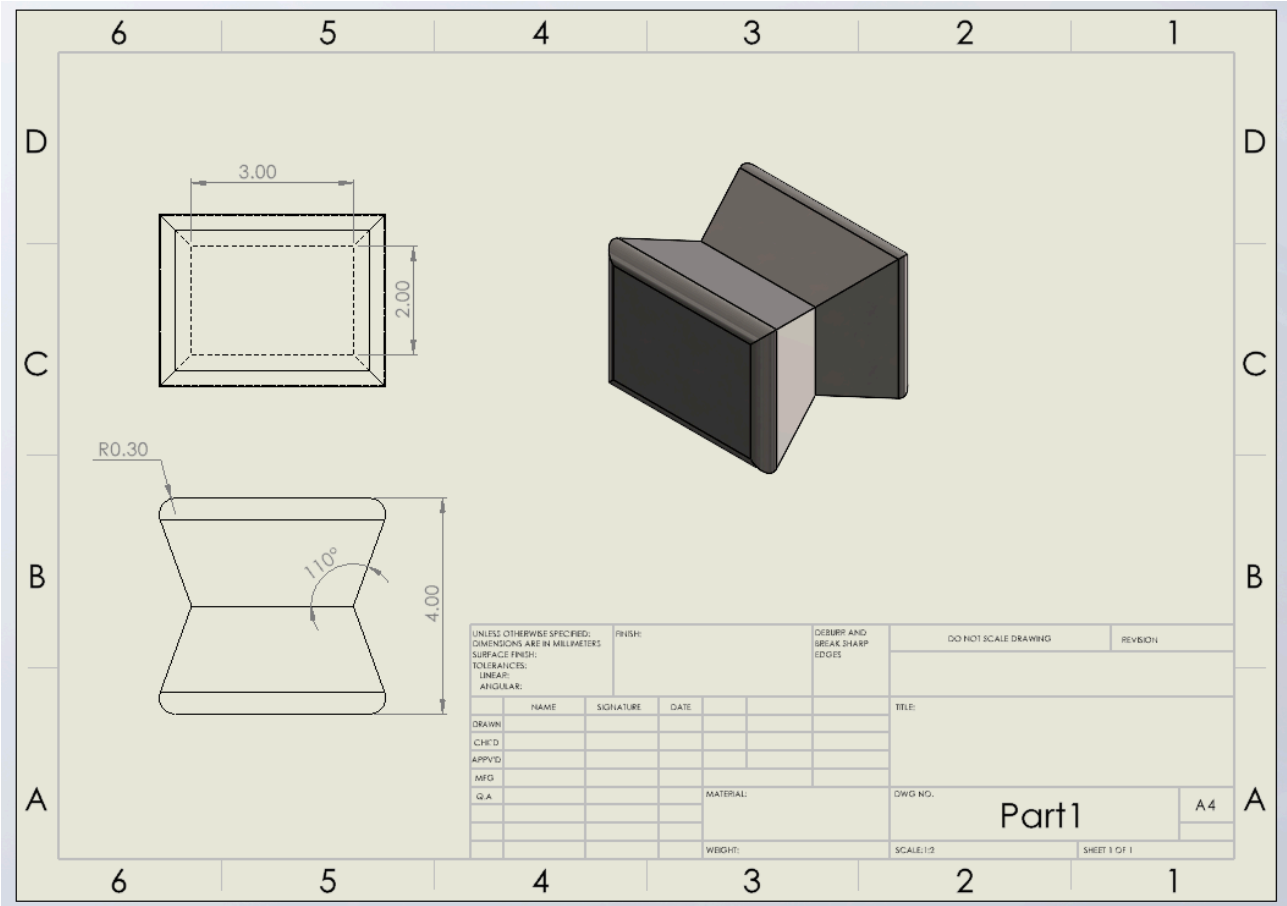
Solidworks Tutorial 1

1. Start **Solidworks**.
2. Click **New**, and then we will be given the options of **Part**, **Assembly**, **Drawing**, and **Advanced**. Click **Part**, then we can select the initial settings including the **Units**.
    a. The largest window is where we can start to draw.
    b. You can choose any units for now, you will be creating a new part (ie. new file) for Part 1.
3. On the left side, we can see menus including **Features**, **Sketch**, **Evaluate**, etc, where the Features and Sketch are the two that we will use the most.
4. Let's select the **Sketch** menu, then click **Sketch**. We will be given the options of three planes: **Front Plane, Top Plane,** and **Right Plane**. Let's start with the **Front Plane** by right clicking it, then click sketch (top left option).

