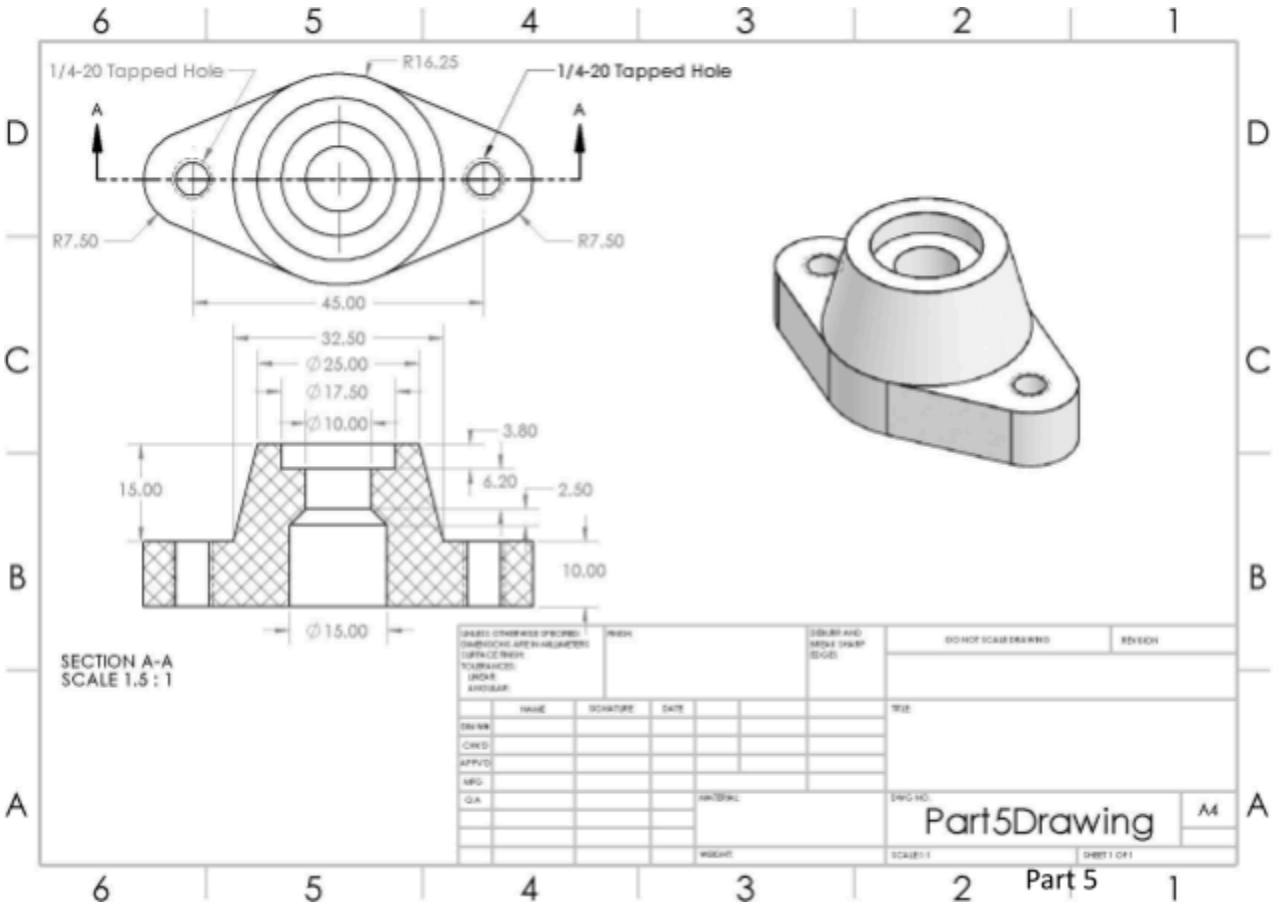
Part #5

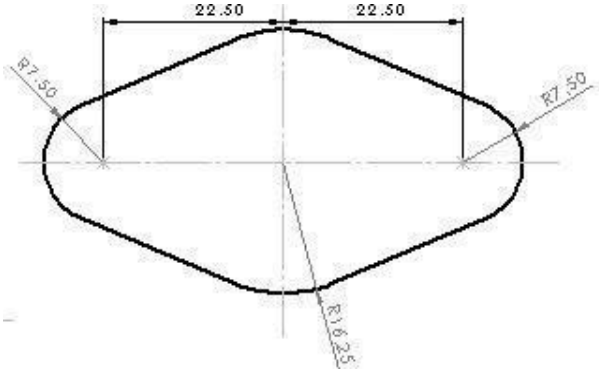


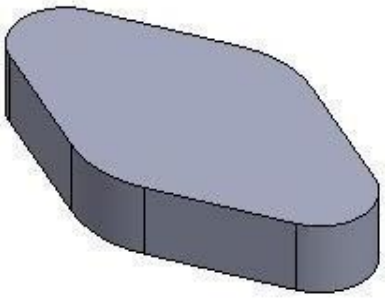
Drawing for Part 5 [unit: MMGS (mm)].

Please note that the following are the suggested steps but not necessarily the only way to develop this part.

1) Build the "base" in the following figure on the **Top plane**.
2) Draw two **Centerlines**, one **vertical** and one **horizontal**, both passing through the origin point.
3) Draw a large circle, centered at the origin point. Set the diameter as **32.5** mm.
4) Draw two smaller circles, one at each side of the large circle, both with the center sitting on the horizontal Centerline. Set the diameters of one of the smaller circles (e.g. the left one) as **15 mm**. Set the distance between its center and the origin as **22.5 mm**. Set the two smaller circles to be **symmetric** around the vertical Centerline.
