Module 6A:
Creating Poly-Cylindrical (Gore) Sheet Metal Pieces for a Spherical Space

The sphere is a "solid of revolution" geometrically formed by revolving a curved profile around a central axis; and the surface of a sphere is a "double-curved" surface because the once curved profile is revolved or "curved twice" (*Figure 6A-1A*). A double-curved 3D surface cannot be accurately developed into a 2D flat pattern surface; however, it can be substituted by segments of single-curved surfaces, such as cylindrical (called the "Gore") or conical surfaces (*Figure 6A-1B* and *Figure 6A-1C*), which creates an approximate development of a 2D flat pattern for the sphere.

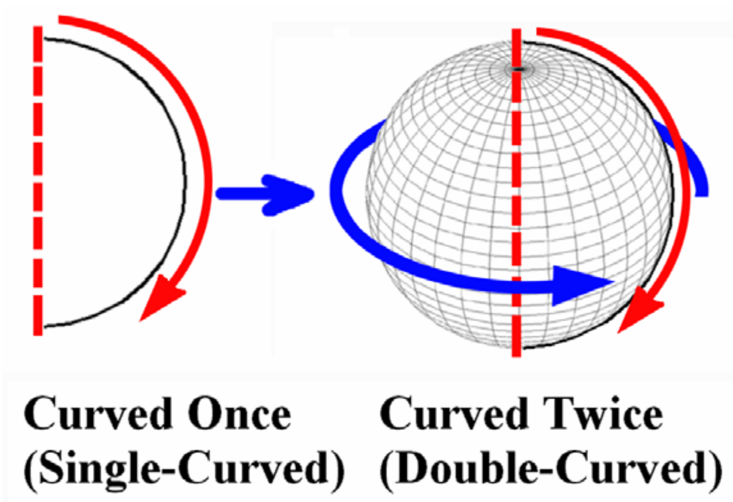


Figure 6A-1A: The "curved once" semi-circular profile used to generate a "curved twice" or "double-curved surface of a sphere.

The technique of using cylindrical (called the "Gore") or conical surfaces for the creation of spherical surfaces is used in the construction of domes of astronomical observatories, etc., which are normally semispherical; or of large models of the Earth in public spaces, which are normally fully spherical.

The number of substituting cylindrical or conical surface segments depends on the sizes of the spherical or semi-spherical spaces, as well as the sizes of the available sheet metal materials; the greater number the spherical surfaces are divided into non-spherical surface segments, the better approximation; however, in the real-world design situation, such number is limited by the above-mentioned factors; and for a 10-foot diameter dome, 12 Gores or 6 conical segments are enough.

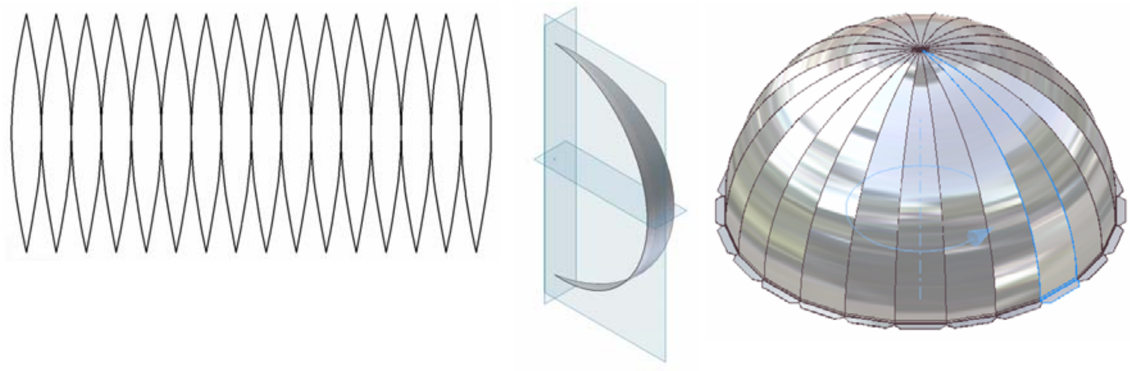


Figure 6A-1B: The poly-cylindrical Gore Method for the approximate development of a spherical surface by substituting the non-developable double-curved spherical surface with segments of "Gores" or developable single-curved cylindrical surfaces. Segments of Gores (left and middle); the assembled semi-Gores for a domical surface of an astronomy observatory (right). In any typical textbook on descriptive geometry, people normally call the top vertex of the Gore the North Pole (abbreviated as N) and the bottom vertex of the Gore the South Pole (abbreviated as S).

In Module 6A, we will learn how to create a folded 3D model of and an approximate 2D flat surface pattern for a 120-inch or 10-foot diameter semi-spherical dome of an astronomy observatory, by the poly-cylindrical (Gore) method, which consists of cutting the surface of the semi-sphere into segments of cylindrical surfaces, assembled normally in the East[West direction or along the latitudinal lines, as shown in *Figure 6A-1B*.

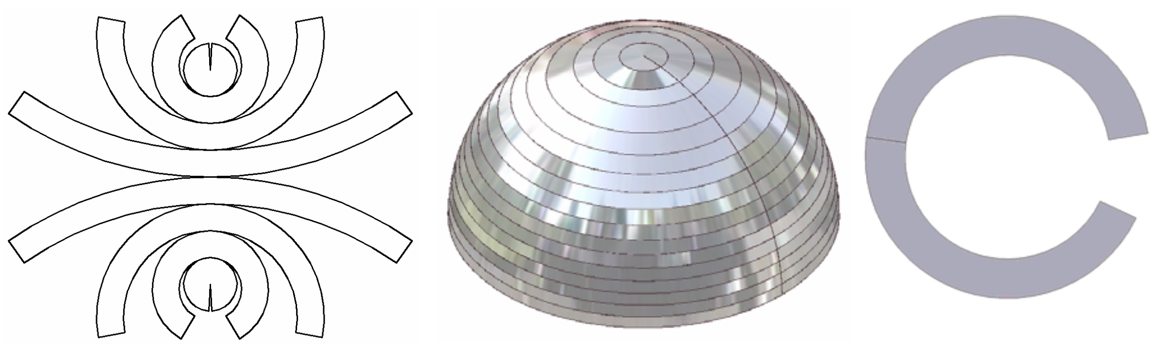


Figure 6A-1C: The poly-conical Method for the approximate development of a spherical surface by substituting the non-developable double-curved spherical surface with segments of developable single-curved conical surfaces. Full spherical surface (left); semi-spherical dome surface (middle); a single conical segment (right).