Learning the Solidworks Interface

5. In the center of the plane is the X-axis and Y-axis, as well as the origin.
6. Now pay attention to our mouse cursor. As we can see when the cursor is moving across the plane, we can read out its position by looking at the numbers changing at the bottom of the screen (the 1st one is the position on x-axis, the 2nd one is the position on y-axis, the 3rd one is the position on z-axis).
7. At the bottom of the screen, we can also find the unit used, e.g., **IPS** (inch, pound, second), or **MMGS** (millimeter, gram, second). We always want to check it and make sure that is the unit we want.
8. Start from drawing lines by click **Line** in the **Sketch** menu:
    a. Continuous draw a line until double clicking or pressing **ESC**
    b. If we want to select a line (or other sketches), use the mouse cursor to draw a box on the screen to cover the line (the selected line will change color from dark blue to light blue), then press **Delete**. When we draw the box, we can start from left to right. In this way, we can select an entity only when it is entirely included in the box. We can also draw a box from right to left. This way, any entities touched by the box are selected.
    c. Also try to practice drawing of circles, rectangles, etc.
    d. The wheel of the mouse is for zooming in and out. We can put the mouse cursor at different positions on the screen and then roll the wheel. In this way, we will be able to move our screen and focus on different areas of the part. By pressing and holding the wheel and then moving the mouse cursor, we can rotate the part.
    e. If we want to go back to the original plane, click **View Orientation** (should be at the middle top of the screen) and then click **Normal To**. In general, **Normal To** will put us in a view orientation that is normal to the plane we are working on. The **View Orientation** and **Section View** give us many other options to view the part.
9. **Note:** the **Undo** button, which we might need to use very often, is at the top of the screen.

Task #1 (Part 1)



Drawing for Part 1 (**unit: IPS**)

10. Click **New**, then click **Sketch**, and then select **Front Plane**. Set units to **IPS**.
11. Select **Corner Rectangle**. Then draw a rectangle on the main screen.
    a. Once the rectangle is done, we can see 4 blue lines and 4 green boxes. The green boxes indicate **Relations** (e.g. **horizontal, vertical, midpoint, coincident, tangent**), which define the properties of the sketches and their relative positions, orientations, and angles related to other sketches. For example, if we put the mouse cursor on the top green box, we can see a Relation of "**Horizontal**". When we click and hold this top line, we can move it ONLY along the horizontal direction and cannot tilt. Even if we hold the corner point and move, this line is still always horizontal. We can select this Relation box of "Horizontal" and delete it. Then, when we hold the corner point, we can move its position and tilt the line. This is because the "Horizontal" relation is gone. We can add a relation by first selecting the line and then click "**Add Relations**" → "**Horizontal**".
    b. Now all four lines of this rectangle are all blue, meaning they are **Under Defined**, as can be seen from the bottom line of the sketch window. The rectangle is Under Defined since we haven't defined its position and sizes along the vertical and horizontal directions.
    c. Now let's try to define this rectangle:
12. First, let's define its vertical dimension by first selecting **Smart Dimension**, and then selecting any vertical line. A pop up window allows us to define its dimension. We will set the dimension to **2 inches**.
    a. We can try to select the other vertical line and define it. A window will pop up and ask

