

Module 1A: Parallel-Line Sheet-Metal 3D Folded Part Modeling and 2D Pattern Development-Truncated Right Cylinder

After you run through all of Inventor program's Tutorial lessons at least once, proceed to this Module. In this Module, we will learn how to build a 3D model of a sheet metal part wrapping a truncated cylinder. The basic techniques to complete this job in Inventor is to first create a 3D regular right cylindrical shell, then change it to a truncated one, and finally create a flat pattern. Step-by-step procedures are explained in the following text.

Section 1:

Creating a Sheet Metal Part Wrapping A 3D Space of A Regular Right Cylinder

Step 1: Launch Autodesk Inventor and create a sheet metal (In).ipt file

First, we will create a sheet metal file with inch as units. Launch Autodesk Inventor. Create a new sheet metal file (go to the menu **File**→**New**, or click on the **New** icon on **Standard Tool Bar** (*Figure 1A-1A and 1A-1B*). The Open window appears. Select the **English** tab, then select the **Sheet Metal (In).ipt** template, click **OK** button (*Figure 1A-2A*).

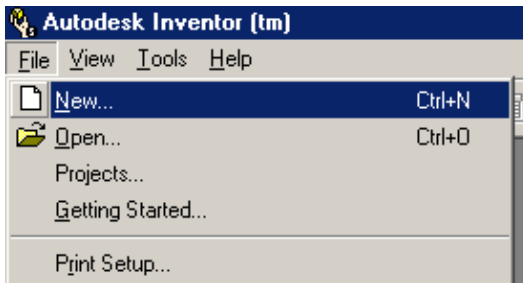


Figure 1A-1A: Creating a new file from the **File** → **New** pull-down menu.

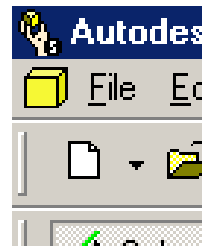


Figure 1A-1B: Creating a new file from the **Standard tool bar**.

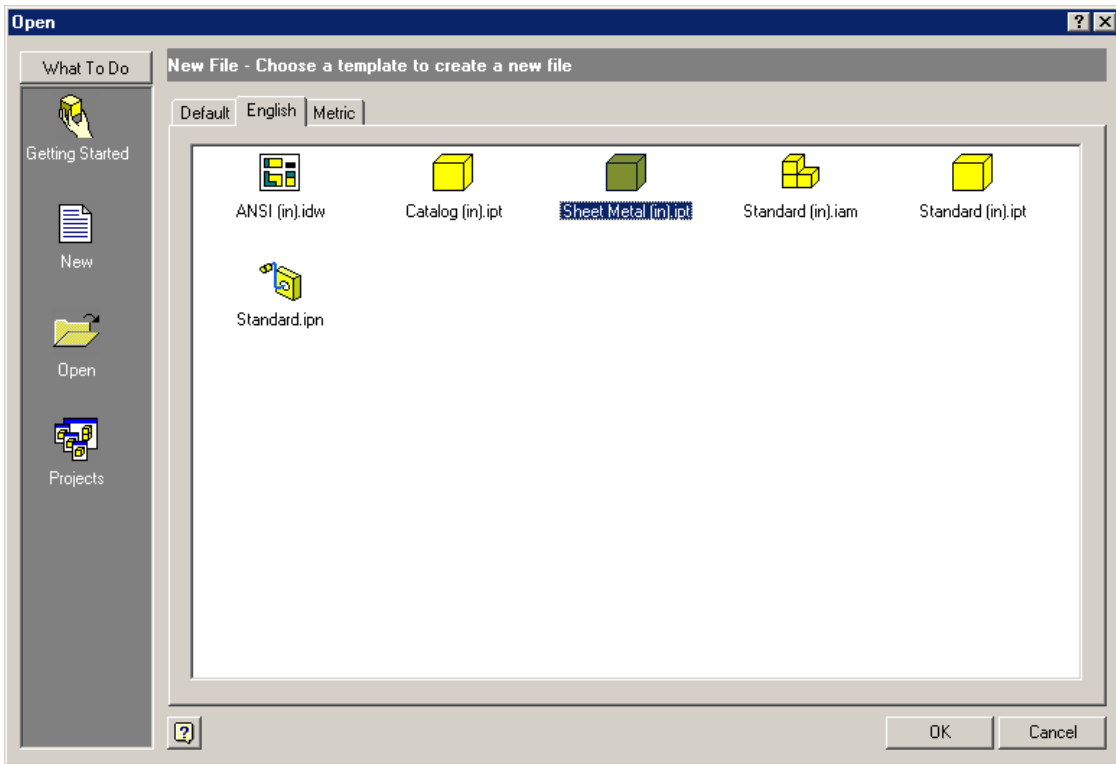


Figure 1A-2A: Open a new file window.

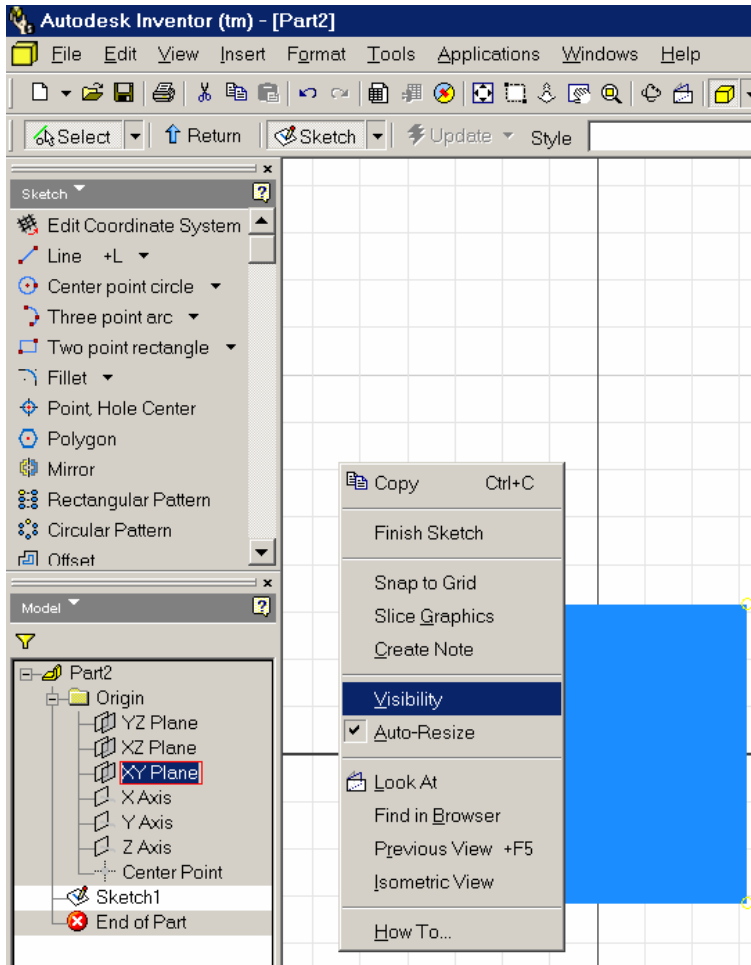
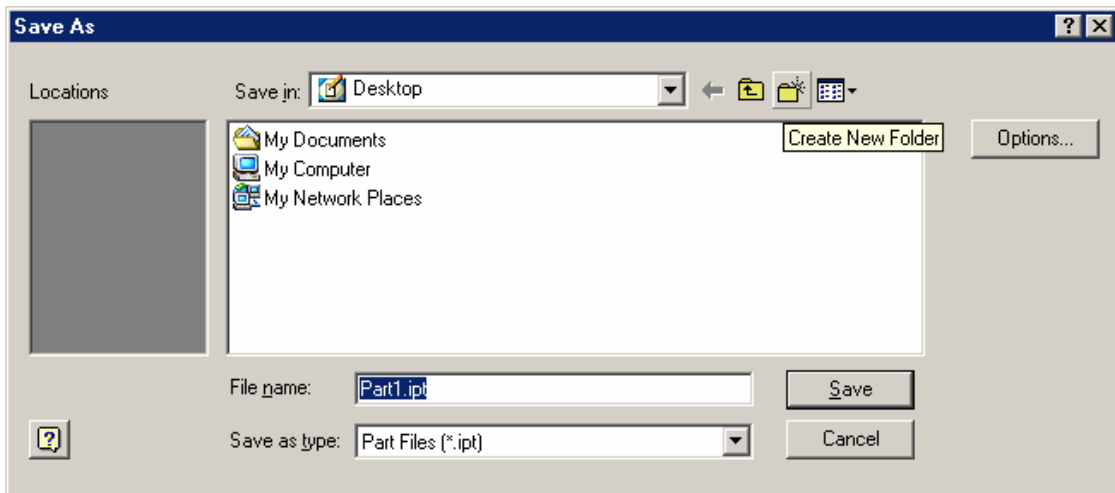


Figure 1A-2B: Drawing screen with Sketch and Model panels.

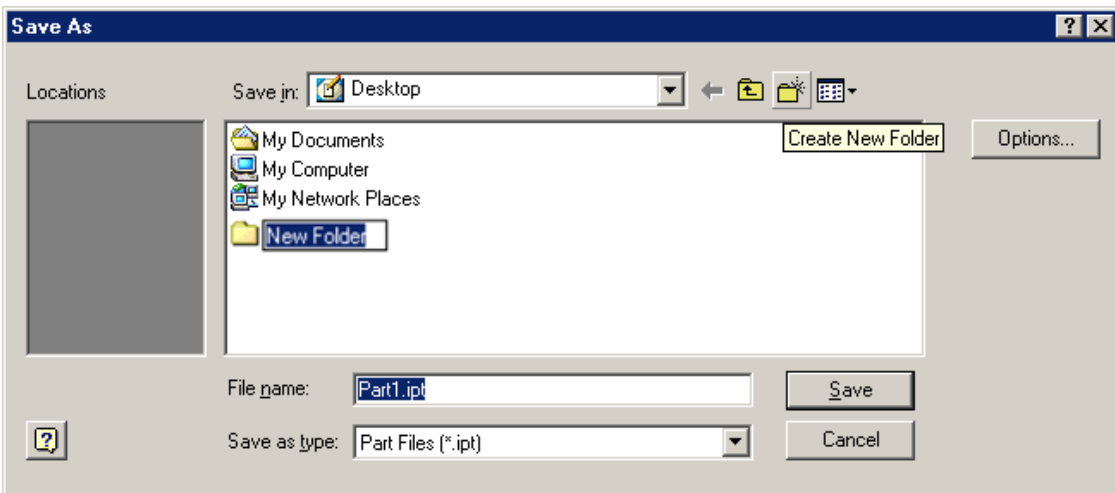
A new sheet metal file opens with two panels docked on the left side of the screen: **Sketch** (top) and **Model** (bottom). In the **Model** panel, click the cross on the left of the **Origin** folder to view the **Planes**, **Axis** and **Center Point** features. Select the **XY Plane**, then right-click to open the short cut menu and check the **Visibility** option (*Figure 1A-2B*). Now, start saving the file. Press **Ctrl** and **s** keys simultaneously to save the file as *Tut 1-Truncated Right Cylinder.ipt* with **Part File (*.ipt)** for **Save As Type** field, in a convenient directory location, inside a folder named *Tut 1-Truncated Right Cylinder* that can be instantaneously created in the **Save As** window (*Figure 1A-2C*). Remember to save your file often, at least at the end of each step.

Step 2: Create the regular right cylinder

Next, draw a rectangular wall section (10 inch high, 0.0625 inch thick) that will be used as a profile to generate a revolved right cylindrical tube (*Figure 1A-3*). Select the **Two Point Rectangle** tool in the Sketch panel (click on the tool's icon, and the name field becomes "recessed" when the tool is selected), click any point on the screen, drag the mouse and click again any point on the screen to draw a rectangle.



The Create New Folder button.



The New Folder name highlighted.