Step1: Creating the basic semi-cylindrical sheet metal piece with connecting Flange

Launch Inventor, start a new **Sheet Metal (in).ipt** file under the **English** tab. Turn **Visibility** on for the **XY Plane** and **XZ Plane**. "*Sketch1*" is created by default on the **XY Plane** (the one parallel to your computer's screen); rename Sketch1as *Cylindrical Profile Sketch* in the **Model** panel; use the **Project Geometry** tool to project the **Center Point** onto the sketch for a snap point; use the **Circle Center Point** and **General Dimension** tools to draw a 120-inch circle snapped-centered at the projected **Center Point** (*Figure 6A-2A*); use the **Offset** and **General Dimension** tools to offset a larger circle with added 0.12-inch radius for the thickness of the sheet metal material (*Figure 6A-2B*); use the **Line** tool to draw a horizontal line upward and a vertical line rightward from the projected **Center Point**; and use the **Trim** tool to trim off all unneeded line and circle segments, so as to leave an upper-right quarter of the section profile, with an inner 120-inch diameter and an outer 120 + 0.12 x 2 = 120.24 inch diameter circular curves connected at both ends by a 0.12-inch long horizontal and a .12-inch high vertical lines (*Figure 6A-2C* and *Figure 6A-2D*); click the **Return** button to exit the *Cylindrical Profile Sketch*. Save the file as *Tut-Sphere (Gore).ipt* in a new folder named *Tut-Sphere*. Save often throughout the entire project.

Next, use the **Extrude** tool with Join as Type, 120 (inches) and **Distance** for **Extents** and **Midplane** for **Direction**, extrude the sketch profile into a quarter cylindrical sheet metal part, and rename the **Extrude** feature *Cylindrical Piece* in the **Model** panel (*Figure 6A-2E*).

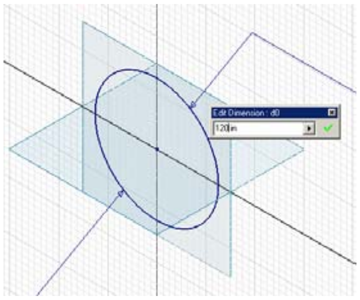


Figure 6A-2A: The 120-inch circle.

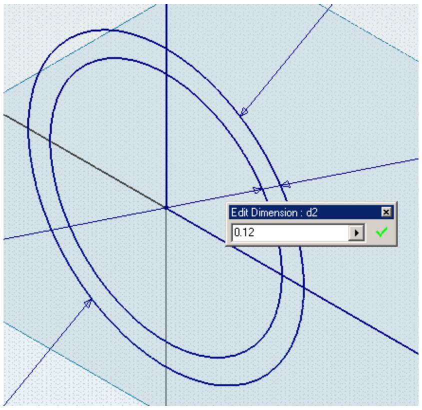


Figure 6A-2B: Offsetting a 0.12-inch circle outwardly.

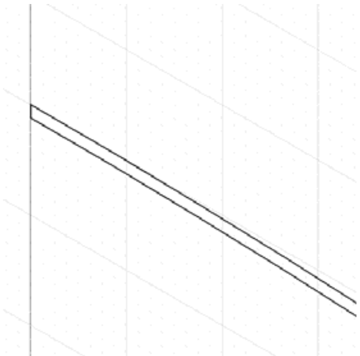


Figure 6A-1C: The top-left detail of the Cylindrical Profile Sketch.

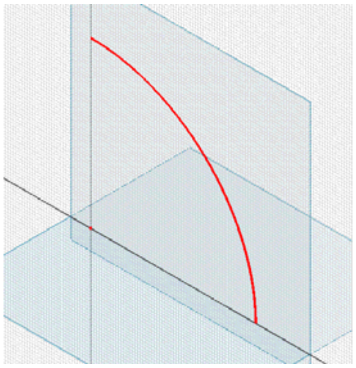


Figure 6A-2D: The Cylindrical Profile Sketch.