5) Draw four **tangent** lines connecting the three circles. If the Tangent Relation is not set automatically, you need to set it manually (select both the line and the circle and then add relation of **Tangent**). The part should be fully defined only when the 4 lines are all tangent with the circles.
6) **Power Trim** all unnecessary lines, and the 2D sketch should look like the one in the following figure. (**Note:** This step may make the sketch Under Defined again. If so, you need to reset some dimensions.)

7) Turn this 2D sketch into 3D by clicking **Extruded Boss/Base**. Set Depth as **10 mm**.



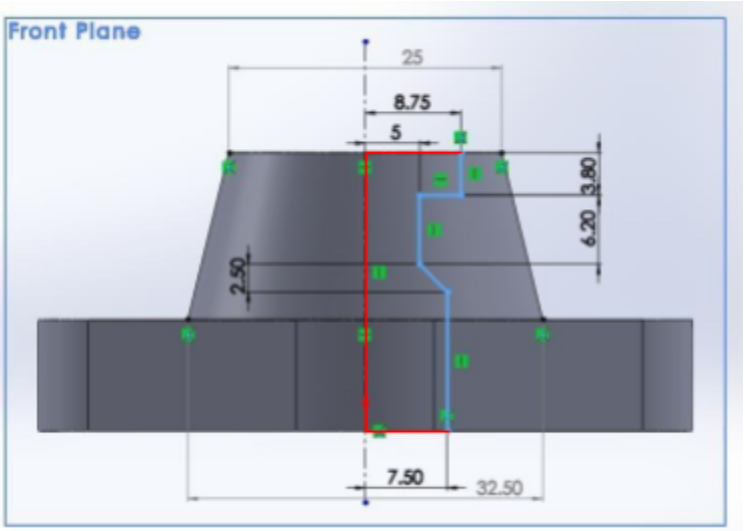


Next, we need to work on the upper feature. We will first draw a 2D sketch on the Front Plane and then revolve it using Revolved Boss/Base to turn the 2D sketch into 3D.