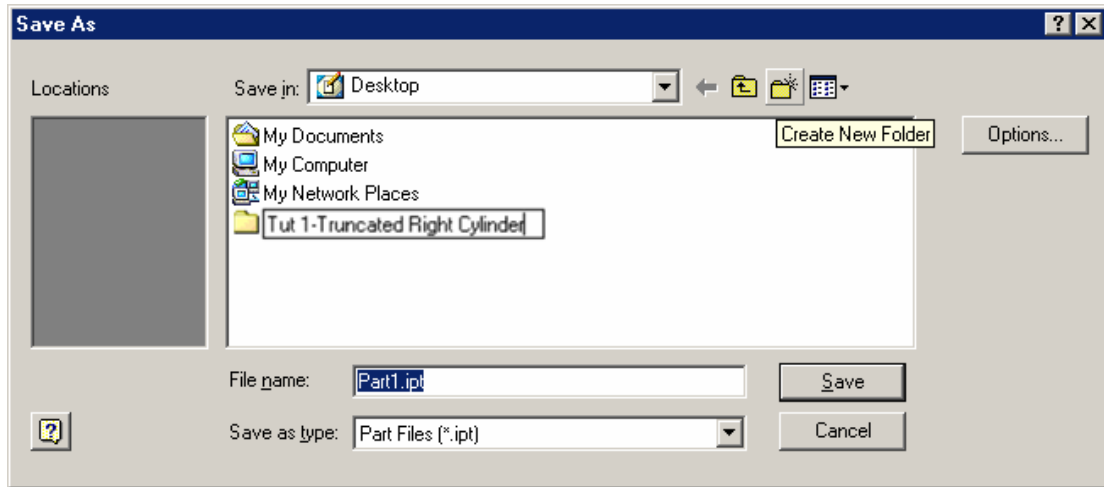
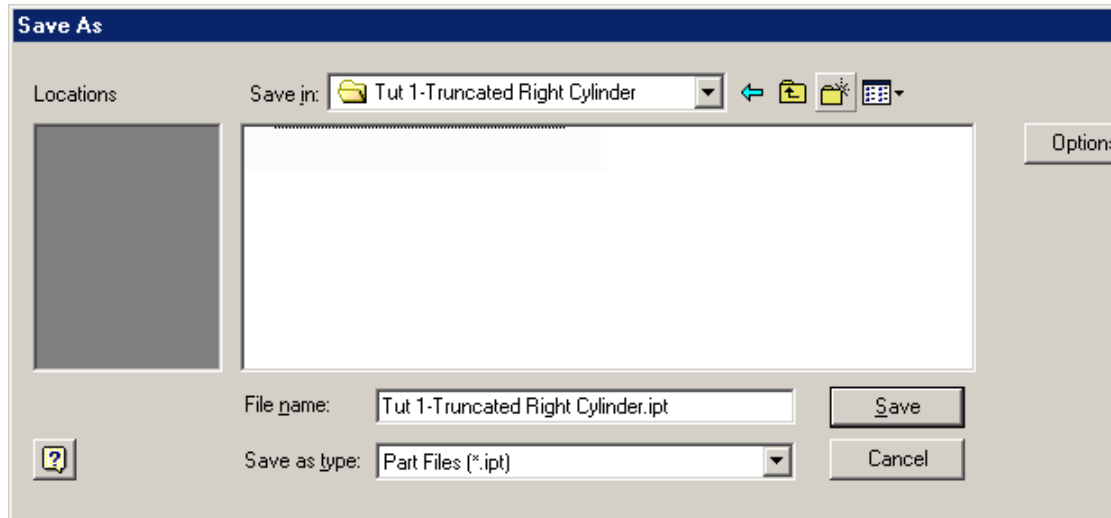


Figure 1A-2C: Creating a new folder and saving file. Typing the specific name for the new folder



Double-clicking the new folder icon to open it.

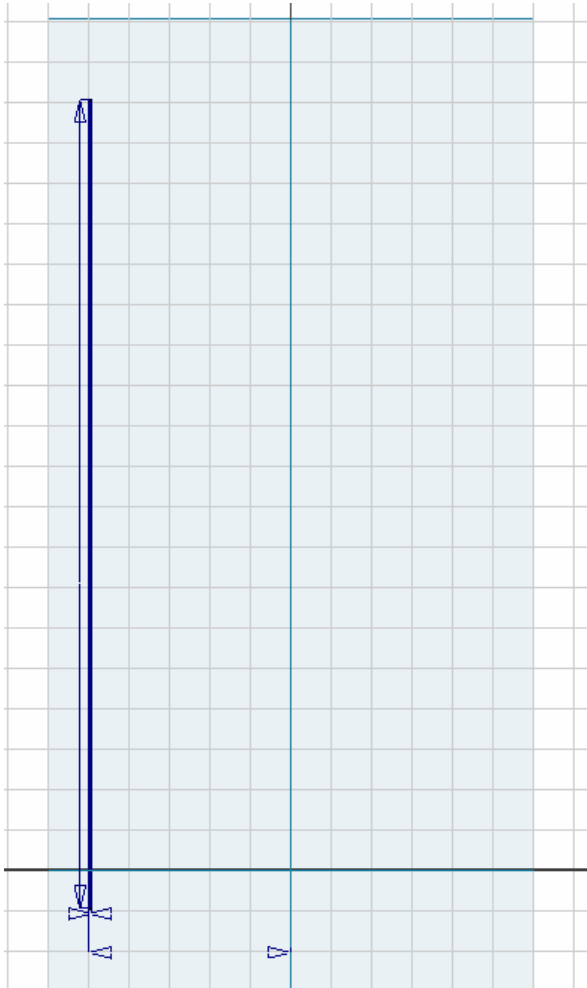


Figure 1A-3: Sectional profile for a revolved cylindrical tube (“section profile”).

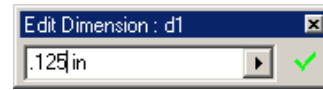
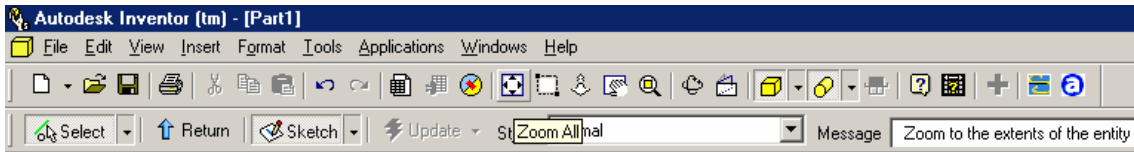


Figure 1A-4: Edit Dimension text field

Next, select the **General Dimensions** tool, click on the edge line of one side of the rectangle, drag the mouse out to create a dimension; double-click the dimension line to open the **Edit Dimension** text field (Figure 1A-4A), highlight the default dimension value and type the height value (10 inches), click green check mark. The height immediately changes. Repeat the same procedures to change the thickness of the sheet metal piece to 0.0625 inch (If for any reason, you want to change any dimension, then double-click the dimension line again to open up the dimension value field, highlight the value, retype the new value, and click on the green Check mark). If the sketch overflows out of the screen, then click on the **Zoom All** tool on the **Standard Tool Bar** to see the whole sketch (Figure 1A-4B).

Next, on the Model panel, under Origin feature folder, select the Y Axis and right-click for the short-cut menu and check Visibility option. The Y Axis turns orange; while it is still selected, click on the Style pull-down menu on the Standard Tool Bar and select the Construction option. This is the “revolution axis” that will be used to create the cylindrical tube with the Revolve tool from the Features panel.



*Figure 1A-4B: The top portion of Inventor’s standard interface: The pull-down **Menus** (top row);*

*The **Standard Tool Bar** (middle row; when you move the cursor closer to any tool icon button, the “tool tip” or a boxed name of the appear on the screen closer to the tool icon; notice that most of the tool icons in the Standard tool bar are similar to what can be found in any Window-based programs);*

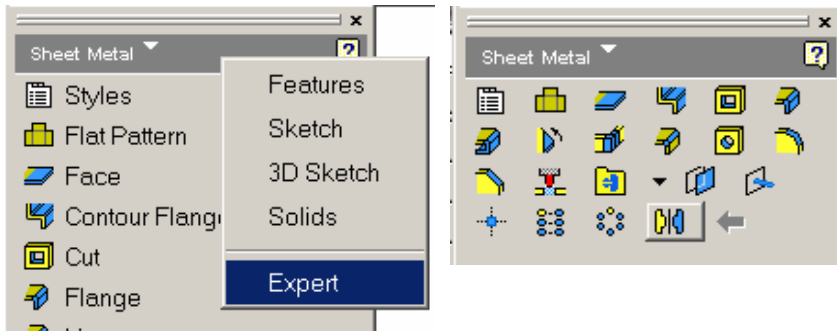
*The **Command Bar** (bottom row).*



*Please note: when the sketch (instead of 3D model) is shown, then the **Color** field in the **Command Bar** changes to **Style** field for sketch line type selection.*



Figure 1A-4B (Continued): These Panel Bars are Normally docked on the left Side of the screen. They can be torn away by click-hold on the double-bars above title stripe and drag away. To re-dock a torn away Panel Bar, click-hold the title stripe, drag and “dump” on the left side.



The default settings (left) show the names of each tool. To save space on the screen, “**Expert**” mode (Right) can be used. Click on the gray title stripe for the shortcut menu and choose “**Expert.**”

Next, click the **Return** button on the **Command Bar** to exit the **Sketch** mode and return to **Sheet Metal** mode (the **Sketch** panel automatically changes to **Sheet Metal** panel). Click on the inversed triangle on the right-hand side of the **Sheet Metal** panel name to pull down a list of options (*Figure 1A-4C*); select the **Features** option to switch to the **Features** panel. Select the **Revolve** tool. In the **Revolve** tool dialog box, click on the **Axis** square button, then select the “revolution axis” on the screen; next, select the **Profile** square button and then select the “section profile” on the screen; next, in the **Extent** section, select **Angle** and type 359.999 degree; next click the **OK** button. We choose 359.999 degrees instead of 360 degrees to leave a seam (gap) and to allow the flat pattern to be created later. The 3D model of a right cylindrical sheet metal part appears on the screen (*Figure 1A-5B*).

Step 3: Viewing the 3D model in different modes (perspective, isometric and orthographic)

To view the 3D model in isometric mode, go to **View**→**Isometric** menu. To view the 3D mode in perspective mode, go to the **Standard Tool Bar** and change the **Display** mode from **Orthographic Camera** (*Figure 1A-5B*) to **Perspective Camera** (*Figure 1A-5A*); click the inversed triangle next to the **Orthographic Camera** icon for a pull-down list and select **Perspective Camera**; these options determine how the 3D models are displayed on the screen. In typical engineering drafting courses today, the skills of drawing perspectives are not emphasized due to difficulty in creating them in 2D. Most of 3D modelers today include the tools to automatically generating perspective view out of 3D models, either for a screen shot (created by pressing the **Print Scrn SysRg** button on the computer’s keyboard and use the **Paste** tool to paste it in Photoshop for further manipulation), or for an exported rendered picture file (in tiff, gif or other formats); which can be imported into CAD drawings for client presentation. In mechanical engineering design, isometric views are generally used except for presentation of large system (aircraft, machinery, etc.). Perspective views are generally used for architectural and civil engineering design presentations.

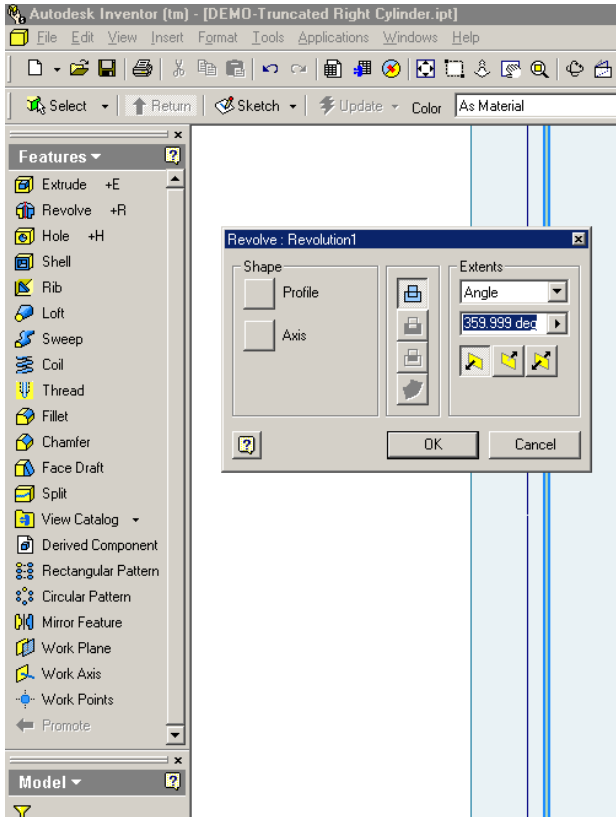


Figure 1A-4C: Revolve tool dialog window

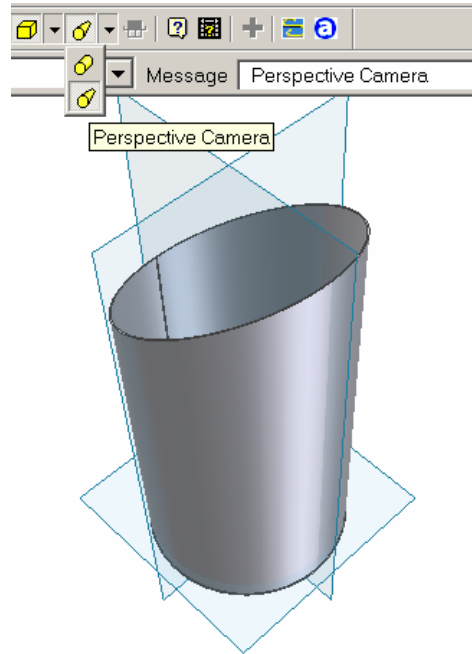


Figure 1A-5A: Perspective Camera.