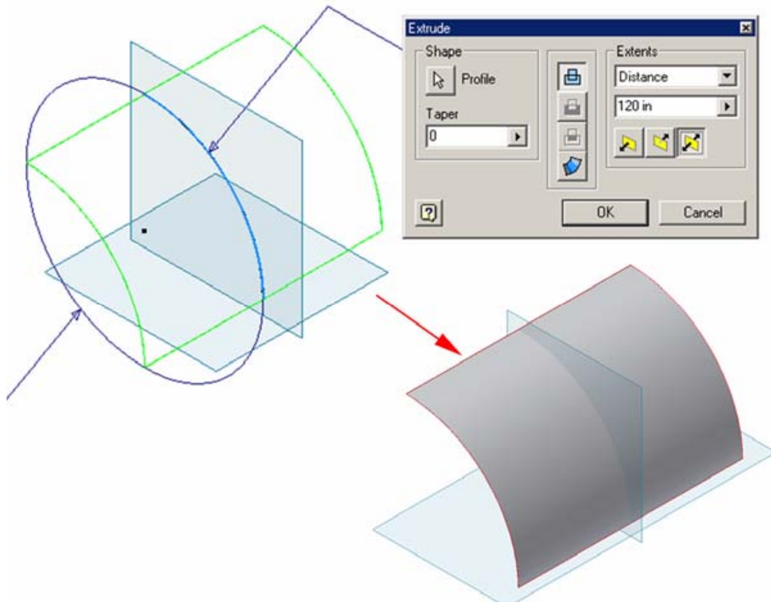


*Figure 6A-2E: The Cylindrical Piece **Extrude** feature .*

Next, create a patching **Face** feature at the bottom of the quarter cylindrical sheet metal part for the addition of a seam **Flange** feature. Click-select the **Work Plane** tool, move the cursor closer to the **YZ Plane** (red outlines immediately appear to indicate a new **Work Plane** feature, as shown in (*Figure 6A-3A*), click-select it; then move the cursor closer to the outer bottom edge of the *Cylindrical Piece* model and click-select it (*Figure 6A-3B*); orange outline of a perpendicular work plane and an **Angle** text field with 90 deg text (meaning a perpendicular work plane) appears (*Figure 6A-3C*); highlight the text and delete it; then type 0 for a parallel **Work Plane** feature, the outline of the default suggested perpendicular **Work Plane** changes into that of a parallel **Work Plane** (*Figure 6A-3D*); click the green check mark to create the new **Work Plane** and rename it *Flange Patch Work Plane* in the **Model** panel.

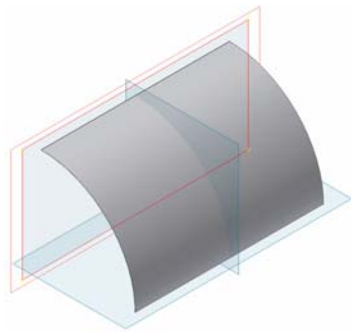


*Figure 6A-3A: Picking the **YZ Plane**.*

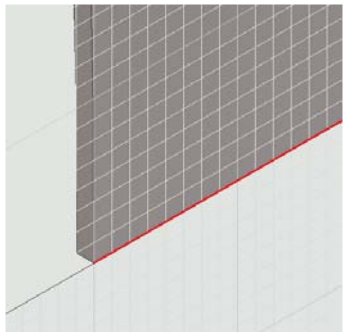


Figure 6A-3B: Picking the bottom outer edge.

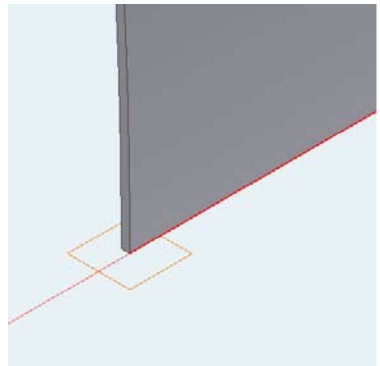


*Figure 6A-3C: The default outlines suggesting a perpendicular **Work Plane** feature with 90 deg in the text field.*

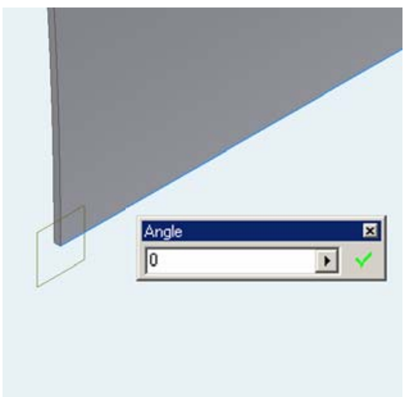


*Figure 6A-3D: Deleting the 90 deg **Angle** text and type 0 for a parallel work plane.*

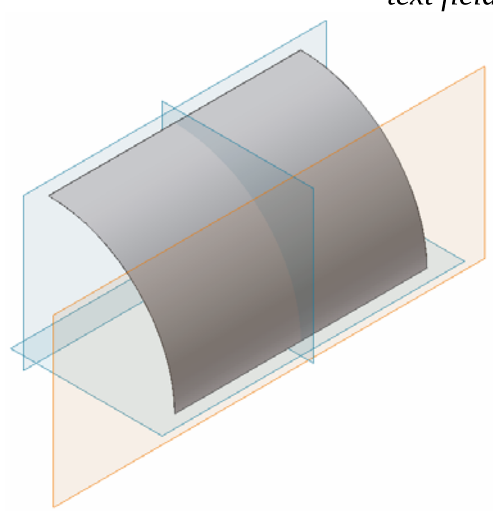


Figure 6A-3E: The Flange Patch Work Plane.

Next, select the new *Flange Patch Work Plane*, click the **Sketch** button to start a new sketch, and rename it *Flange Patch Sketch* in the **Model** panel; use the **Project Geometry** tool to project the outer bottom edge onto the sketch, and use the **Line** tool to add additional vertical and horizontal lines to complete a rectangular profile; and use the **General Dimension** tool to apply a 0.25-inch linear dimension to the vertical edge of the profile (*Figure 6A-3F*); click the **Return** button to exit the *Flange Patch Sketch*. Next, use the **Face** tool to create the patching **Face** feature, with the **Offset** direction arrow pointing inwardly toward the **Center Point** (*Figure 6A-3G* and *Figure 6A-3H*); rename the **Face** feature *Flange Patch face* in the **Model** panel. Next, use the **Flange** tool to add a seam as show in *Figure 6A-3I*, and *Figure 6A-3J*, with the bottom outer edge selected, 2 in for **Distance** and 90 deg for **Angle**, the **Offset** arrow pointing upwards and the

Flange feature extending in the rightward direction (or outside of the domical space, on the roof's edge of the astronomy observatory); click the **OK** button to create the **Flange** feature and rename it *Bottom Flange* in the **Model** panel. The basic cylindrical part is now ready to be cut into a Gore.

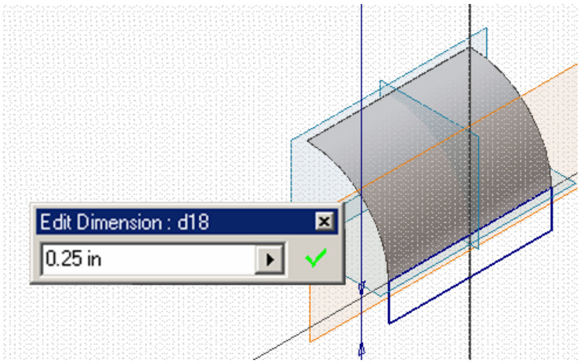


Figure 6A-3F: The Flange Patch Sketch.