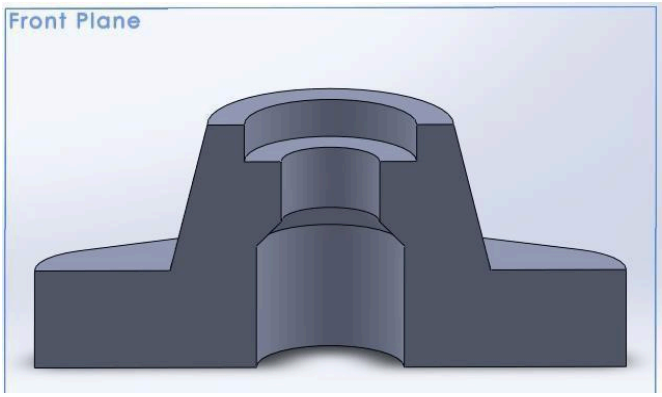
8) To make the **Front Plane** visible, click **Front Plane** in feature tree, then in the pop up menu click **Show**. Click **Sketch** to start drawing on the Front Plane.
9) First draw a **Vertical Centerline** passing through the origin point.
10) Using **Line**, draw a **Right Trapezoid** above the base, with one leg on the **Vertical Centerline**. Set the length of the top base of the trapezoid as **25/2 mm**; Set the length of the bottom base as **32.5/2 mm**; Set the height as **15 mm**.
11) Use feature of **Revolved Boss/Base**. In the Revolve option window, make sure the vertical Centerline is selected as the **Axis of Revolution.**

Then we will perform a **Revolved Cut** around the **Vertical Centerline**. To see the center plane better, we can click **Section View** which is at the top of the drawing window.

12) In the **Section View** option window, make sure **Front Plane** is the one selected. Click **Sketch** to start sketch on the Front Plane.
13) First draw another **Vertical Centerline** through the origin.
14) Draw **5 lines,** continuously from top to bottom, as shown by the blue lines in the following figure. Define the dimensions of these 5 lines.
Lengths from top to bottom: **3.8 mm, 6.2 mm, 2.5 mm**, 12.5 mm (the last one does not need to be defined). Radius from top to bottom: **17.5/2 mm, 10/2 mm, 15/2 mm.**


Front Plane
25
8.75
5
3.80
6.20
2.50
7.50
32.50

15) Before **Revolved Cut**, we need to notice that the sketch is still open. We have to close it by **adding 3 lines** (The positions of these 3 lines are shown by the red lines in the figure above).
16) Perform **Revolved Cut**. By checking the Section View, and/or by changing the view direction, you can see the result from the cut. It should look like the one in the following figure.



As the last step, we will add two holes on the base of this part.

17) Uncheck the **Section View**, and also hide the Front Plane (click **Front Plane** in the Feature tree, and click the **Hide**).
18) Click **Hole Wizard**. In option window, select **Straight Tap**; Set the Size as **1/4-20**; Set **End Condition** as **Through All**; uncheck **Near side countersink**.
19) Select the **Positions** of the two holes. Click the surface on the base where you want to drill the two holes, and then click **Normal To**.
Note: The center of the left hole should be concentric with the center of the left curve (same for the right hole). When you put your mouse cursor on the curve, the center point of the curve will be visible.