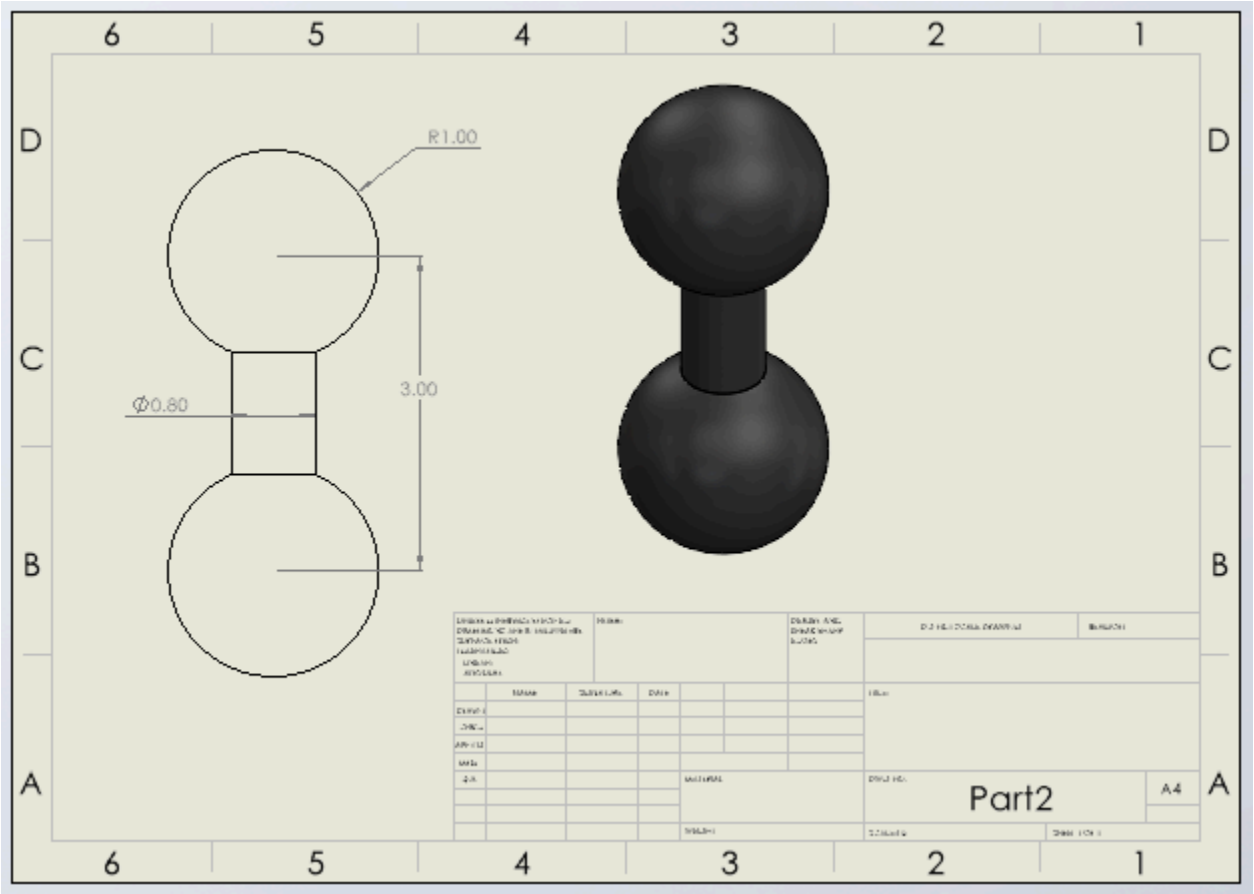
whether we want to “Make Dimension Driven?”. We will be given two options,
    b. “Making this dimension driven”. A driven dimension will only measure the dimension.
    c. “Leave this dimension driving”. A driving dimension can change the dimension. If we select “Driving”, an error will pop up saying “**Over Defined**” which means this dimension has already been defined. This is a very common error for beginners. Deleting this additional dimension will fix this error.
13. Now let’s define the horizontal dimension by clicking a horizontal line and setting the dimension to 3 **inches**. We can see that the rectangle is still blue in color, indicating that it is still “Under Defined”. This is because we haven’t defined the rectangle’s position yet. We can still hold the rectangle and move it around.
14. Let’s first click the **lower-left corner point** of the rectangle, then hold the “**Shift**” button and click the **origin point**. On the left of the screen, a **Properties** window will pop up which allows us to define the relations between these two points selected. We can see these two points in the Selected **Entities**. There is no relation now in **Existing Relations**. We can add one by clicking any options in the **Add Relations**. Let’s add **Coincident** which means putting the two points together. Now we can see that the rectangle is **Fully Defined** (change from blue color into black color). At this time, if we delete any relation, the rectangle will become “Under Defined” again.
15. Click “Exit Sketch”. Now, let’s make the 2D rectangle into a 3D part.
16. Click **Features**, then select **Extruded Boss/Base**. We will see an orange arrow at the plane surface. Use the mouse cursor to drag the arrow and control the extrude direction and length. There are options in the left **Boss-Extrude** window for us to select.
17. In the option window, change from **Blind** to **Mid Plane** so that extrude develops to both sides. For **Depth**, put in **4 in**.
18. Click **Draft On/Off**, and give a **20 degrees** in **Draft Angle** to change the angle during extrusion. Click **Draft outward**.
19. Once we are done, click the **OK** button.
    a. Now our first 3D part is developed!
20. Before celebrating, let’s make it nicer by smoothing the edges and corners.
21. Click **Fillet** which is in the top menu.
22. In the **Fillet** option window, we can select an item by clicking it in the sketch. The item can be an edge or a face. If a face is selected, then all the edges of this face will be processed.
23. Let’s now select both the **front** and the **back** surfaces of this part
24. To select the back surface, use the wheel of our mouse to turn the part over.
25. Selection of the **Full preview** option allows us to see the results in advance.
26. Let’s change the **Radius** in **Fillet Parameters** to **0.3 in.**
27. Click **Ok**. Now a part with nice and smooth edges is generated.
28. We can now define the material of this part. Select **Materials** in the Feature Tree. After right clicking **Materials**, click **Edit Materials**. Select “**Stainless Steel (ferritic)**” in “**Steel**”.
29. Save it as **Part1.sldprt**
30. At the left side of the screen, we can see a **Feature tree**, which includes several features we have just created. We can see the **Boss Extrude**, and the **Fillet** we have defined. For each one we can click it and see the details. We can also redefine each feature by clicking **Edit Feature**. Let’s try:
    a. Click **Boss-Extrude1** in the feature tree, then click **Edit Feature** in the pop up window. Unselect **Draft Outward**, then click OK. Now the whole part changed accordingly, and we don’t have to redo the Fillet since the fillet feature is left intact.
    b. Click **Sketch1** under the **Boss-Extrude1**, then click Edit Sketch in the pop up window. For convenience, we might need to select **Normal To** for the **View Orientation** so that we