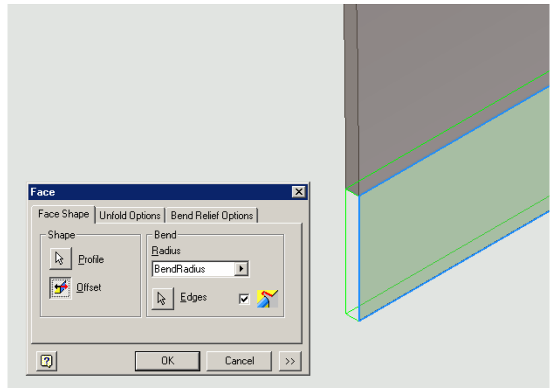


Figure 6A-3G: The Flange patch Face.

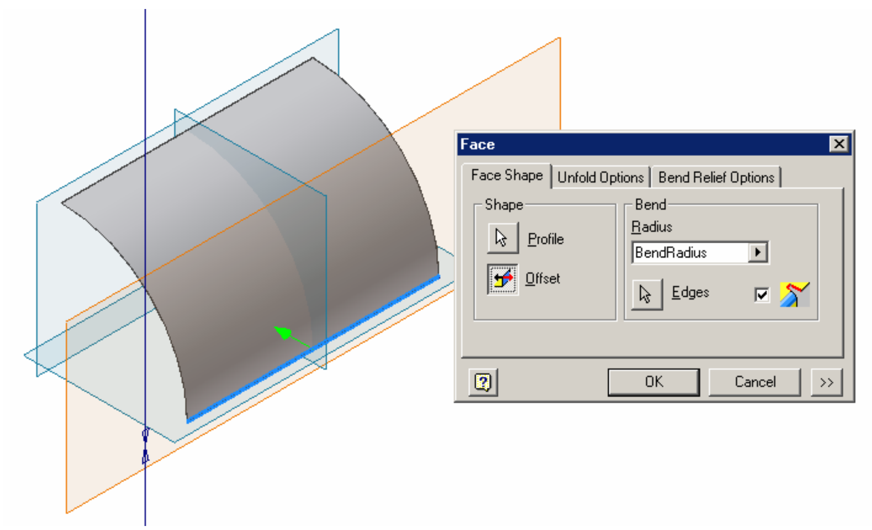


Figure 6A-3H: The direction of the Face extrusion.

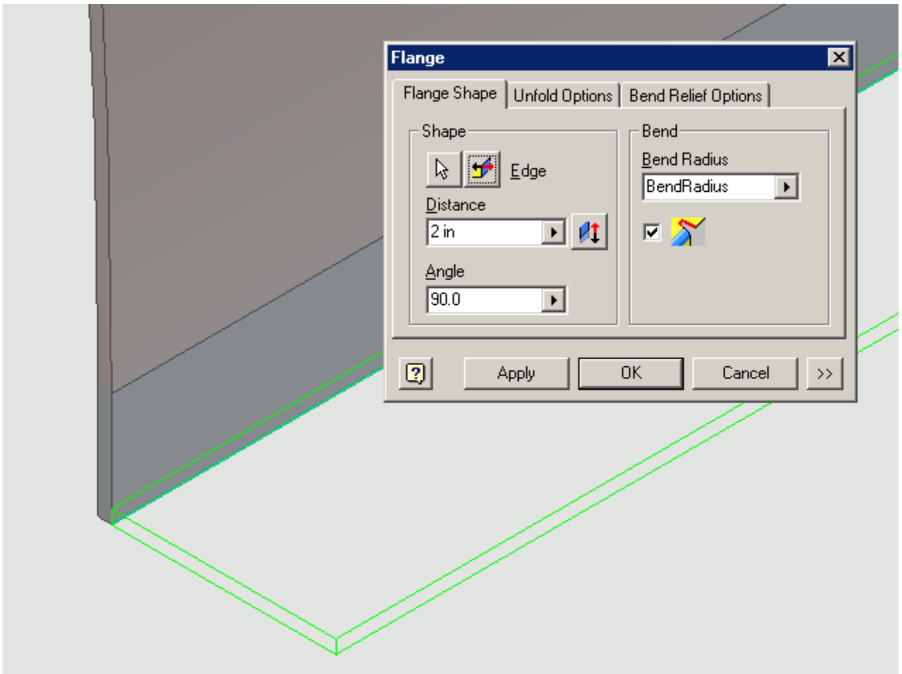


Figure 6A-3I: The upward and rightward Bottom Flange.

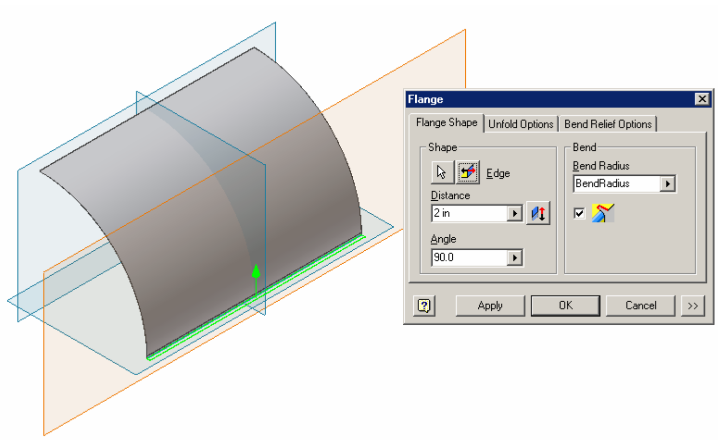


Figure 6A-3J: The upward Offset direction and the rightward Flip Direction.

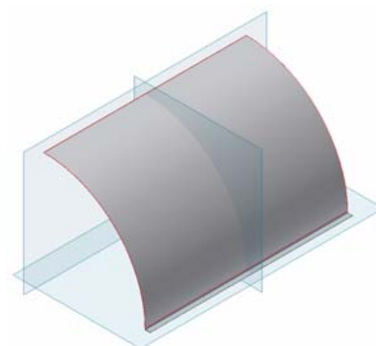


Figure 6A-3K: The basic cylindrical part with bottom seam complete.