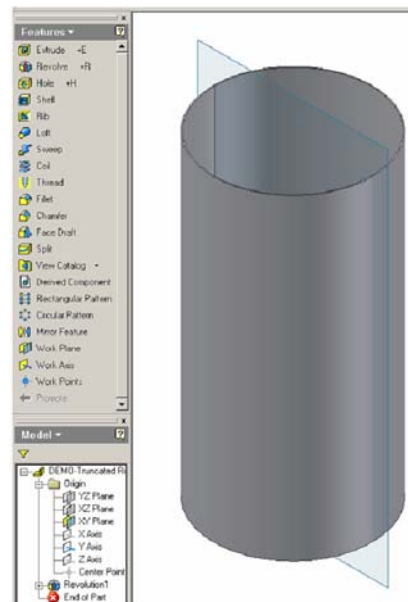


Figure 1A-5B: Orthographic Camera (Isometric view)

To switch among the six different orthographic views (top and bottom, front and back or “rear,” right and left side), the four standard isometric views (rotated around the X-, Y-, or Z-Axis), or isometric views at any “free” viewing angles during the design process, use the **Rotate** tool. Select the **Rotate** tool from the **Command Bar**; a default “**Free Rotate**” symbol (a large circle with four handles) appears (*Figure 1A-6A*); click outside of the circle and drag to rotate the view of the 3D model around an imaginary plane parallel to the screen (*Figure 1A-6B*); click any handle and drag to rotate the 3D model horizontally or vertically (*Figure 1A-C*); click inside the circle and drag to “free rotate” the 3D model around any coordinate axis in the digital space (*Figure 1A-D*). To switch from the default “free rotate” mode to orthographic and standard isometric views, right-click for a short-cut menu and select **Common Views**; the circular **Rotate** symbol disappears and the grayish “cube” with green arrows appears (when the mouse is moved closer to the arrow, it turns red). Click any green arrow on the corner of the “cube” to generate any of the standard isometric view (*Figure 1A-6E*); click any of the green arrows on the surface of the “cube” to generate any of the orthographic view, by 90 degrees (*Figure 1A-6G* and *Figure 1A-6H*); click the green arrows at the center of the “cube” to rotate the 3D model by 180 degrees (*Figure 1A-6I*). To exit the **Rotate** tool, right-click for the shortcut menu and select **Done** (*Figure 1A-6F*).

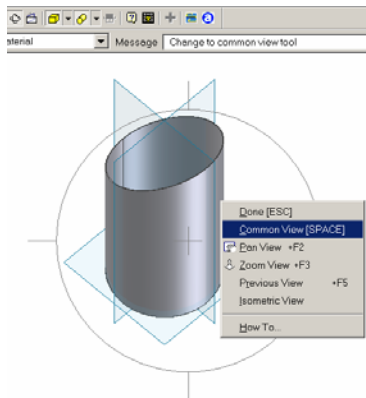


Figure 1A-6A: From Free Rotate to Common View.

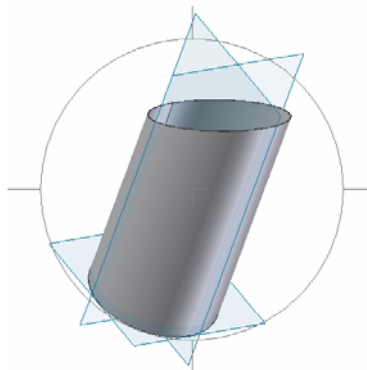


Figure 1A-6B: Outside of the “circle.”

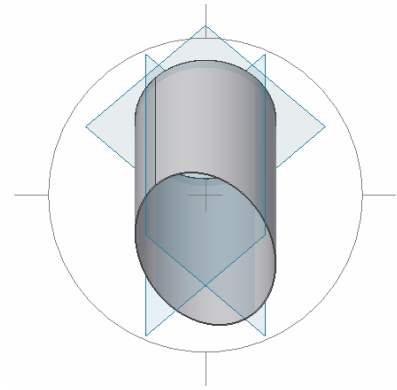


Figure 1A-6C: On the “handle.”

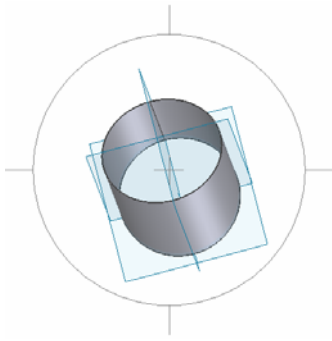


Figure 1A-6D: Inside of the "circle."

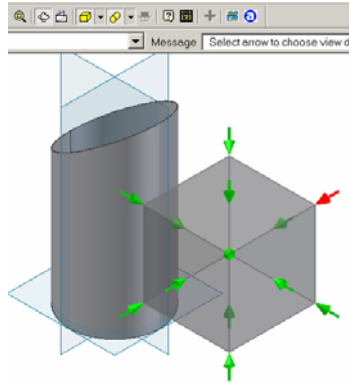


Figure 1A-6E: Rotate to another isometric view

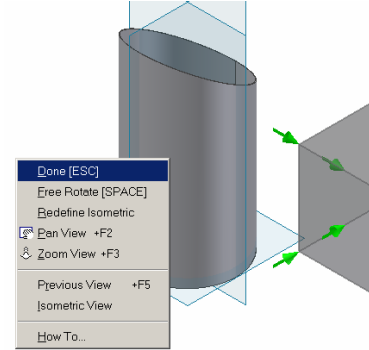


Figure 1A-6F: Exiting the Rotate tool

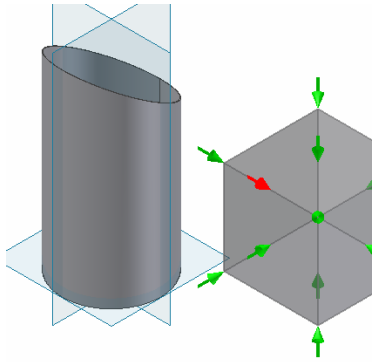


Figure 1A-6G: Rotating to an orthographic view

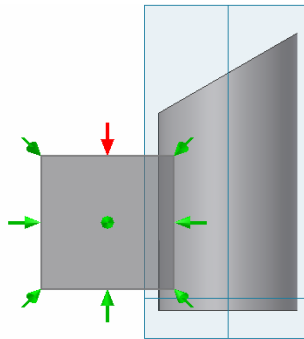


Figure 1A-6H: Rotating the model by 90 degrees

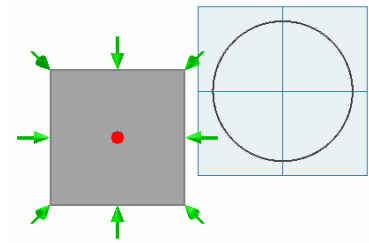


Figure 1A-6I: Rotating the model by 180 degree.

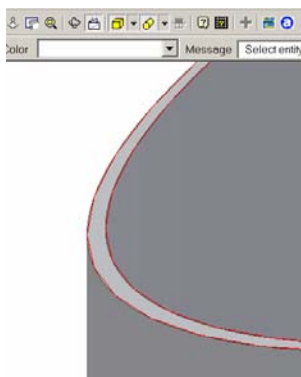


Figure 1A-6J: The "Look At" tool for a slanted orthographic true-shape view of a surface.

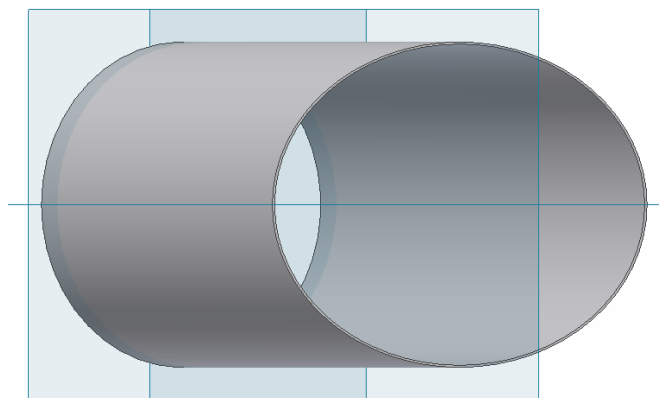


Figure 1A-6K: Create an auxiliary view for a slanted surface.

To view the true-shape (abbreviated as TS in descriptive geometry terms) of any slanted or oblique surface, which is drawn in primary and/or secondary auxiliary view(s) in 2D drawings, use the **Look At** tool. First of all, use the **Zoom Window** and/or **Rotate** tools to move the model so that the slanted surface is large enough to click on. Select the **Look At** tool from the **Command Bar**, move the mouse closer to the slanted surface; the edges of the surface turn red (*Figure 1A-6J*); click on the surface, it turns blue and the 3D model rotates to a position that shows the true-shape view of the slanted surface (*Figure 1A-6K*). Click on the **Zoom All** tool icon on the **Standard Tool Bar** to view the entire 3D model on the screen.

The Sheet Metal Part wrapping a 3D space of a regular right cylinder is now completed.

Section 2: Truncating The Sheet Metal Part Wrapping The 3D Space of A Regular Right Cylinder

Next, we will create a sketch on the **XY Plane** and use it to truncate the cylinder. Select the **XY Plane**; click the **Look At** and **Zoom All** tool buttons (on the **Standard Tool Bar**), to return to the orthographic screen. While the **XY Plane** is selected (or select it from the **Model** panel), click the **Sketch** button. Move the mouse cursor closer to the top horizontal edge of the cylinder, the edge turns red (*Figure 1A-7A*); click-select it and it turns blue immediately. Then click-select the **Project Geometry** tool in the **Sketch** panel. The edge is projected onto the **XY Plane**. Check the line **Style** field on the **Command Bar** to make sure that the projected line is in **Normal** line style (in black color); or change it to **Normal line Style** if needed. Next, select the **Line** tool, move the cursor closer to the right end of the top edge line; a green dot appears, indicating an endpoint snap (*Figure 1A-7B*). Click on the green snap point, and move the mouse to the left at an angle; click to complete an angled line.

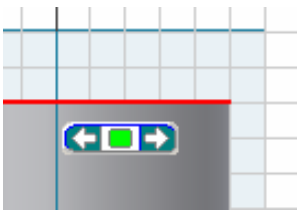


Figure 1A-7A: Move the cursor closer to the top edge.

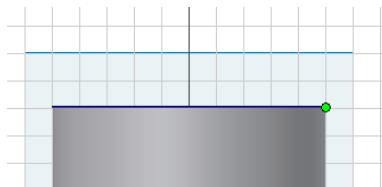


Figure 1A-7B: The endpoint snap point (the green dot)

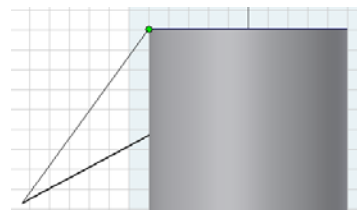


Figure 1A-7C: Picking up the endpoint snap for tracking.

Move the mouse again towards the left endpoint of the top edge line; when the green snap mark appears (*Figure 1A-7C*), move the cursor away leftwards horizontally; a snap tracking symbol appears (*Figure 1A-7D*). Click at any convenient location; draw another line segment connecting the endpoint of this line and the left endpoint of the top edge line; and then right-click and select **Done** to complete the sketch (*Figure 1A-7E*). Rename the Sketch feature *Truncating Profile Sketch* in the Model panel.