can better edit the Sketch in the front plane. We can change the dimension from original **3 in** to **8 in**. Then click **OK**, and then click **Exit Sketch**. We will find that the whole part changes accordingly, and we don't need to redo the Boss-Extrude and Fillet.

Drawing 1

31. Click **New**, and select **Drawing**.
32. In the "Standard sheet size", select "A4 (ANSI) Landscape". To see this option, we might need to uncheck the **Only show standard formats**.
33. In the **Part/Assembly to Insert**, click **Browse** and find the Part1 we have just created and saved.
34. Now we can display the part along different orientations on the sheet, including front, top, bottom, left, right, back, and isometric. We don't really need all the orientations to show the drawing of a part. Let's display this part along three orientations: **Front**, **Bottom** and **Isometric**.
35. For each view, we can change **Display Style**. In the front view, let's change the style to **Hidden Lines Visible**. For the isometric view, let's change the style to be **Shaded With Edges**. We might need to uncheck **Use parent style.** We can change the **Scale**, e.g. use **Custom scale 1:2**. We can also apply a **User Defined** scale.
36. Now click the **Annotation**, and click **Smart Dimension** and mark all the detailed dimensions on this drawing.
    a. For this part 1, we need mark the following dimensions:
    b. Rectangle size: **2 in** by **3 in**;
    c. The Boss Extrude Length: **4 in**;
    d. The Boss Extrude Draft Angle: **20 degree** (shown as **110 degree** in the drawing)
    e. The Fillet Radius: **0.3 in**.
37. Once all the dimensions are marked, right click **Sheet Format** in the feature tree, and add part name, your name, units, and date.
38. **Save the drawing as Part1.slddrw.**