Step 2: Cutting the basic semi-cylindrical sheet metal piece with connecting Flange into a "Gore" piece

First, draw the Gore cutoff profile. Select the **XZ Plane** and click the **Sketch** button to start a new sketch, and rename it *Gore Cutoff Sketch* in the **Model** panel; use the **Look At** tool to switch to an orthographic view; use the **Project Geometry** tool to project the **Center Point** onto the sketch, use the **Line** tool to draw a leftward horizontal line and an inclined line both from the projected **Center Point**, go to the **Command Bar** to change the **Style** of the horizontal line to **Centerline**; next, select the **Mirror** tool (*Figure 6A-4A*), click the arrow button on the left of **Select** button name in the **Mirror** tool dialog window and click-select the inclined line on the screen; then click the **Mirror Line** arrow button in the **Mirror** tool dialog window and click-select the horizontal center line; click the **Apply** button to create the mirrored inclined line on the lower side of the centerline, and click the **Done** button to exit the tool; next, use the **General Dimension** tool to apply a 15-degree angular dimension between the two inclined lines, because in this project, the spherical surface will be substituted by 360º ÷ 24 Gores = 15º/Gore (*Figure 6A-4B*); next draw a large circle enclosing the entire 3D model (*Figure 6A-4C*) and use the **Trim** and other tools if needed to complete the *Gore Cutoff Sketch* (*Figure 6A-4D*); finally, click the **Return** button to exit the *Gore Cutoff Sketch*.

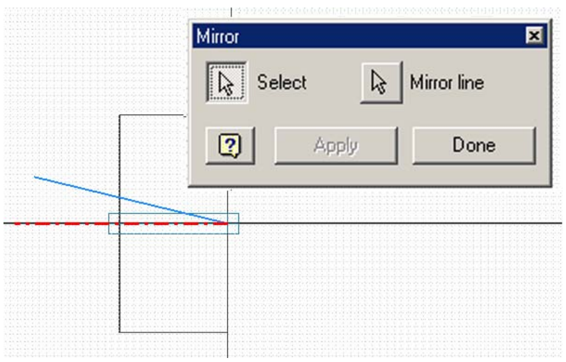


*Figure 6A-4A: The **Centerline Style** horizontal line, the inclined line and the **Mirror** tool.*

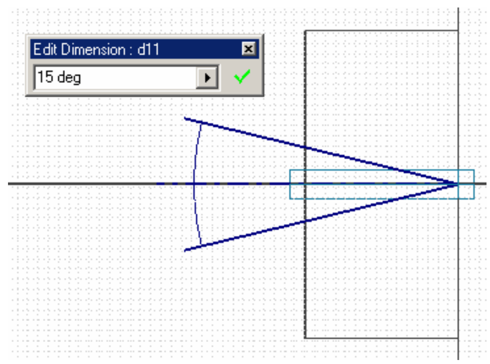


Figure 6A-4B: the inclined line mirrored around the centerline and a 15º angular dimension applied between the two inclined lines.

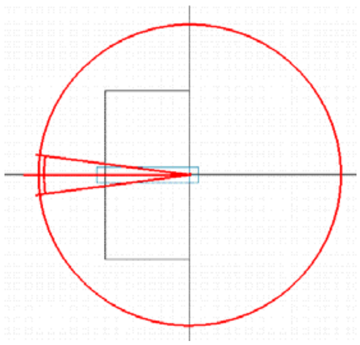


Figure 6A-4C: The basic profile.

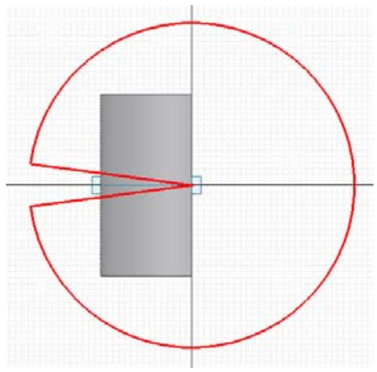


Figure 6A-4D: The complete Gore Cut Sketch profile.

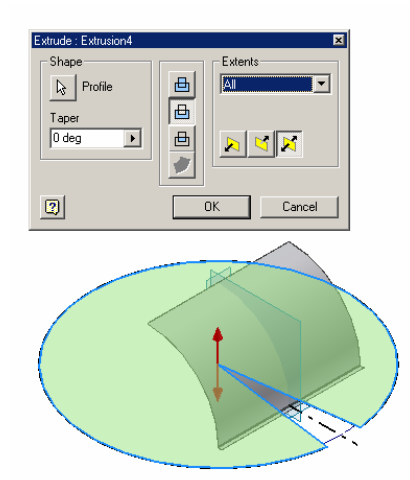


Figure 6A-4E: Cutting the semi- Gore segment.

Next, use the **Extrude** tool with **Cut** for **Type**, **All** for **Extents** and **Midplane** for **Direction** to cut the *Cylindrical Piece* into a semi-Gore segment (*Figure 6A-4E*), and rename the **Extrude** feature *Gore Cutoff* in the **Model** pane. Notice that the bottom flange of the semi-Gore has inclined, not straight end edges, which can be cut into straight edges and chamfered if desirable (*Figure 6A-5A*). Select the top surface of the *Bottom Flange* feature and start a new sketch, and rename it *Flange Ends Trims Sketch* in the **Model** panel; use the **Look At** tool to switch to an orthographic view and use the **Project Geometry** tool to project the top left and bottom left corner of the *Bottom Flange* onto the sketch (*Figure 6A-5B*); from the projected end corner points, use the **Line** tool to draw two rectangular profiles for cutoff (*Figure 6A-5C*); click the **Return** button to exit the *Flange Ends Trims Sketch*.