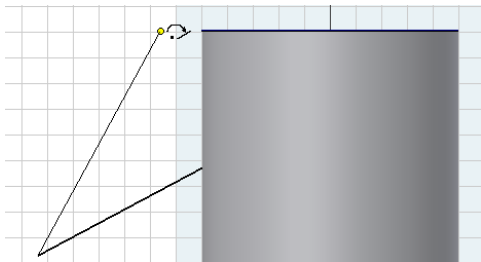


Figure 1A-7D: Tracking.

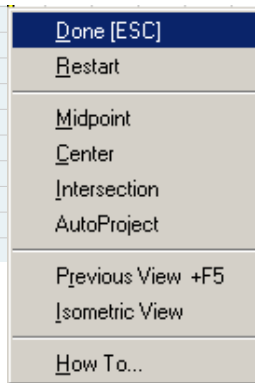


Figure 1A-7E: Shortcut menu

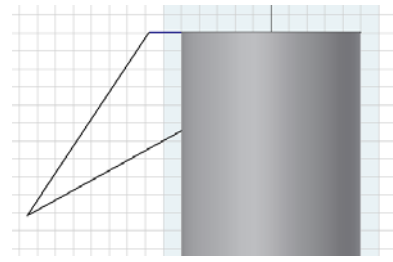


Figure 1A-7F: Rough sectional profile for truncating the regular right cylinder, before applying dimensions.

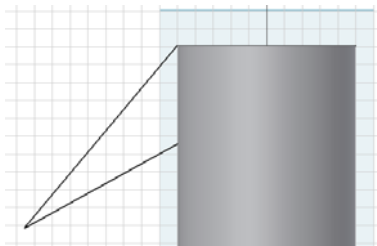


Figure 1A-7G: Avoid this profile.

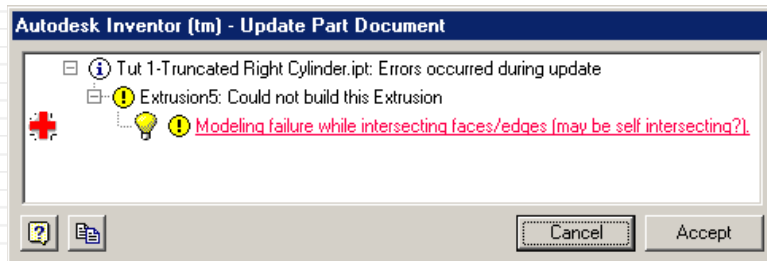


Figure 1A-7H: Problem of “Modeling failure while intersecting faces/edges (may be self intersecting?)”

Avoid drawing a sketch as shown in *Figure 1A-7G*, where all endpoints of the profile coincide with the corner points of the 3D model, because when it comes to cutting, problem of “Modeling failure while intersecting faces/edges (may be self intersecting?)” might occur (*Figure 1A-7H*). Click-select the top horizontal edge line and then the **Fix** tool (on the **Sketch** panel) to fix the horizontal line segment; and use the **General Dimensions** tool to apply a 30-degree angle between the angled line and the fixed horizontal line (*Figure 1A-7I*). The fixed line will stay in place while the unfixed slanted line will move to change its angle.

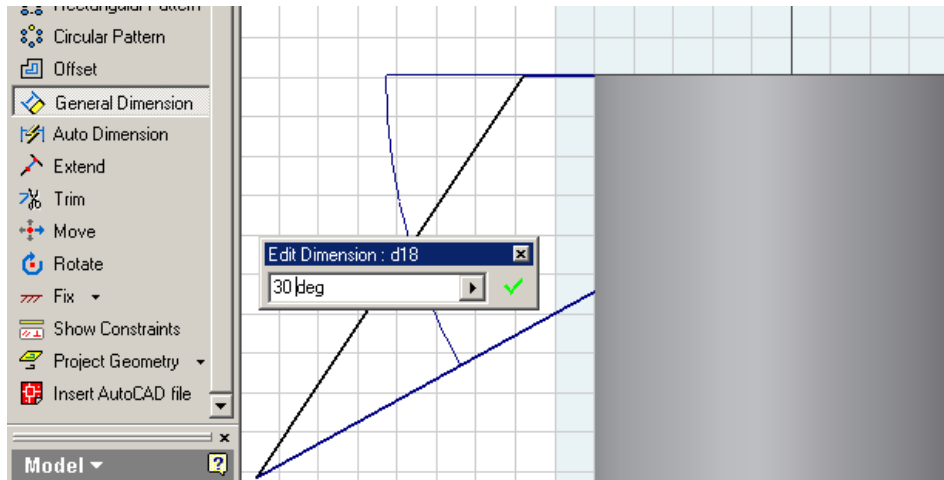


Figure 1A-7I: Figure 1A-7F: Completed sectional Profile for truncating the regular right cylinder.

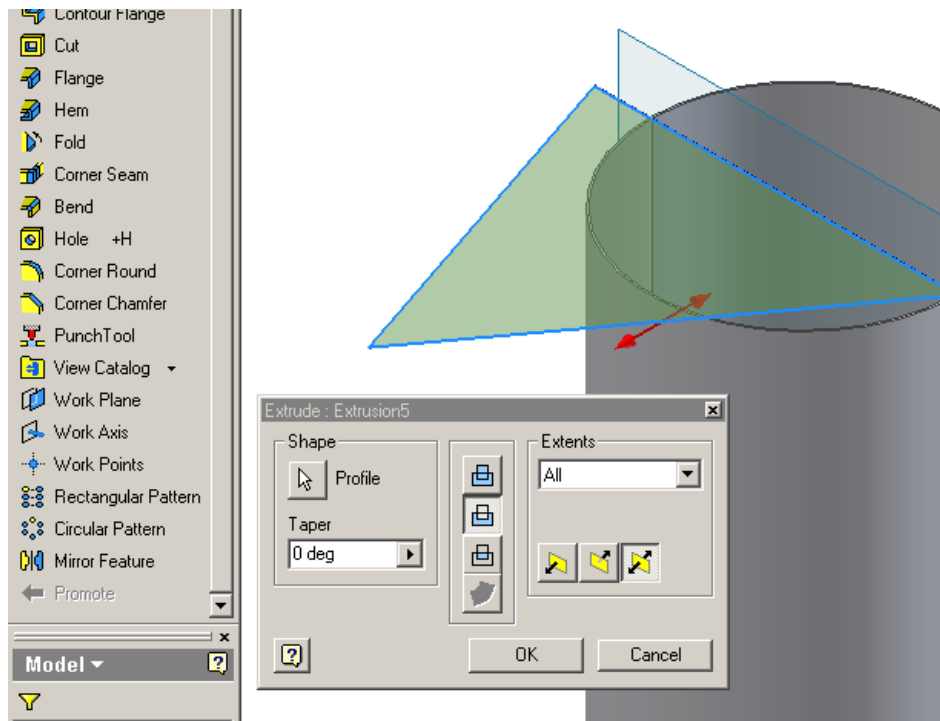


Figure 1A-7J: The Extrusion tool dialog window.

Click the **Return** button to go back to the **Features** panel. Go to **View→Isometrics** for a better view during the truncating operation. Select the **Extrusion**

tool; and in the Extrusion tool dialog window, select **Cut** for extrusion type, **All** for **Extent**, and **Midplane** for **Direction**. Click the **OK** button (*Figure 1A-7J*). The cylinder is now truncated. Now, tell the Inventor program which surface is targeted for flat pattern by clicking on the surface of the cylinder (*Figure 1A-7K*); and in the **Sheet Metal** panel, click-select the **Flat Pattern** tool. The **Flat Pattern** window screen appears (*Figure 1A-7L*). Click the **X Close** button at the upper-right corner of the screen (the one of the file, not the topmost one for the Inventor program), to return to the 3D folded model screen. Notice that a FlatPattern icon appears in the Model panel (*Figure 1A-8A*).

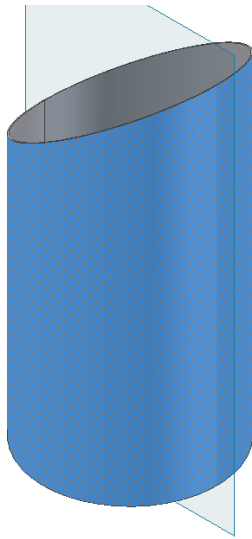


Figure 1A-7K: Select the targeted surface for Flat Pattern projection.

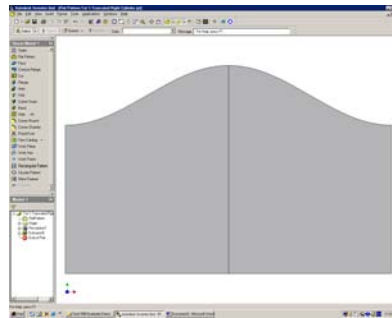


Figure 1A-7L: The Flat Pattern window.

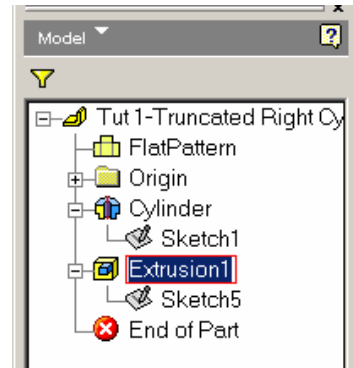


Figure 1A-8A: Select the name of the feature.

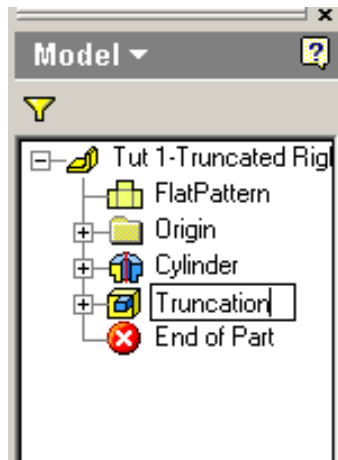


Figure 1A-8C: Change the name of the feature from generic to specific.

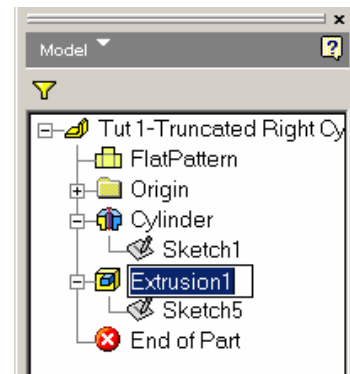


Figure 1A-8B: Select the text.

It is a good design practice to change the generic names (such as *Extrusion1*, etc.) in the **Model** panel to more meaningful names for convenience at feature identification if further editing is needed later. To change the name of a 3D feature (or sketch), click on the name to select it (the name is highlighted and surrounded by a red box (*Figure 1A-8A*)); then pause for 2 seconds; then click again to highlight the text alone, the red box disappears (*Figure 1A-8B*); and type a new name (*Figure 1A-8C*). Now, save the file and close it.

Section 3: Recycling an existing 3D model file to create other 3D model files -creating the top piece of the truncated cylinder

The true-shape of the top piece of the truncated cylinder is an ellipse. In order to make it easy and to assure accuracy, we will employ a special “workaround” technique call “recycling the existing 3D model file,” which is generally speaking not mentioned in standard or generic textbooks or in the official user manual accompanying the Inventor program. This technique basically consists of saving the existing 3D files (“file A”) as other files under different names (“file B,” “file C,” etc.), through the **File→Save A Copy As** menu, or by the Window-standard **Copy** and **Paste** utility; of opening the new files to delete irrelevant features and to keep relevant sketches or features; and of using relevant features or sketches to build new features needed for the new part model files. We will proceed as explained in the following steps.

Locate the *Tut 1-Truncated Right Cylinder.ipt* just created, click-select the file, then right-click to open the short-cut menu and select **Copy** (*Figure 1A-9A*); then click at any convenient location inside the *Tut 1-Truncated Right Cylinder* folder, and right-click for the shortcut menu and select **Paste**. A new file named *Copy of Tut 1-Truncated Right Cylinder.ipt* is created. Change the names to *Tut 1-Top.ipt*.