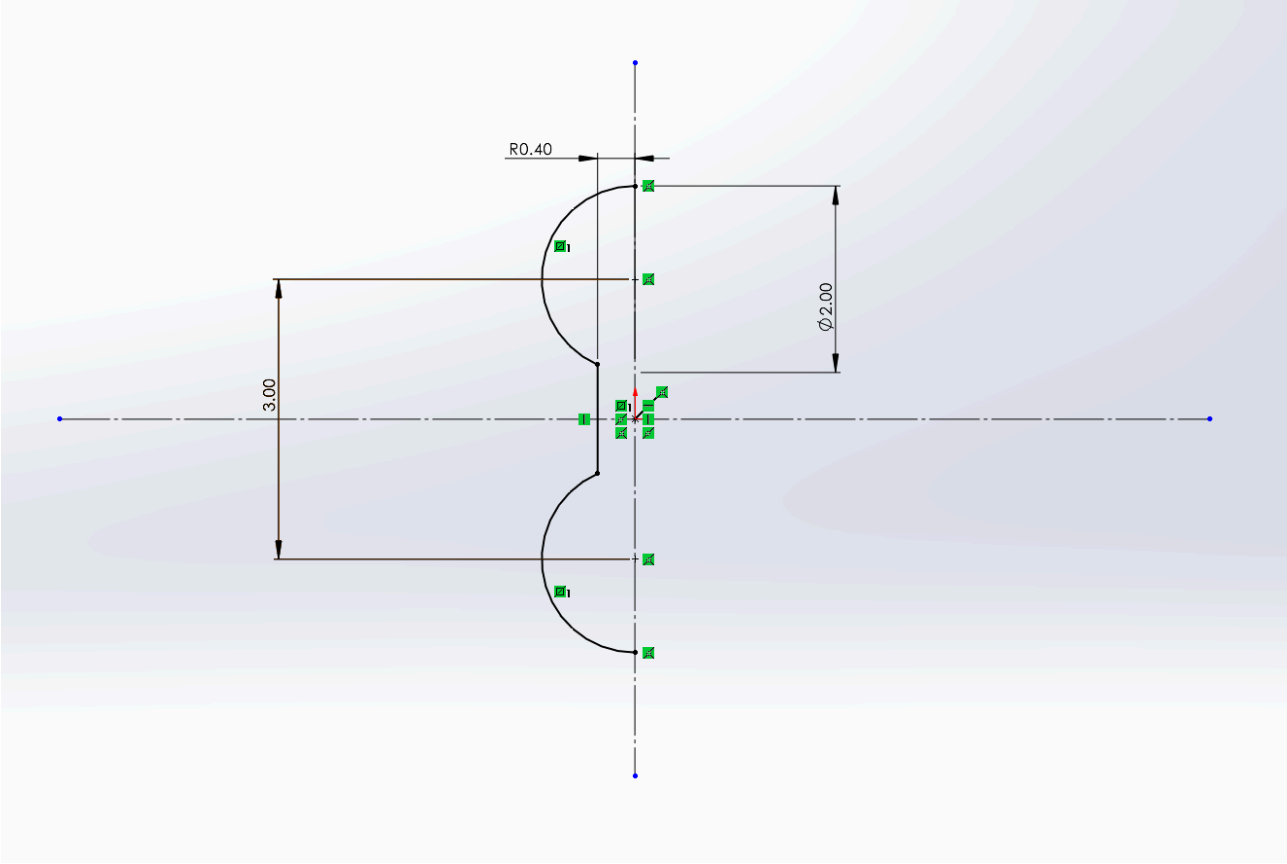
Task #2 (Part 2)



Drawing for Part 2 (**unit: IPS**)

39. Part 2 looks like a dumbbell. We can see the dimensions in the drawing above. There are two balls which are symmetric. The Radius of each ball is **1 inch**. The distance between the centers of the two balls is **3 inch**. The two balls are connected by a cylinder which has a diameter of **0.8 inch**.
40. Make sure the unit is IPS. Start the **Sketch** using the **Front Plane**.
41. Before drawing anything, let's first draw two **Centerlines**, one vertical and one horizontal. We can find **Centerline** in the drop-down menu of **Line**. Make sure the two center lines pass through the origin point (make sure the relations of **Coincidence** & **Parallel** or **Vertical** are assigned to each line. If not, we can add these relations).
42. Draw two **Circles**, both with the center on the **Vertical Centerline**.
43. Use **Smart Dimension** to define the diameter of the upper circle as **2 inches.**
44. Since the two circles are identical in shape, instead of defining the dimensions of the lower circle, we will make the lower circle and the upper circle **symmetric**.
    a. Select the upper circle by clicking it, press and hold the "Shift" button and, at the same time, select the horizontal central line, then keep the "Shift" button held down and select the lower circle. Now, the "**Properties**" window will pop up at the left. We can see three Entities are selected, including two arcs (upper and lower circles) and one line (horizontal central line). Then add a relation of "**Symmetric**". This step will make the two circles identical in shape and also symmetric along the horizontal central line. We can find that any change (e.g. the position) of one circle will automatically affect the other circle.