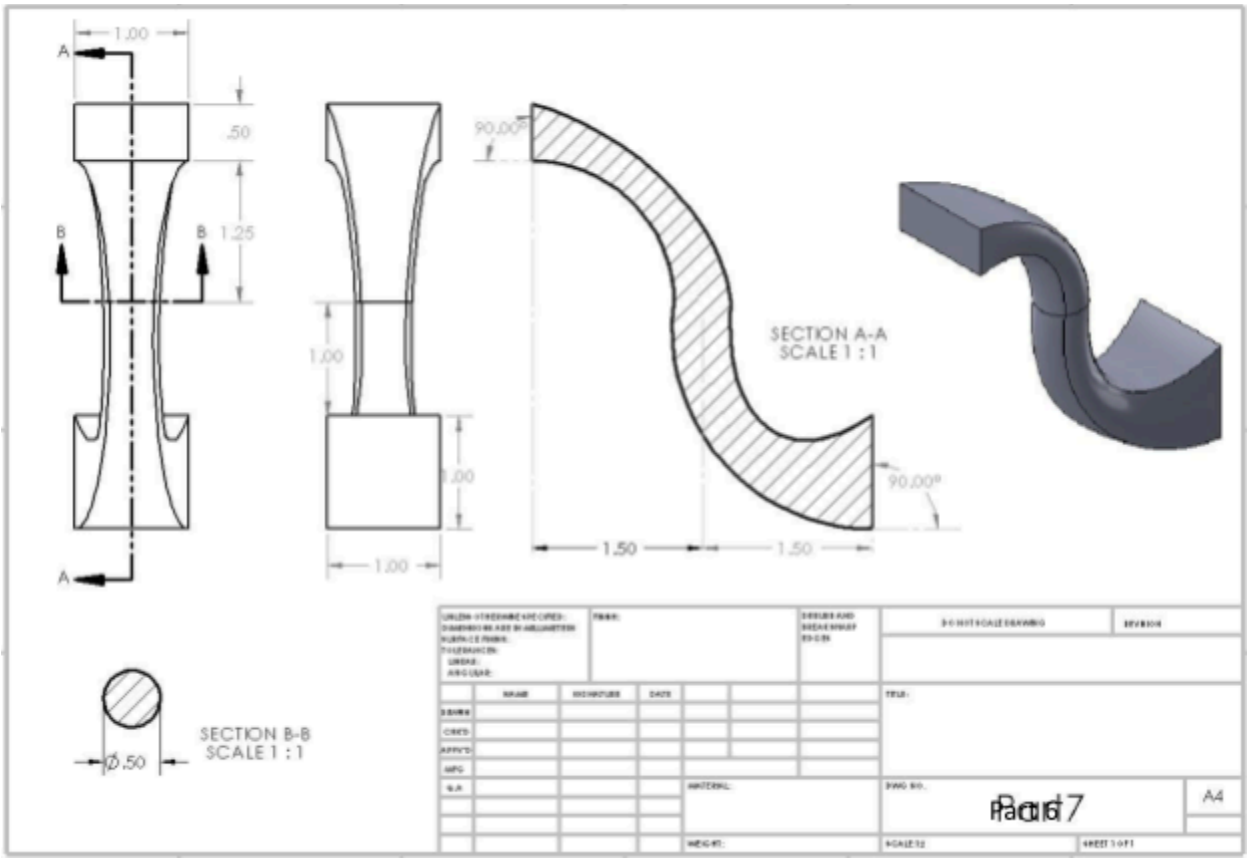
After drilling the holds, you might find that the treads in the holes are not visible. To make the treads visible, click **Tools**, then click **Options**, then click **Document Properties**, then click **Detailing**, then check **Shaded cosmetic threads**. Then click **OK**.

Before finishing, let's select **Materials** in the Feature Tree. After right-clicking **Materials**, click **Edit Materials**. Click **Aluminum Alloys**, then select **Alumina**, then **Apply**. The color of the part will change to white.

Part #6

(This part #6 is relevant to the design project of hip implant.)

In this practice, we will learn how to use Loft feature which allows generation of complex geometry in a single feature.



Drawing for Part 6 (unit: IPS)

1) On **Top Plane**, draw a circle which is 0.5 inch in diameter, and with the center of 1.5 inch from the origin point (along plus x-axis direction).
2) On **Right Plane**, draw a rectangle with 1 inch in width and 0.5 inch in height, and with the center of 1.5 inch from the origin point (along the plus y-axis direction).
3) Create a new plane by click **Features**, then **Reference Geometry**, then **Plane**. Choose **Right Plane** as the reference plane, and set **Offset Distance** as 3 inch.
4) On this newly created plane, draw a square with the size of 1 inch by 1 inch, and with the center of 1.5 inch from the origin point (along the minus y-axis direction).

5) On the **Front Plane**, draw a **3 Point Arc**, connecting the bottom point of the Rectangle and the left point of the Circle. To make sure the Arc connects with the Rectangle and the Circle, you need to add the relation of **Pierce** (selecting a point and a curve, and add relation of **Pierce**. Make sure you click the pre-existing sketch before the arc endpoint). Also, make the top part of the arc to be horizontal (you can draw a horizontal construction line, and then set the relation between the Arc and the horizontal construction line to be tangent).
6) On the **Front Plane**, draw the second **3 Point Arc**, connecting the left point of the Circle and the bottom point of the Square. The two Arcs are connected. Again, to make sure the Arc connects with the Circle and the Square, you need add the relation of **Pierce**. Also make the bottom of the second arc to be horizontal.