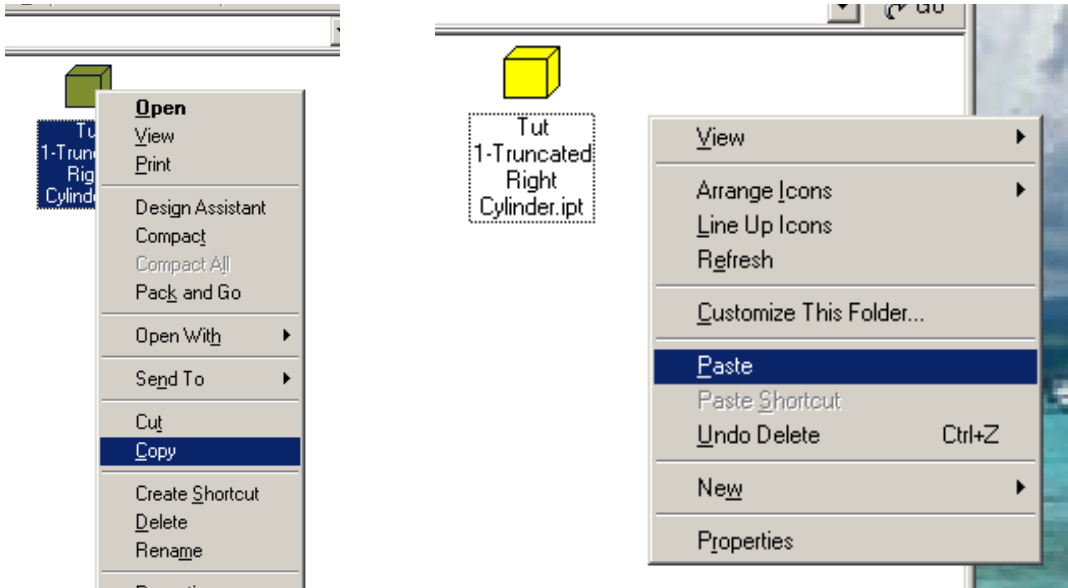


Figure 1A-9B: Duplicate a file.

Figure 1A-9A: Start to duplicate a file.

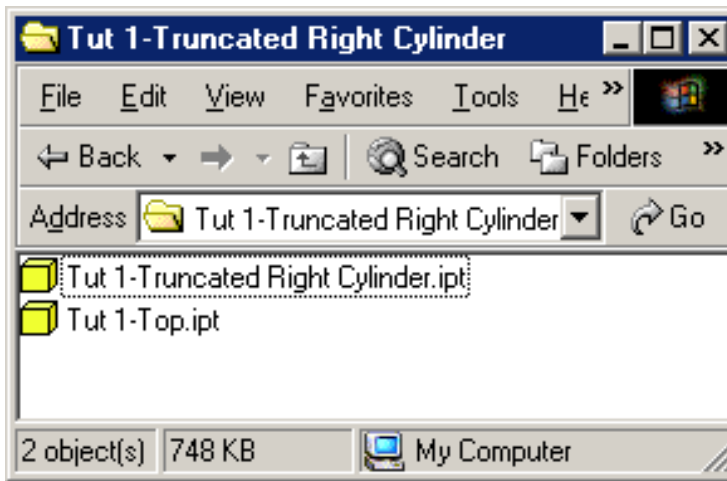


Figure 1A-9C: Change the name of the duplicated file after it is created.

Open the file named *Tut 1-Top.ipt*. Use the **Zoom Window** tool to zoom in the top surface of the cylindrical tube by dragging a selection window (Figure 1A-10A). Click-select the **Work Plane** tool from **Features** panel (Figure 1A-10B). Move the cursor closer to the surface, red outline appears (Figure 1A-10C); click-select the truncated surface (it immediately turns blue) and drag upward; the **Offset** ext field appears with a highlighted value (Figure 1A-10D); type 0 over the highlighted value and

click on the green check mark (*Figure 1A-10E*). A work plane appears on the slanted surface in orange. Rename it *Top piece Work Plan* in the **Model** panel. Click-select the *Top piece Work Plan* and press the **Sketch** button to start a new sketch (*Figure 1A-10F*). Click-select the outer edge of the truncated surface; click-select the **Project Geometry** tool. The outer edge is projected onto the *Top piece Work Plan*; click-select it and go to **Style** text filed on the **Command Bar** to change its **Style** to **Normal**. Save the file.

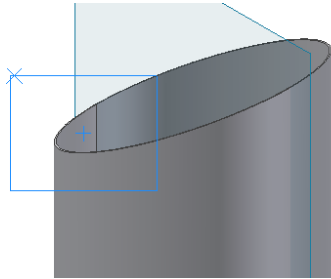


Figure 1A-10A: The Zoom Window tool.

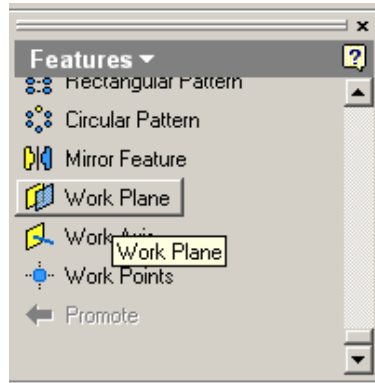


Figure 1A-10B: Solid panel.

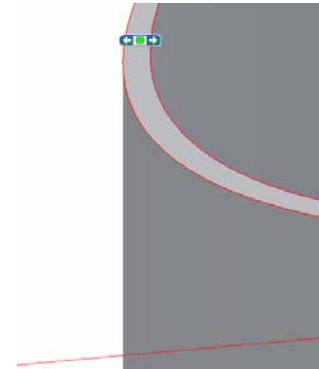


Figure 1A-10C: Move the cursor closer to the surface.

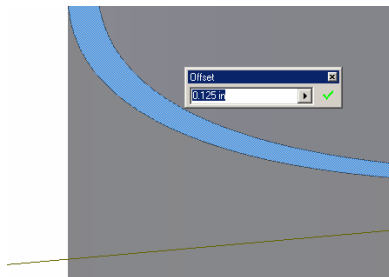


Figure 1A-10D: Selecting the surface and drag out a new work plane.

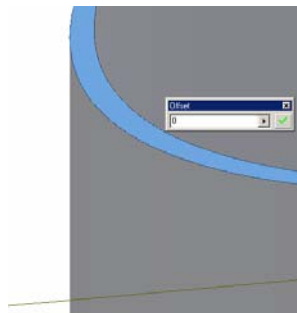


Figure 1A-10E: Typing 0 for Offset.

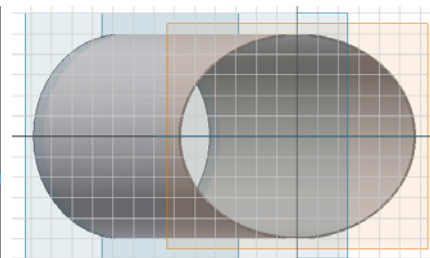


Figure 1A-10F: Starting a new sketch on the new work plane.

Now we are ready to delete irrelevant features from the **Model** panel. Click-select the **Flat Pattern** feature and right-click to select **Delete** from a shortcut menu; a warning windows opens (*Figure 1A-10G*); click the **OK** button and the **Flat Pattern** is deleted. Next, click-select the *Cylinder* feature and right-click to select **Delete** from the shortcut menu, blue geometry indicator lines appears on the solid (*Figure 1A-10H*); the **Delete Feature** window opens (*Figure 1A-10I*); uncheck both “**dependent sketches and features**” and “**dependent work features**” (*Figure 1A-10I*); click the **OK** button. Next, click-select the *Truncation* feature and right-click to select **Delete** option from a shortcut

menu (red geometry indicator lines appear); and in the Delete Features dialog window that opens, click the **OK** button (*Figure 1A-10J*). All irrelevant features are deleted and only the new work plane and sketch for the top piece remain (*Figure 1A-10K*).

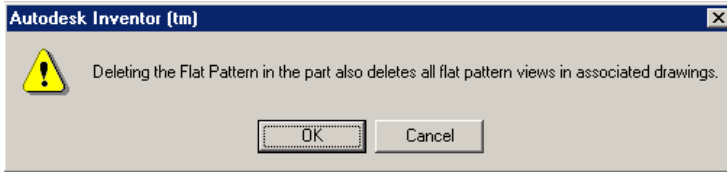


Figure 1A-10G: Delete Flat Pattern warning message.

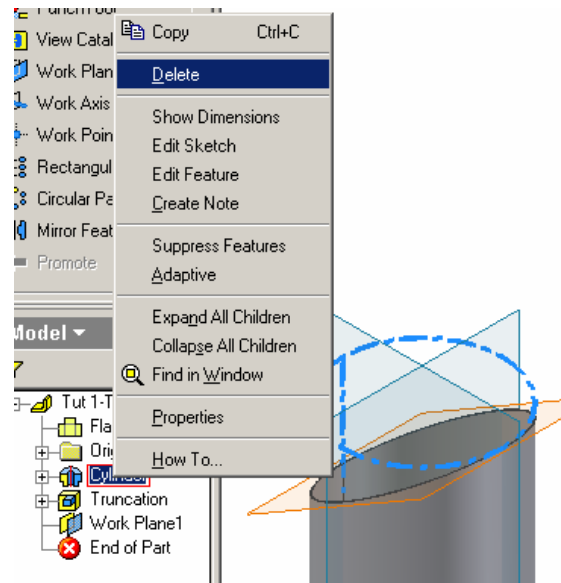


Figure 1A-10H : Delete the Cylinder

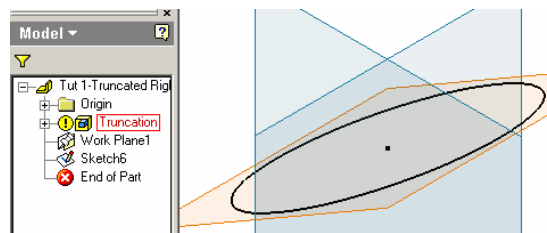
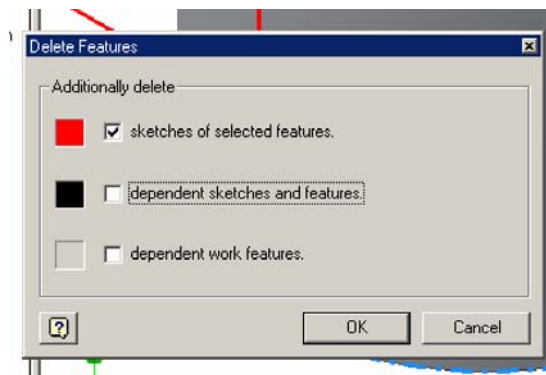


Figure 1A-10K: The remaining work plane and sketch.

Figure 1A-10I: Delete Features dialog window.

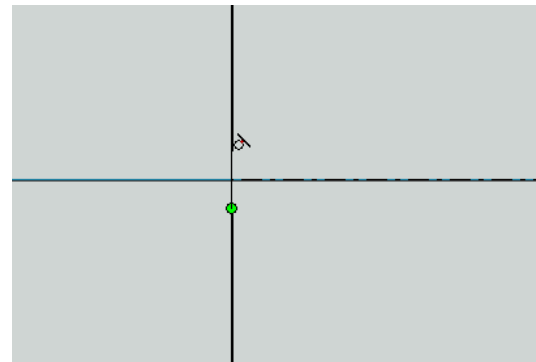
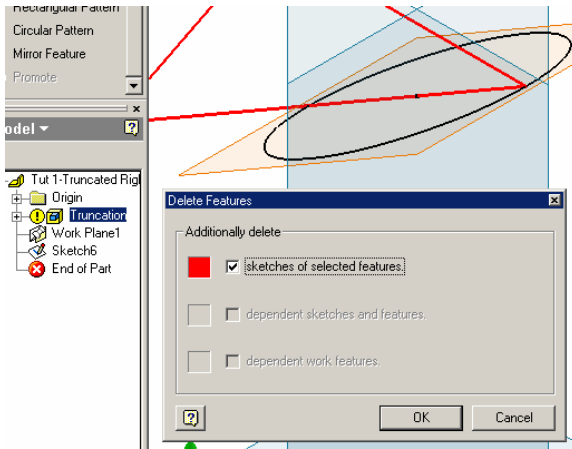


Figure 1A-10L: Filling in the gap.

Figure 1A-10J: Delete Features dialog window.

Remember that this is a gap in the original cylindrical tube, which is also projected onto the new sketch for the top piece; and this gap needs to be filled for the sketch to be extruded into a 3D solid. Click-select the new sketch from the Model panel; and right-click for the shortcut menu and select the **Edit Sketch** option. Click-select the **Look At** tool button and then the new sketch feature from the **Model** panel. Use the **Zoom Window** tool several times to see the gap. Use the **Line** tool to draw a line filling in the gap; remember to click on the two endpoints when the green snap mark is shown (Figure 1A-10L). Click the **Return** button to exit the sketch.