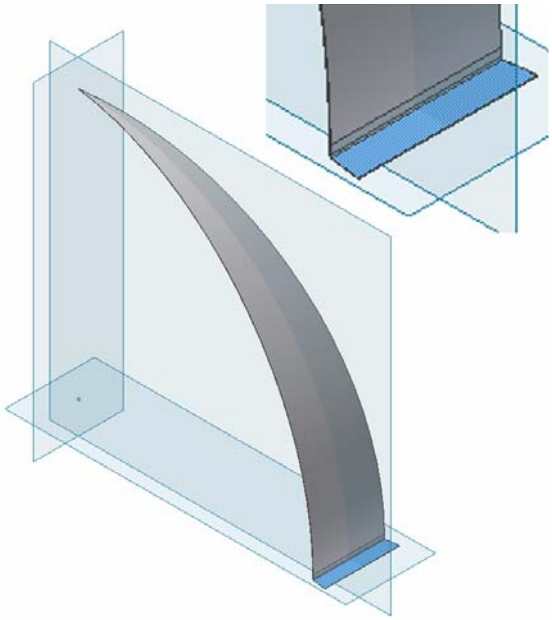


Figure 6A-5A: The inclined end edges of the Bottom Flange of the Gore.

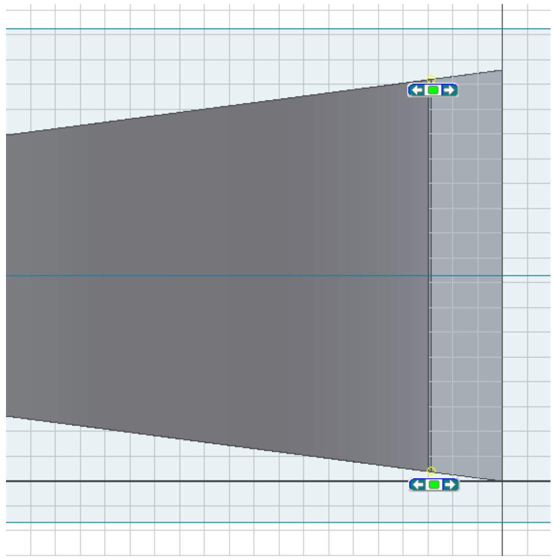


Figure 6A-5B: the top left and bottom left corner of the flange.

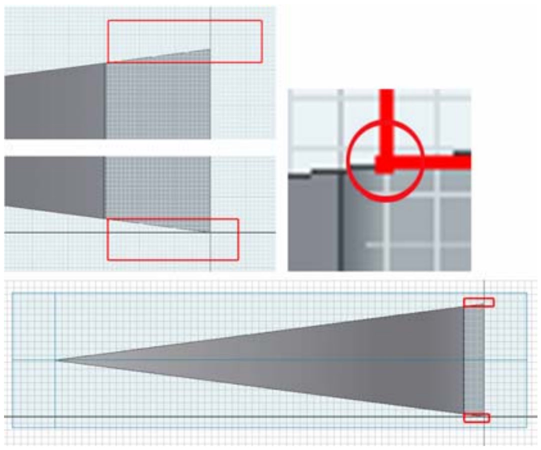


Figure 6A-5C: The rectangular cutoff profiles.

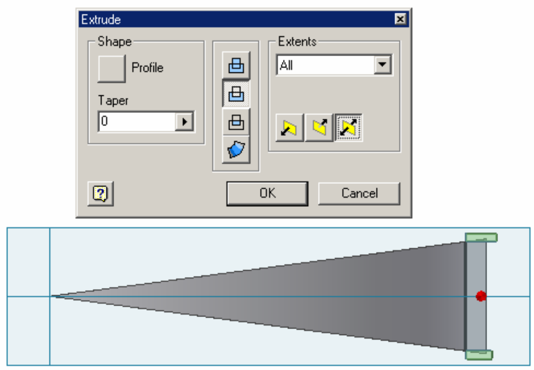


*Figure 6A-5D: Cutting off the inclined ends with the **Extrude** tool.*

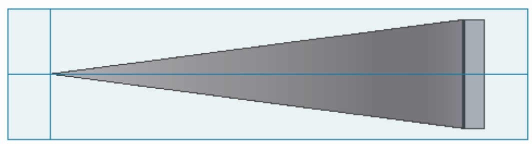


Figure 6A-5E: The Bottom Flange with straight ends.

Next, use the **Extrude** tool with both rectangular shapes selected as **Profile**, **Cut** as **Type**, **All** for **Extents** and **Midplane** for **Direction**, cutoff the inclined edges (*Figure 6A-5D* and *Figure 6A-5D*). An alternative tool is the Cut tool from the Sheet Metal panel, which by default will cut off a **Sheet Metal** feature such as **Flange** or a **Face** with the same sheet metal thickness as established in the previously mentioned **Sheet Metal Style** tool dialog window.

Next, use the **Corner Chamfer** tool to add 1-inch chamfers to both ends of the *Bottom Flange* seam (*Figure 6A-6A*).

Next, cut a vertex relief at the North Pole (*Figure 6A-6C*). Select the **XY Plane** to start a new sketch; rename it *Vertex Relief Cut Sketch* in the **Model** panel; use the **Project Geometry** tool to project the **Center Point** onto the sketch; use the **Line** tool to draw a vertical line from the projected **Center Point** up and beyond the vertex point; draw an inclined line starting from the projected **Center Point** and extending upward and rightward, or in the North-East direction; and use the **General Dimension** tool to apply a 0.5-degree angular dimension between the two lines (*Figure 6A-6B*); next, use the **Line** tool to draw a short line to connect the vertical and the inclined lines at the top to form an inversed triangular profile; next, click the **Return** button to exit the sketch. Next, select the **Extrude** tool and choose **Cut** as **Type**, **All** for **Extents** and **Midplane** for **Direction**, the inversed triangle of the *Vertex Relief Cut Sketch* as **Profile,** cut an opening at the North Pole vertex point of the *Gore Cutoff* **Extrude** feature. Rename the new **Extrude** feature *Vertex Relief Cut* in the **Model** panel.