7) In **Feature**, click **Lofted Boss/Base**. In **Profiles**, select the **Rectangle**, the **Circle**, and the **Square**. Then in **Guide Curves**, select the two **3 Point Arcs**. Then this part is made.

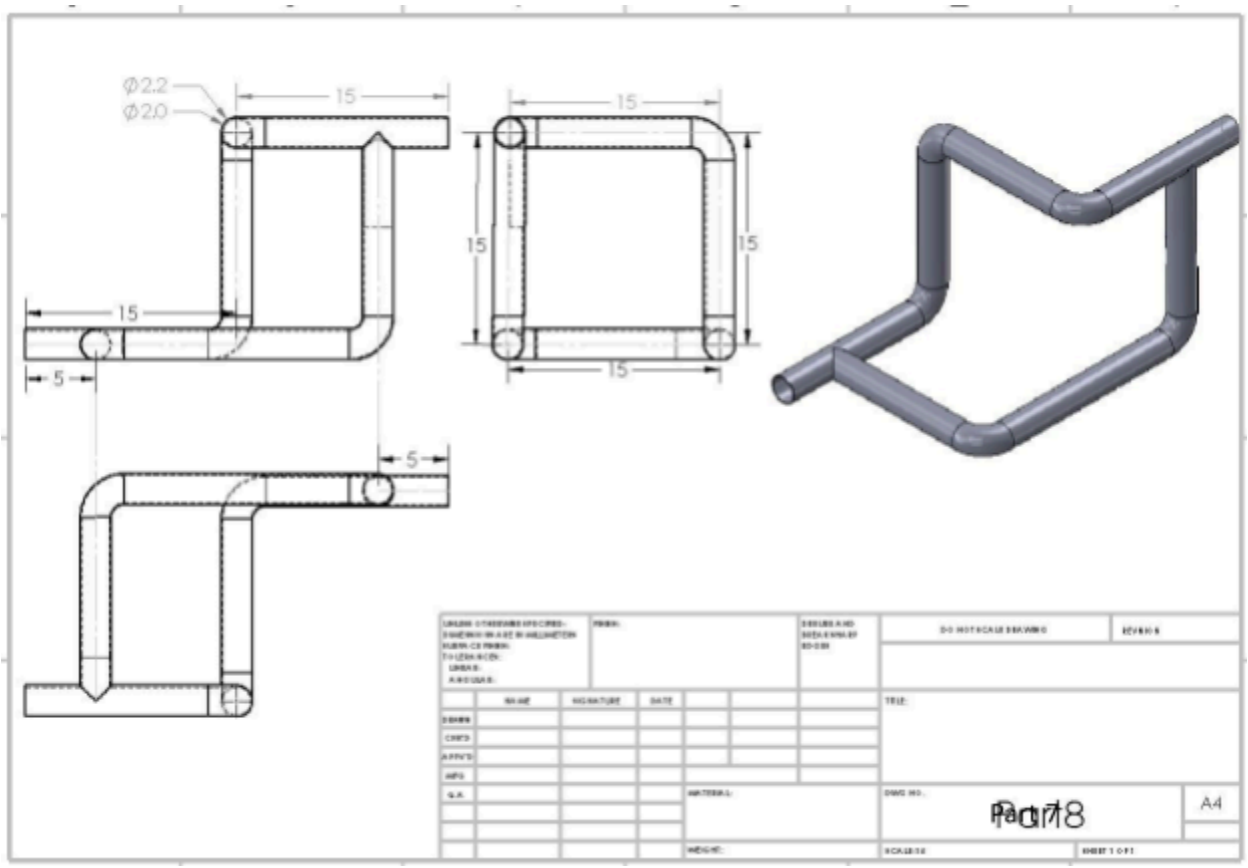
Note: You can edit the Loft feature further. For example, by changing the position of the Connector, you can somewhat "twist" the part.
The connected blue dots are called "connectors". They make it easy to identify the locations on the profiles that SOLIDWORKS is using when interpolating between them. You can drag these connectors to further adjust, and have more control over the geometry produced. If you right click on this preview, and select "Show All Connectors", several more connectors will appear. All of these dots can be positioned independently to adjust the loft results. If you want more control, you can also "add connector".