45. Using Smart Dimensions, define the distance between the two circles as **3 inches.** Now the two circles are fully defined.
46. At the left of the vertical center line, draw a vertical line connecting the two circles. Make sure it has a relation of **Vertical**.
47. Define the distance between this line and the vertical center line to be **0.8/2 inch = 0.4 inch**.
    a. Now, we need to turn the 2D sketch into a 3D part by performing the **Revolved Boss/Base** of the 2D sketch around the vertical center line.
48. In the next step, let's delete lines that we don't need for revolving (otherwise, the sketch will "cut" itself when revolving around the vertical center line).
    a. We will click **Trim Entities** in the top menu, and then select **Power Trim**. **Power Trim** allows us to remove any unneeded curves/line segments by clicking and dragging over them with the cursor. It will remove a line and stop at the first intersections. After this step is done, the sketch should look like

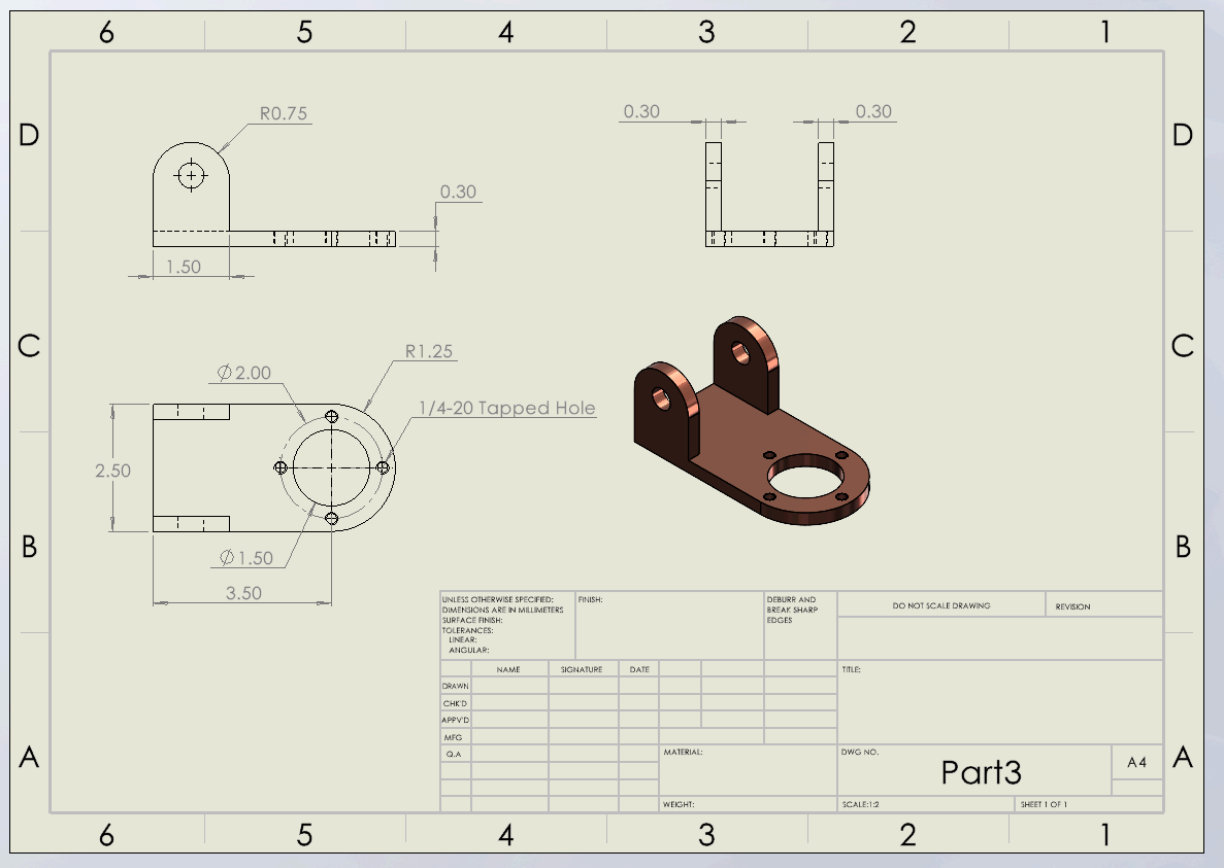


    b. Now, this 2D sketch is still "open", and we need to make it into a closed loop before performing **Revolved Boss/Base**.
49. Draw an additional vertical line to connect the top and the bottom points. Now this sketch is self-contained (closed). If we don't do this step, a warning will pop up when we perform **Revolved Boss/Base**.
50. Click **Features** → **Revolved Boss/Base**.
    a. In the Revolve option window, you need to select the axis of revolution, which is the vertical center line. Leave Direction1 as **Blind** and **360 deg**.
    b. **Thin Feature** should NOT be checked (thin feature is for making a shell instead of a solid part). Before clicking the OK button, we already can preview the result from revolving.
51. We can now define the material of this part. Select **Materials** in the Feature Tree. After right

clicking **Materials**, click **Edit Materials**. Select “**Gray Cast Iron**” in “**Iron**”.