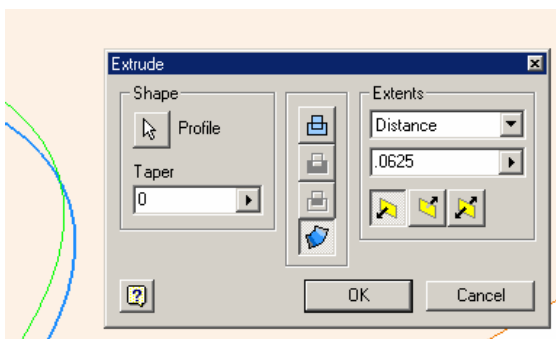


Figure 1A-11A: If the sketch is not closed, then the **Surface** type is automatically selected in the **Extrude** tool's dialog window.

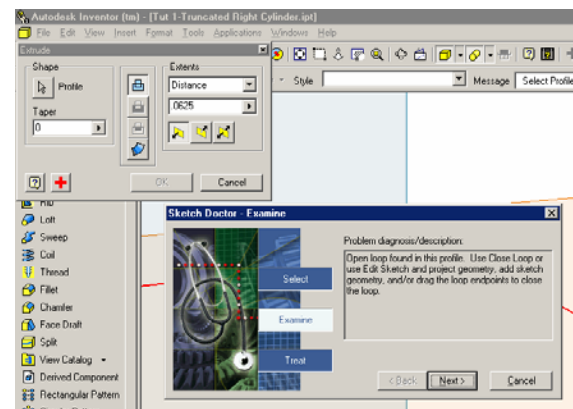


Figure 1A-11C: The **Sketch Doctor-Examine** dialog window.

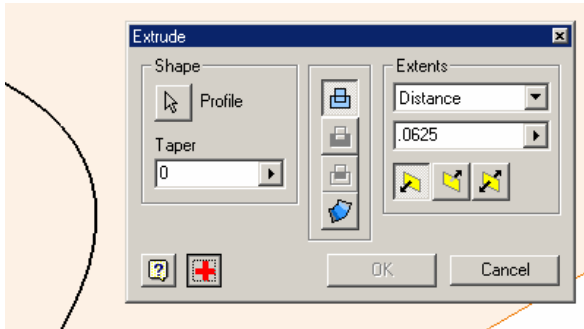


Figure 1A-11B: The **Join** type; and the Red Cross Button.

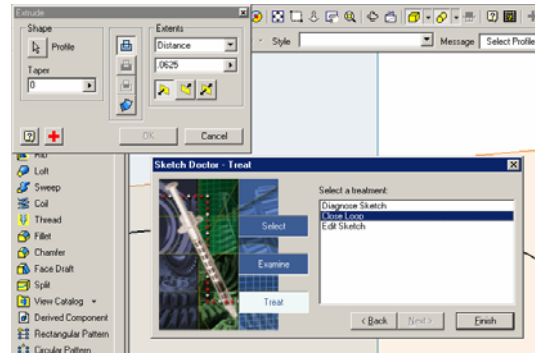


Figure 1A-11D: the **Close Loop** option.

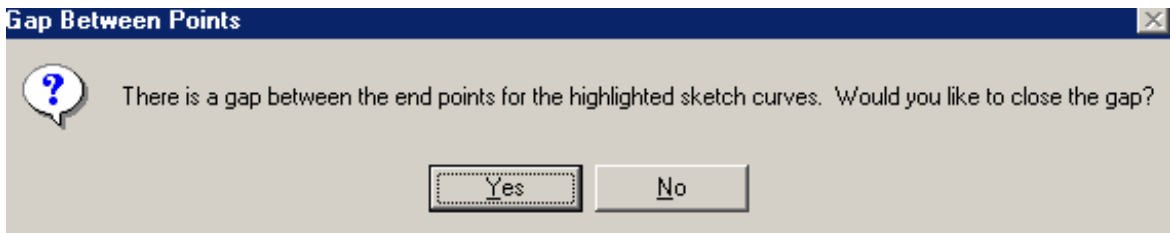


Figure 1A-11E: The **Gap Between Points** warning message window.



Figure 1A-11F: the **Close Loop** message window.

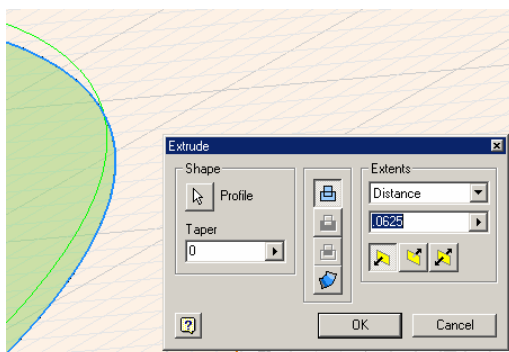


Figure 1A-11G: The loop closed and the **Join** option works.

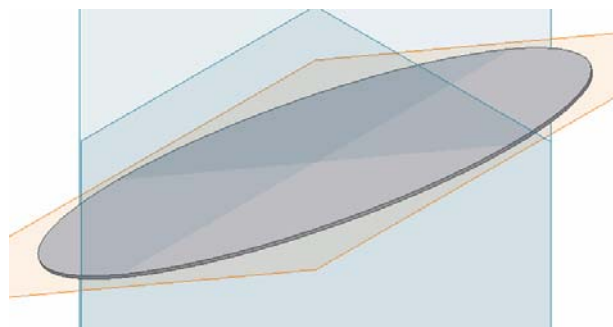


Figure 1A-11H: The Top piece extruded.

Next, extrude this sketch into a solid sheet metal piece. In the **Features** panel, click-select the **Extrusion** tool, set 0.0625 (the thickness) for Distance in the **Extrude** dialog window; if the sketch is a closed loop, then the **Join** option (the top button in the middle column) will show as recessed or selected; a dark green 3D extruded edge line and a light green area inside the 2D profile will be shown (*Figure 1A-11G*); sometimes, like in this case, the sketch is not a “closed loop,” then the **Surface** button (the bottom one in the middle column) is recessed or selected; and only a dark green 3D extruded edge line is shown (*Figure 1A-11A*). Click-select the **Join** button and a “Red Cross” button pops up on the lower left corner, next to the “?” button (*Figure 1A-11B*); click on the “Red Cross” button to open the **Sketch Doctor-Examine** dialog window (*Figure 1A-11C*); select the “**Close Loop option**” option and click the **Finish** button (*Figure 1A-11D*); click the “**Yes**” button in the “Gap Between Points” message window that opens (*Figure 1A-11E*); click the “**OK**” button in the “Close Loop” message window (*Figure 1A-11F*). The loop closed and the Extrusion option button is recessed or selected, and both light green area and dark green outline are shown on or above the sketched profile; click the “**OK**” button (*Figure 1A-11G*). The Top piece extruded. The top piece of the truncated cylinder is thus created (*Figure 1A-11H*). In the **Model** panel, change the name of this feature to *Top*. Click-select the surface of the *Top* model and click the **Flat Pattern** tool button in the **Sheet Metal** panel to open the **Flat Pattern** window (*Figure 1A-11I*).

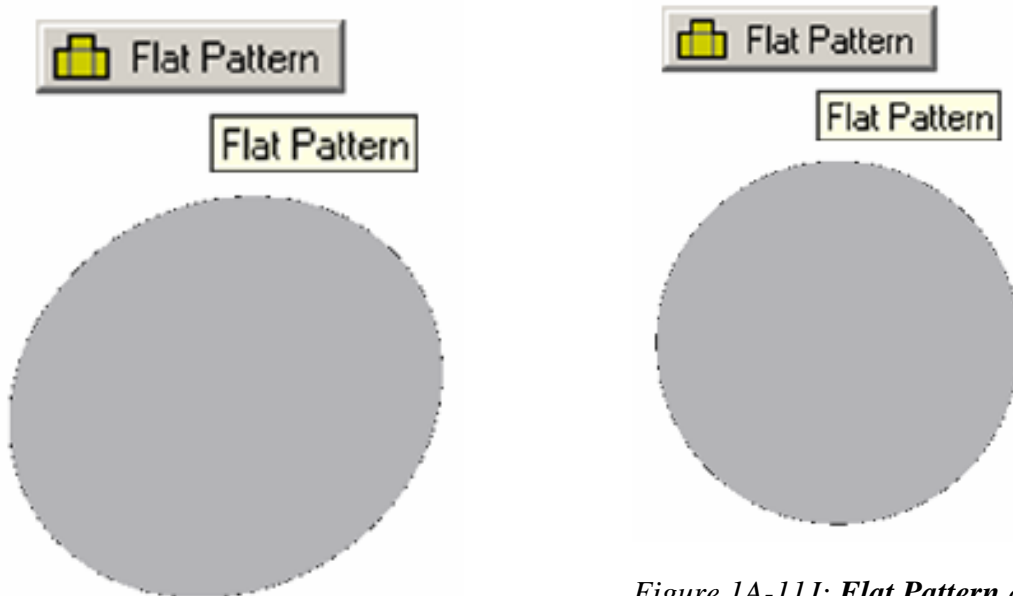


Figure 1A-11I: Flat Pattern of the Top Piece.

Figure 1A-11J: Flat Pattern of the Base Piece.

Section 4: Create the base piece of the truncated cylinder

We will use similar plus new tools and procedures to create the base piece of the truncated cylinder (*Figure 1A-11J*). Since we know that the shape of the base piece is exactly a circle with a 10-inch diameter, we will create a new sheet metal 3D file; but in order to make assembly of the parts easier later on, we will create this *Base* part in such a way that at least two of their three XY, XZ, and YZ coordinate Planes match those of the *Cylinder* part file. Go to the menu **File**→**New**, or click on the **New** tool icon on **Standard Tool Bar**. The **Open** window appears. Select the **English** tab, then select the *Sheet Metal (In).ipt* template, click the **OK** button. A new sheet metal file opens. In the **Model** panel, click the cross on the left of the *Origin* folder to view the **Planes, Axis and Origin** features. Shift-select the **XZ Plane** and the **Center Point** features, then right-click for the shortcut menu and check the **Visibility** option. The **Center Point** appears as a yellow dot on the light blue **XZ Plane** on the screen. Save the file as *Tut 1-Top.ipt* inside the folder named *Tut 1-Truncated Right Cylinder*. Click the **Return** button in the **Command Bar** to dismiss the default sketch plane. Go to **View**→**Isometric** menu to switch to isometric view for a more convenient visualization. Select the **XZ Plane** and then click the **Sketch** button in the **Command Bar**. The sketch grids are shown on the screen.

Click-select the **Project Geometry** tool in the **Sketch** tool panel and then the **Center Point** feature in the **Model** panel to project it onto the new sketch for snapping purpose. Select the **Circle Center Point** tool in the **Sketch** panel, move the mouse cursor closer to the projected Center Point on the **XZ Plane**; when the green snap mark appears, click to establish the center of the circle; draw the mouse outwards and click at any convenience location. Use the **General Dimensions** tool in the **Sketch** panel to apply a 10-inch diameter dimension. If the circle overflows out of the screen, then use the **Zoom Extent** tool to return it to full view. Click the **Return** button in the **Command Bar** to exit the sketch. Rename the sketch just created *Base Profile Sketch* in the **Model** panel. Save the file as *Tut 1-Truncated Right Cylinder.ipt* in the same *Tut 1-Truncated Right Cylinder* folder

Next, switch from **Sketch** panel to **Sheet Metal** panel. Click the **Styles** tool button in the **Sheet Metal** panel (*Figure 1A-12A*). In the **Sheet Metal Styles** dialog window, select **Sheet** tab, type 0.0625 for **Thickness**, and click the **Save** button. Close the dialog window by clicking the “X” button at its upper-right corner. Next, select the **Face** tool from the **Sheet Metal** panel; and in the **Face** tool dialog window, click the arrow button named **Profile** and click-select the circular profile of the sketch just created on the drawing screen; the area inside the profile turns light green and dark green 3D outline of the **Face** feature plus a red direction arrow appears; click the **Offset** button to make sure that the red arrow points downwards; and click the **OK** button to create the **Face** feature (*Figure 1A-12B*). The Base piece is completed. Rename the Face feature

Base in the **Model** panel. Select the circular surface of the *Base* model; click-select the **Flat Pattern** tool from the **Sheet Metal** panel to open the **Flat Pattern** window (*Figure 1A-11J*). Click the X button at the upper-right corner of the Flat Pattern window to return to the 3D folded model window. Save and close the file.