8) Before finishing, let's set the material as **Titanium Ti-8Mn, Annealed** in **Titanium Alloys**.

Part #7

(This part #7 is relevant to the design project of bypass graft.)

In this practice, we will learn how to do 3D sketch, and how to use the Swept Boss/Base function to develop a pipe.



Drawing for Part 7 (unit: IPS)

First, we want to draw a main pipe which has four straight pipes connected.

1) In **Sketch**, click **3D Sketch**.
   a) Starting from the origin point, make a 3D sketch of four connected lines: the 1st line along x-axis for 15 inch, the 2nd along y-axis for 15 inch, the 3rd line along z-axis for 15 inch, the 4th line along x-axis for 15 inch. Make sure that each line is along the x, y, or z axis, and all the lines are fully defined.
2) Click **Sketch Fillet**, and then select all the three joints to make them smooth curves. Set **Fillet Parameters** as 2 inches.
3) **Exit 3D Sketch.**
4) Click **Swept Boss/Base** in **Features**. In the setting panel, check **Circular Profile**; Make sure the 3D sketch (i.e. the 4 connected lines) is selected as the **Path**; Set the **Diameter** as 2 inch; Set the **Thin Feature** to be 0.1 inch.

Next, we want to draw a bypass pipe, connecting the 1st section (correspondent to the 1st line) and the 4th section (correspondent to the 4th line) of the main pipe.

1) Select the **Front Plane**, draw a horizontal center line, starting from the origin point, and set its dimension as 5 inch. Exit the **Sketch**.
2) Click **3D Sketch**, and draw three connected lines, starting from the right end of the horizontal center line that you just put on the Front plane. For these three connected lines, draw the 1st line along the z-axis, then draw the 2nd line along the x-axis, then draw the 3rd line along the y-axis. Define the length of the 1st line as 15 inch, the length of the 2nd line as 20 inch, and the length of the 3rd line as 15 inch. Make sure this 3D sketch is fully defined, and the end point is at the center of the 4th section of the main pipe.
3) Use **Sketch Fillet** to make the two joints of the three connected lines to be smooth curves with **Fillet Parameters** set as 2 inches. **Exit 3D Sketch.**
4) Click **Swept Boss/Base** in **Features**. In the setting panel, check **Circular Profile**. Make sure the 3D sketch (i.e. the 3 connected lines) is selected as the **Path**; Set the **Diameter** as 2 inch; Uncheck **Merge result** in the **Option** (this is important for using the Split in a following step); Set the **Thin Feature** to be 0.1 inch.

Finally, we need to finalize the two connectors connecting the bypass pipe and the main pipe. Briefly, at each connecting point where two pipes are connected, we need to remove the additional materials (without doing so, the connectors are blocked). There are two options to finish this step:

*1) Using **Split** in the **Feature** (click **Insert**, and **Feature**, and select **Split**):*
Please note that, at each connection, you will need to perform two Split to open both pipes. Start the first Split. In the option panel of Split, use the internal surface of the bypass pipe for **Trim Tools**; Check **Selected bodies**, select **Sweep-Thin1**; Click **Cut Bodies**; Then in the **Resulting Bodies** check **1** only; check **Consume cut bodies**; click **OK** to finish the first Split. After finishing the first Split, start the second Split. In the option panel of Split, use the internal surface of the main pipe for **Trim Tools**; Check **Selected bodies**, select **Sweep-Thin2**; Click **Cut Bodies**; Then in the **Resulting Bodies** check **1** and **2**; check **Consume cut bodies**; click **OK** to finish the second Split. Repeat the above steps for the second connection.

*2) Using **Extruded Cut** of a 2-inch circle centered in the pine:*
Please note that, at each connection, you will need to perform two cuts in order to open both pipes. To make each cut, draw a 2-inch circle centered in the pipe; Then use Extruded Cut with direction along the pipe to remove the additional materials.

Define the materials as **ABS** in **Plastics** before saving the file.