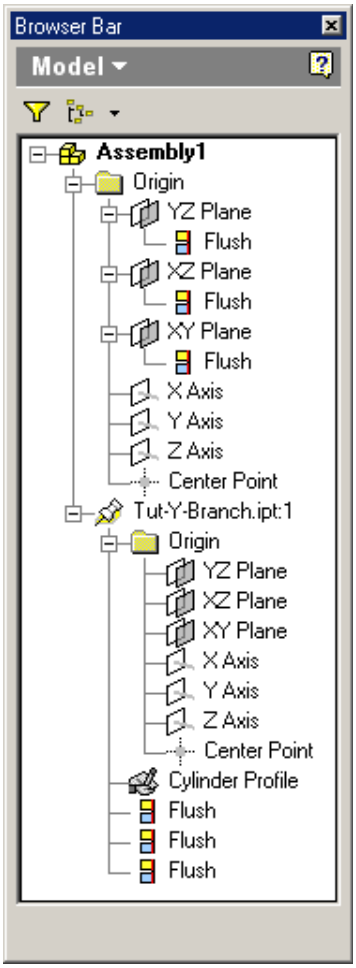

Figure 3B-3F: The Flush Place Constraint symbols in the Model panel.

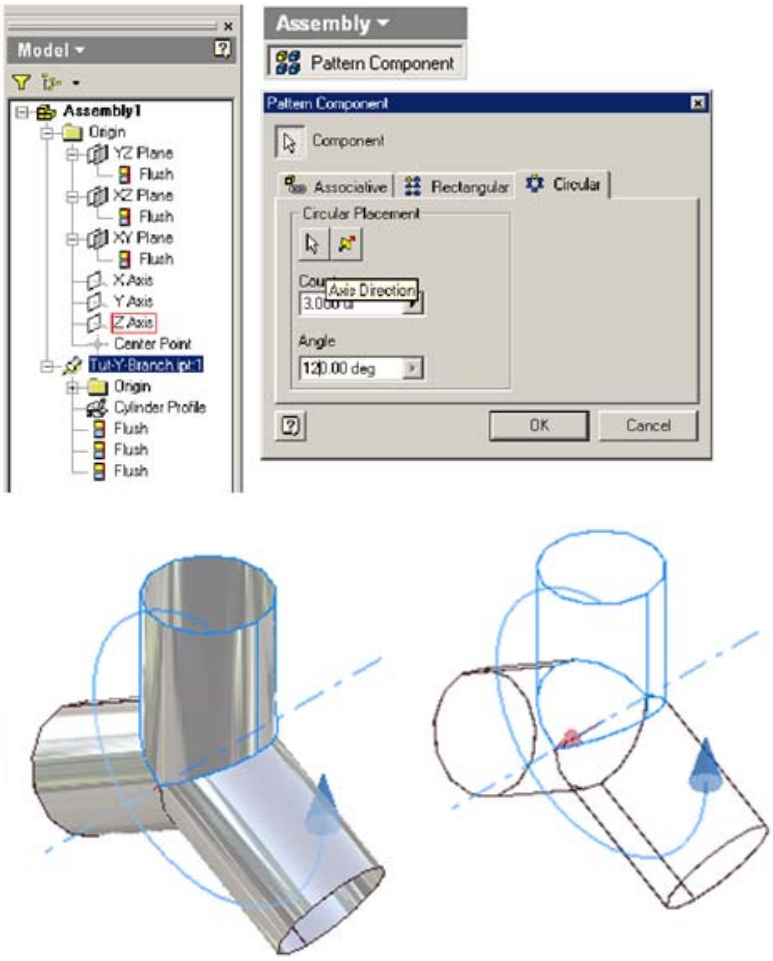

Figure 3B-3B: Completing the Y-branch with the Pattern Component tool.

Next, select the **Pattern Component** tool; in the tool's dialog window, type 3.0 ul in the **Count** text field; and type 120.00 deg in the **Angle** text field; next, select the **Circular** tab; click the **Component** arrow button, and click-select the *Tut-Y Branch.ipt.1* **Place Component** either on the screen or from the **Model** panel; then click the **Axis Direction** arrow button, then click-select the **Z Axis** feature from the **Model** panel; the blue central axis line and arrowed rotation symbol appear on the screen together with the two additional parts; click the **OK** button to create the **Pattern Component** feature (*Figure 3B-3B*); and rename it *Y Branch Component Pattern* in the **Model** panel. The Y-branch is completed; and all features are listed in the **Model** panel (*Figure 3B-3H*). Save and close the file.

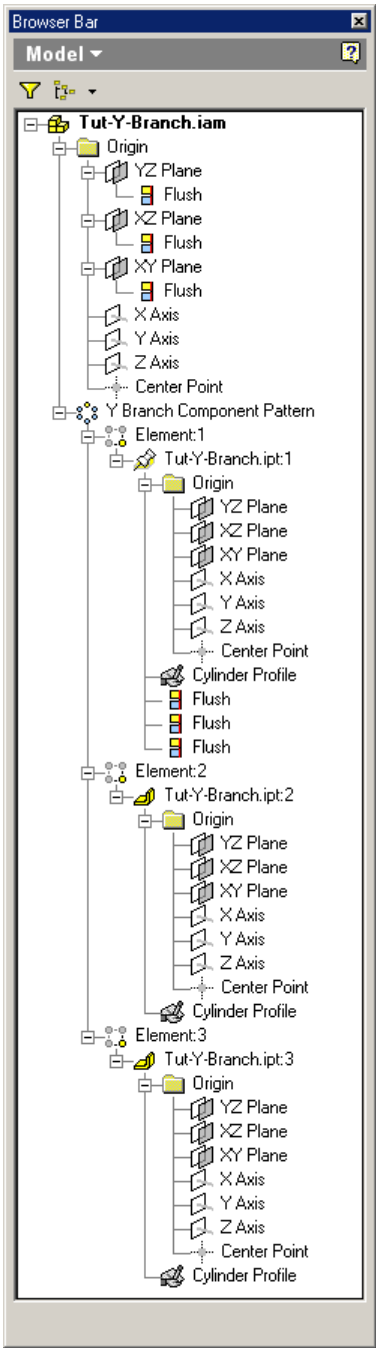

Figure 3B-3H: The features of the Y-branch listed in the Model panel.

Congratulations! You have leaned how to create a Y-branch in this Module.