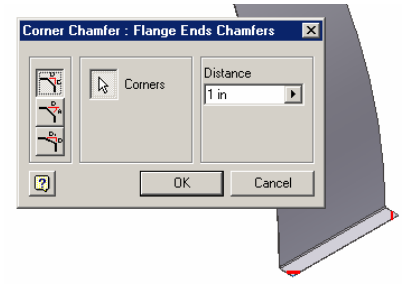


Figure 6A-6A: Adding ***Corner Chamfers****.*

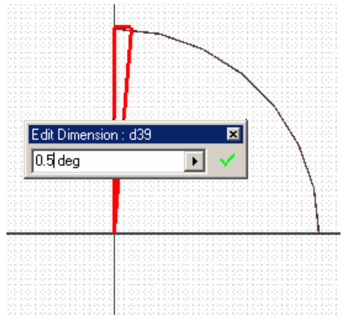


Figure 6A-6B: Applying a 0.5-degree angle.

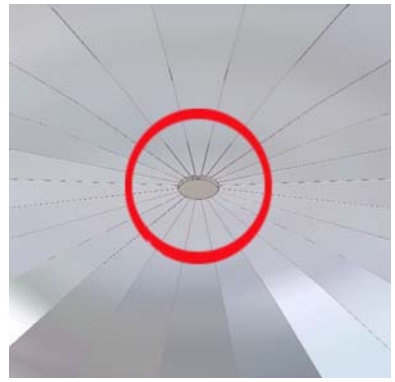


Figure 6A-6C: The vertex relief.

The half portion of a Gore or cylindrical piece is complete; click-select the outer surface of the 3D model and click the **Flat Pattern** button in the **Sheet Metal** panel to create a **Flat Pattern** view (*Figure 6A-6D*). The features of the 3D model are listed in the **Model** panel (*Figure 6A-6E*). Photo-realistic rendering of the 3D model can be applied by opening the Color pull-down menu from the **Command Bar** and select a desired material map (*Figure 6A-6F*). Save and close the file.

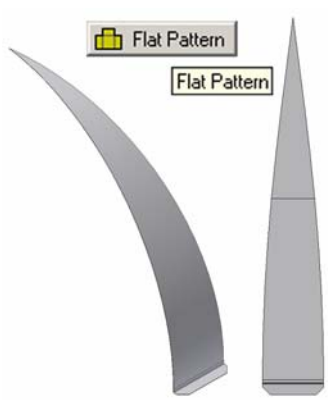


Figure 6A-6D: The complete single Gore piece and its ***Flat Pattern*** *view.*

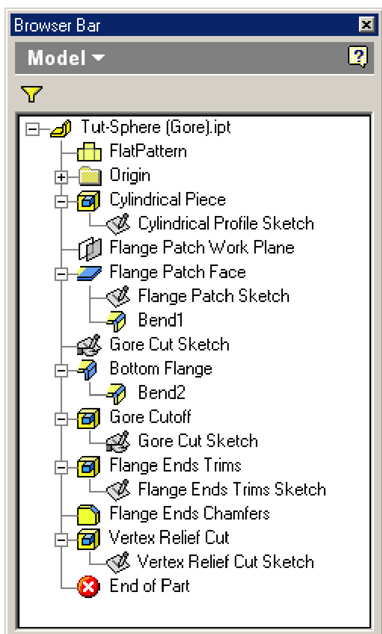


Figure 6A-6E: The features listed in the ***Model*** *panel.*

Figure 6A-16F: Rendering the 3D model with the ***Color*** *pull-down menu from the* ***Command Bar****.*

Step 3: Duplicating the "semi-Gore" piece in an Assembly (.iam) file to create a semi-domical surface

The last step in this project is the assembly of 24 cylindrical semi-Gore pieces to cover the semi-spherical surface of an astronomy observatory dome. Start a new Inventor **Standard (in).iam** assembly file under the **English** tab; use the **Place Component** tool to place the 3D model of the semi-Gore from the *Tut-Sphere (Gore).ipt* file into the new assembly file; and save it as *Tut-Sphere (Gore) Assembly.iam* in the same *Tut-Sphere* folder.

Next, use the **Place Constraint** tool with **Mate** as **Type** and **Flush** as **Solution** (*Figure 6A-7A*), mate the **XY Plane**, **XZ Plane** and **YZ Plane** of the "placed" semi-Gore model and the corresponding **Planes** of the assembly file, pair by pair, by click-selecting them in the Model panel, and clicking the **Apply** button after the program gives a loud sound.

Next, select the **Pattern Component** tool; in the **Pattern Component** tool's dialog window, select the **Circular** tab; click the arrow button on the left of the **Select Component** name text and click-select the placed 3D model, either in the **Model** panel or on the screen (red outline appears on the model when the cursor is moved closer to it; and when click-selected, red outlines turn blue); next, click the **Axis Direction** arrow button in the **Circular Placement** section and click-select the **Y Axis** in the **Model** panel (a red upward or downward arrow along a blue axis, and blue circular symbol of rotation, as well as multiple pieces of the patterned models appear on the screen); if needed, click the **Flip Direction** button on the right of the **Axis Direction** arrow button to change the direction of rotation; type 24.000 ul in the **Count** text field for 24 pieces, and 15.00 deg for 15º rotational angle between the neighboring semi-Gores (*Figure 6A-7B* and *Figure 6A-7C*); click the **OK** button to create the **Circular Pattern** feature with 24 copies of the semi-Gore model (*Figure 6A-7D*) and rename the **Circular Pattern** feature *Gore Pattern* in the **Model** panel (*Figure 6A-7E*).