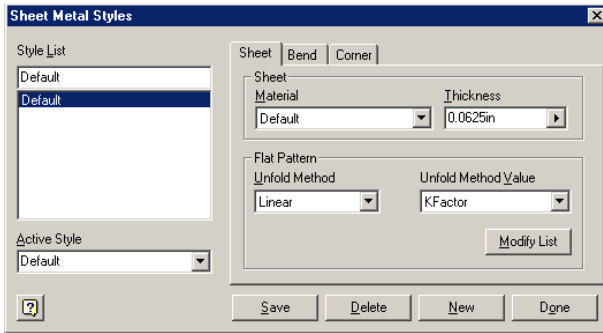


Figure 1A-12A: The Sheet Metal Styles dialog window.

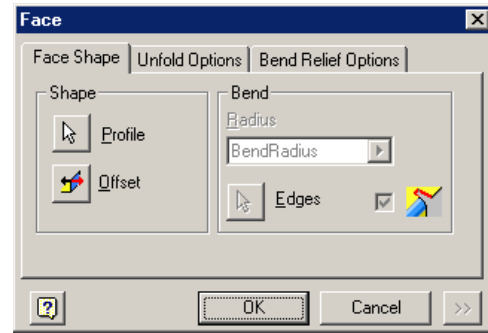
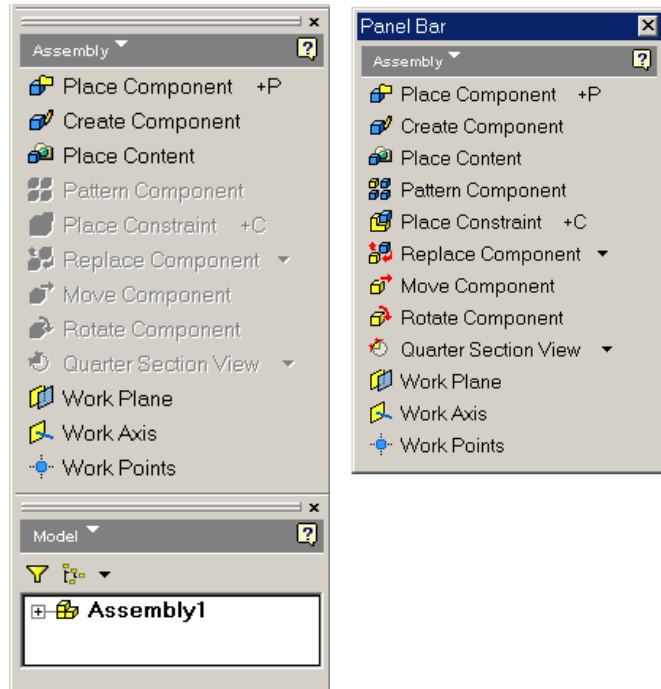


Figure 1A-12B: The Face tool dialog window.

Figure 1A-13A: The Assembly Panel Bar has many tools grayed out when a new assembly (.iam) file is created with no component placed (left); all tools are available with placed components (right).



Section 5: Create an assembly file

The primary objectives of creating an assembly file for sheet metal design are presentation and instruction for installation.

Go to the **File**→**New** menu, or click on the **New** tool icon on the **Standard Tool Bar**. The **Open** window appears. Select the **English** tab; then select the **Standard (in).iam** template; then click the **OK** button. A new assembly file opens with an **Assembly** panel (top left) and a **Model** panel (bottom left) on the left side of the drawing screen, as shown in *Figure 1A-13A*. Save the file as *Tut 1-Assembly.iam* inside the *Tut 1-Truncated Right Cylinder* folder. Go to the **View**→**Isometric** menu to switch to an isometric view.

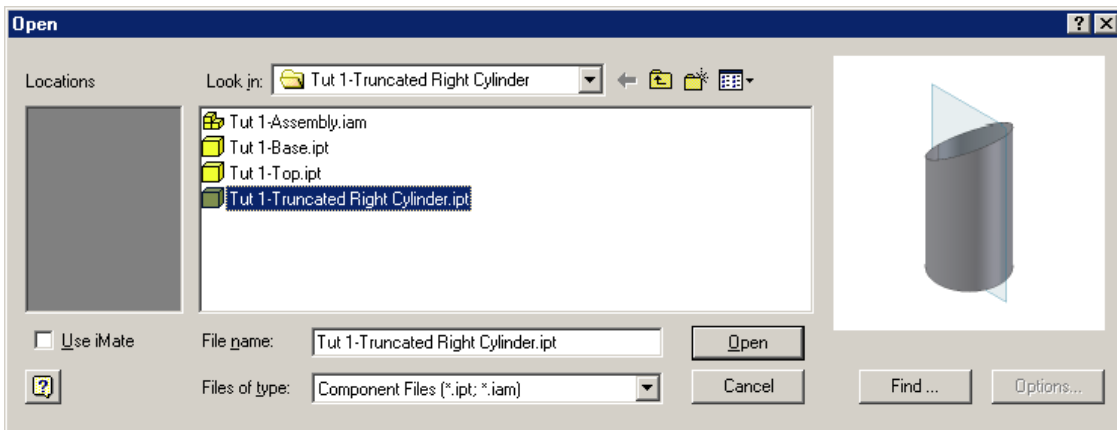


Figure 1A-13B: The Open window for the Place Component tool.

Click-select the **Place Component** tool in the **Assembly** panel; the **Open** dialog window opens (*Figure 1A-13B*); select the *Tut 1-Truncated Right Cylinder.ipt* file and click the **Open** button. The cylinder part appears on the screen. Click at any convenient location (but away from the three **Planes** of the assembly) to create a copy of the part. Right-click for the shortcut menu and select the **Done** option. Repeat the same process for the *Top* and *Base* parts (*Figure 1A-13C*). Next, go to the **Model** panel, click on the crossed box on the left of each part and then of **Origin** folder to show the **Planes**; hold the **Shift** key and select the invisible or “gray” **Planes**, then right-click for the short cut menu and check the **Visibility** option to turn on **Visibility** for all remaining planes of all imported parts. All three **Planes** of the imported parts appear in the screen in light blue). This step of turning on the **Visibility** option for all **Planes** can help you visualize what happens in the assembling process.

Next, constraint the three pairs of similar planes between the Top, the Truncated Cylinder, and the coordinate planes of the assembly, two parts at a time. Select the **Place Constraint** tool in the **Assembly** panel. In the **Place Constraint** dialog window, select **Mate** for **Type** and **Flush** for **Solution** (Figure 1A-13D). Move the mouse cursor to the **XY Plane** of the Cylinder (the edge of the Plane turns blue); click on the **Plane** (the Plane turns blue with a red arrowed coordinate indicator); move the mouse close to the **XY Plane** of the Top; click on this **XY Plane**; click the **Apply** button in the **Place Constraint** dialog window (the two parts mate immediately). Repeat the same process to mate the **XZ Plane** and **YZ Plane** as well. An alternative but more efficient approach is to go to the **Model** panel, click the same **Plane** features of both parts to be mated, one by one, and after you hear a loud sound, click the **Apply** button in the **Place Constraint** dialog window.

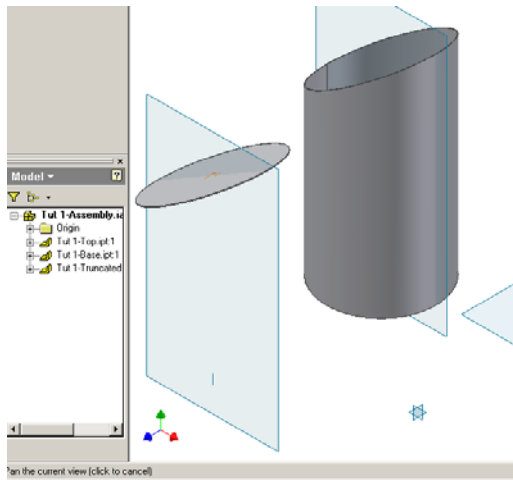


Figure 1A-13C: Placing Components.

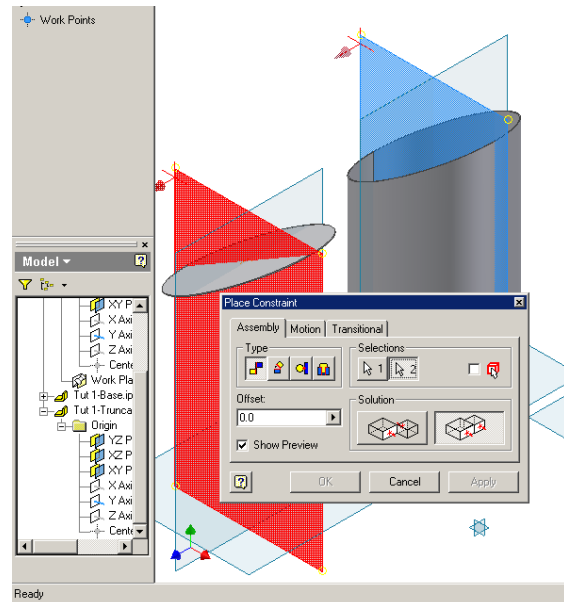


Figure 1A-13D: Constraining components with *Mate Type* and *Flush Solution*.

Next, mate the *Base* with the *Truncated Cylinder*. First move the *Base* closer to the bottom of the *Cylinder*. Select the **Move Component** tool in the **Assembly** panel (Figure 1A-13A); click on the *Base* model to select it and drag it closer to the *Cylinder* model; click again to set the *Base*; then right-click and choose the **Done** option in the shortcut menu to dismiss the tool. Next, select the **Place Constraint** tool again; choose **Mate** for **Type** and **Mate** for **Solution**, type 0.00 in the **Offset** text field; then click the top surface of the *Base* to select it (Figure 1A-13E); next, use the **Rotate** and **Zoom Window** tools in the **Standard** tool bar to rotate the model and zoom in so that the bottom section surface of the *Truncated Cylinder* can be seen and selected (Figure 1A-

13F). Move the cursor closer to and click-select this section surface; then click the **Apply** button in the dialog window (Figure 1A-13G and Figure 1A-13H). The top surface of the *Base* plate and the bottom sectional surface of the *Cylinder* are mated. However, the two parts are not concentric yet.

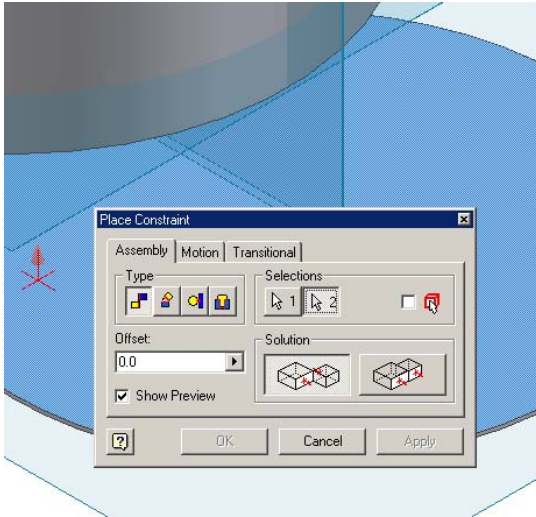


Figure 1A-13E: Constraining components with **Mate Type** and **Mate Solution**, with 0.0 **Offset** value.

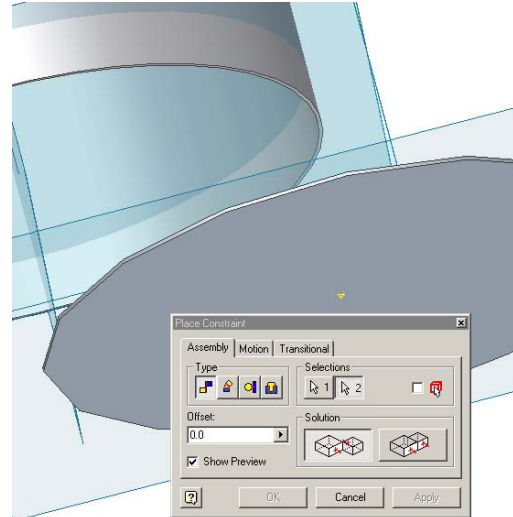


Figure 1A-13F: Using the **Rotate** tool to rotate the model.