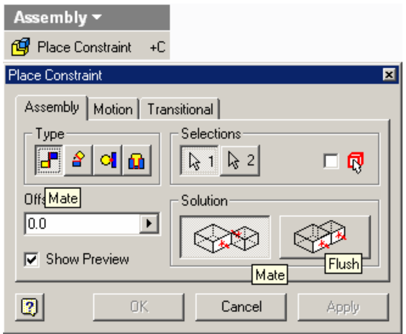


Figure 6A-7A: The ***Place Constraint*** *tool and dialog window.*

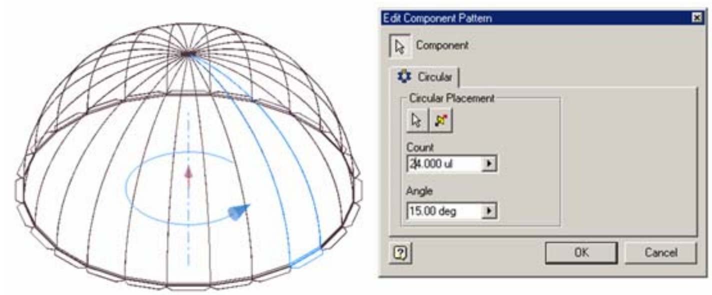
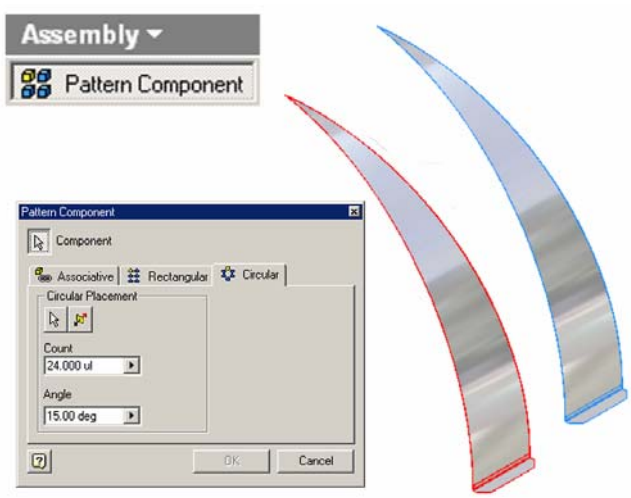


Figure 6A-7B: Red arrow along a blue axis, and blue circular symbol of rotation, as well as multiple pieces of the patterned models.



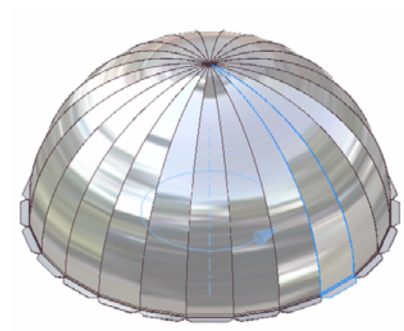
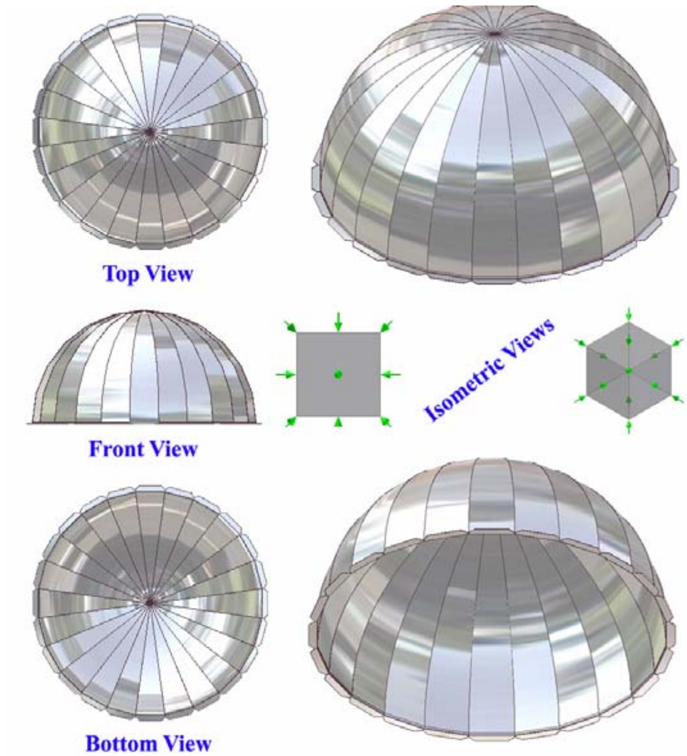


Figure 6A-7D: The completed assembly.

Figure 6A-7C: The ***Pattern Component*** *tool.*

The astronomy observatory's dome has been completely covered with 24 pieces of semi-Gore sheet metal parts. The orthographic and isometric views of the model and the features listed in the Model panel are shown in *Figure 6A-7E*.



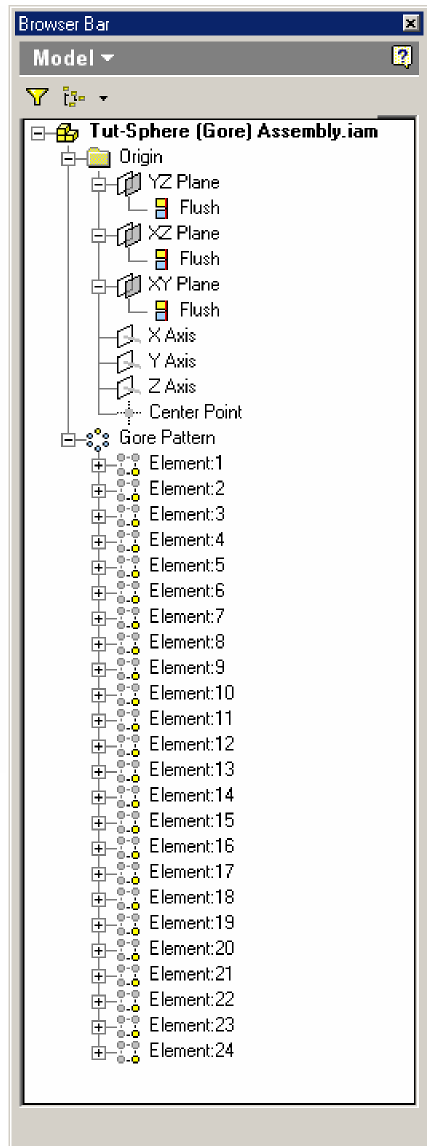


Figure 6A-7E: Orthographic and isometric views of the model (left); features of the model listed in the Model panel (right).