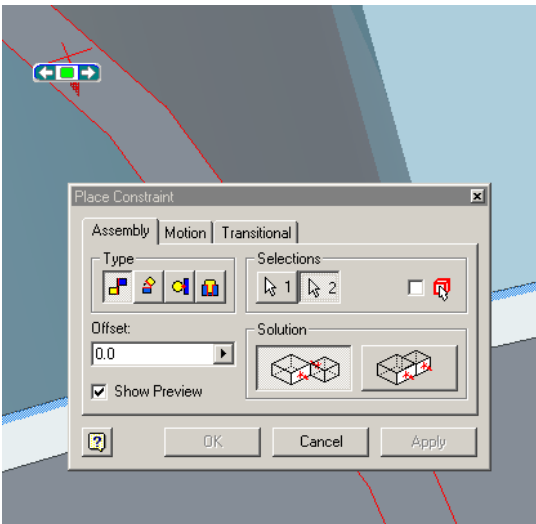


Figure 1A-13G: Using **Zoom Window** tool to zoom in. Outline of the section surface turns red when the cursor is moved closer to it.

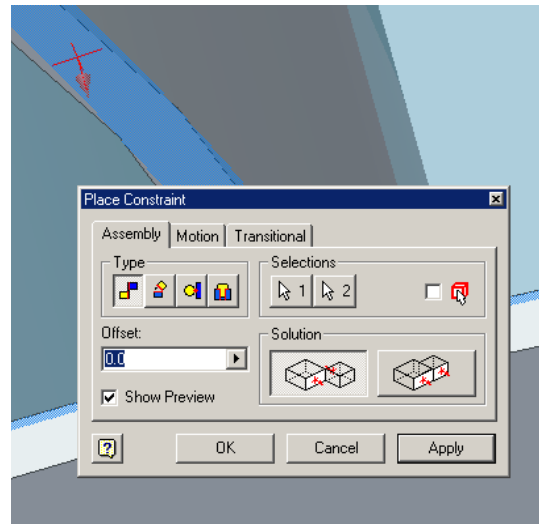


Figure 1A-13H: The surface of the section turns blue at the mouse click with the **Place Constraint** tool.

Next, apply an **Insert** constraint to make the *Cylinder* and the *Base* concentric. While the **Place Constraint** dialog window is still open, choose **Insert** for **Type** and **Opposite** for **Solution**. Move the mouse cursor closer to the *Cylinder*; red axis and circular edge appears; click once to select the *Cylinder* and the circular edge turns blue (*Figure 1A-13I*); next, move the mouse cursor closer to the top surface of the *Base*, red arrow pointing to the same direction and circular edge appears (*Figure 1A-13J*); click once to select the *Base*, and the red circular edge turns blue; click the **Apply** button in the **Place Constraint** dialog window and the *Base* jumps to the *Cylinder* (*Figure 1A-13K*). The two parts are now concentrically mated. Close the **Place Constraint** window. Save and close the file.

Since our only purpose for mating the parts in an assembly file is presentation (not for interference checking or any other purposes), an alternative process of mating the parts with the **Place Constraint** tool is to first create all parts in the correct position in the 3D space in terms of their relations to each other, and to use the **Mate** as **Type** and **Flush** as **Solution** options to mate each of their **XY**, **YZ**, and **XZ Planes** to those of the assembly file. This simpler technique requires some simple strategic planning before you start creating the parts, and will be covered in subsequent Modules.

Sometimes, it is worthwhile to see the model in **Wireframe Display** mode. To view the model in wireframe, go to the **Standard** tool bar, click on the inversed triangle next to the yellow “**Shaded Display**” icon for a pull-down menu and select the **Wireframe Display** icon (*Figure 1A-13L*).

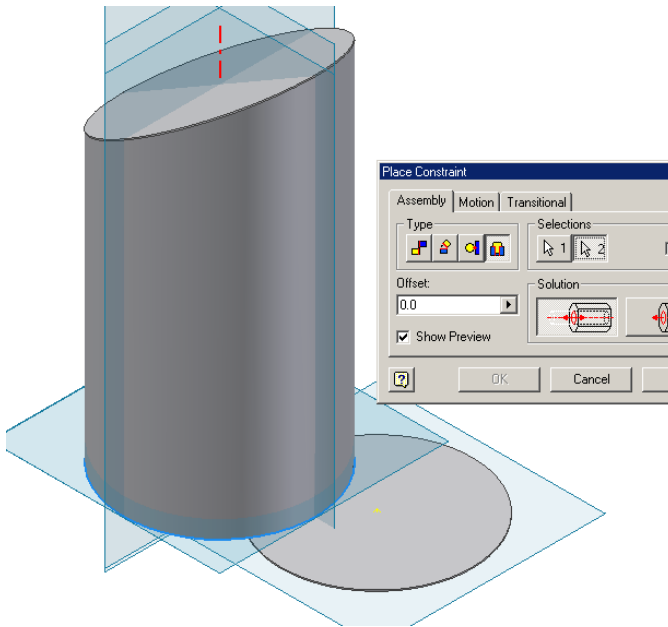


Figure 1A-13I: **Insert** for **Type** and **Opposite** for **Solution** for a concentricity mate.

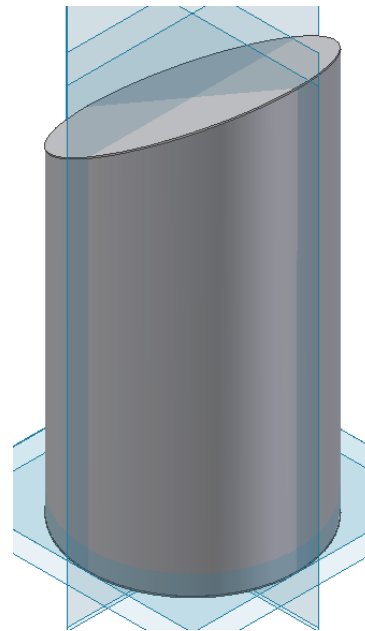


Figure 1A-13K: The two parts mated concentrically.

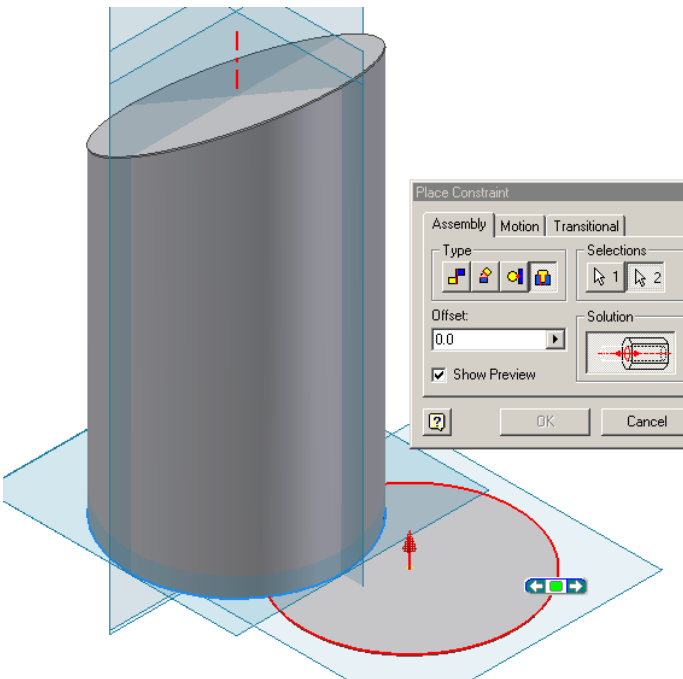


Figure 1A-13J: The concentricity mate.

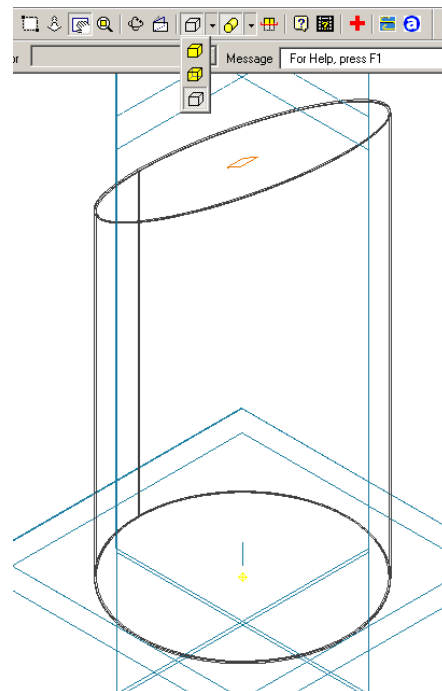


Figure 1A-13L: The **Wireframe Display** mode.

The *Module 1A* is completed so far in terms of the application of Inventor program in solving parallel-line development for a truncated right cylinder. Similar project can be created to have seams added to the parts (*Figure 1A-14A* through *Figure 1A-14D*), which is beyond the scope of this *Module*, due to associated trade-specific technical details. Such skills will be covered in another *Module*, or in a sheet metal design course.

Since the *Module 1A* is the starting *Module* of the *Manual* and has to cover a lot of basic interface and skills of the Inventor program, it is somehow lengthy; subsequent *Modules* would be shorter or much shorter. For users with no previous experience with Inventor, it is hereby recommended that this *Module* be tried at least twice, so as to build a smooth path to the subsequent *Modules*. For those of you who have previous experience with Inventor or fell experienced after completing the Inventor's on-line Tutorial lessons plus this *Module*, proceed immediately to the next *Module*.

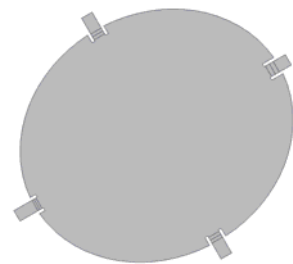
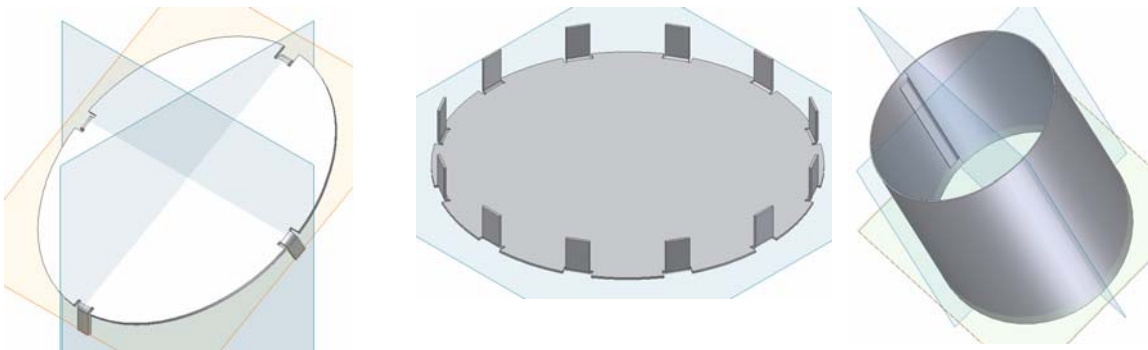


Figure 1A-14A: Top piece with flanges and flat pattern.

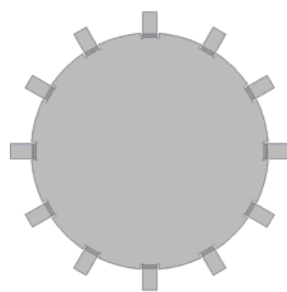


Figure 1A-14B: Base piece with flanges and flat pattern.

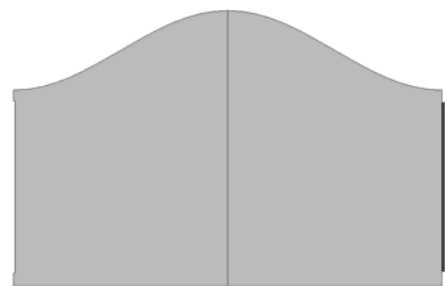


Figure 1A-14C: Lateral Cylinder piece with flanges and flat pattern.

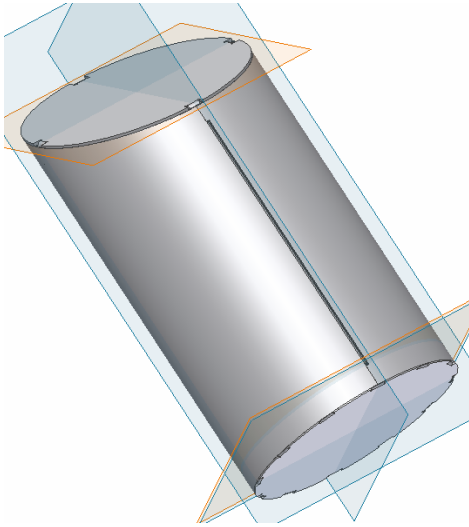


Figure 1A-14D: Assembly of all three pieces