52. Click **OK**, then we have our second part. Save it as **Part2.sldprt**

Task #3 (Part 3)

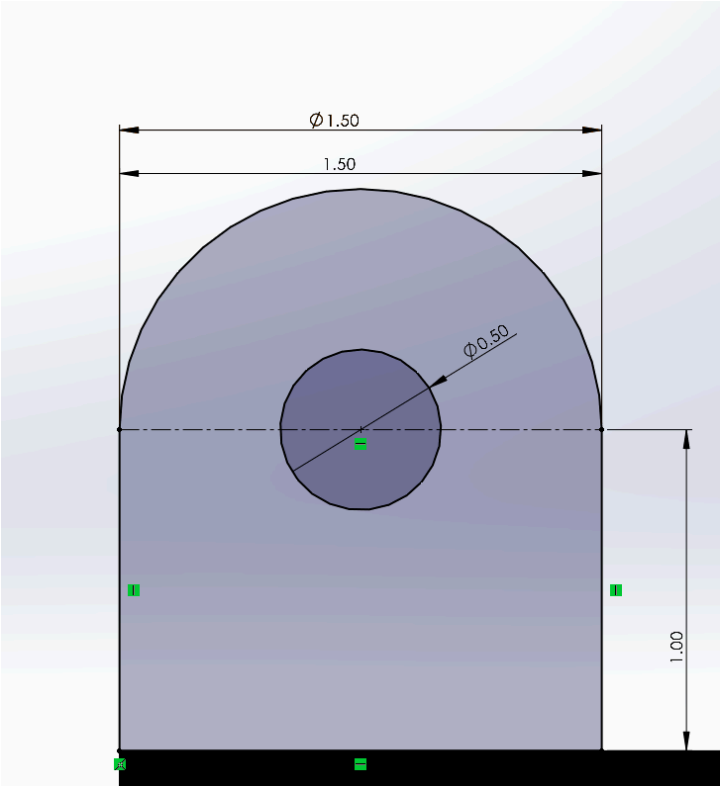


Drawing for Part 3 (**unit: IPS**)

53. For this part, please consider which surface should be treated as a base, or in other words, which plane we want to start with. My suggestion is to start from the sketch shown in the lower-left view in the drawing above (base). The reason is that the projection of this part on this top plane has the most complicated features. After finishing the base, we will then start to develop the two "wings", and then the four holes on the base.
54. Make sure the unit is IPS. Start the **Sketch** by using **Top Plane**.
55. Draw two concentric circles, centered at the **origin point**. Use Smart Dimensions to define their diameters as **2.5 in** and **1.5 in**, respectively.
56. Draw the three remaining lines that define the outline of the lower left view. Pay attention to all the **relations**, especially the two **Tangent** relations at the two points where the lines touch the large circle. If the Tangent relations are not automatically added, add the relation by first selecting both the line and the circle, and then add the relations.
57. Define the lengths of the lines. Here only the length of the horizontal lines needs to be defined (**3.5 in)**. We don't need to define the length of the vertical line since it is equivalent to the diameter of the circle. If we try to define it, you will see an error of "over defined". Remove unneeded lines using **Power trim**.
58. Make this 2D sketch into a 3D part by using **Extruded Boss/Base**. Set the **Depth** as **0.3 in.**
59. We now need to consider how to develop the next Feature (the two wings). We need to remember that we always start from a 2D sketch and then extrude or revolve it into a 3D feature. We first need to decide on which plane we will draw the 2D sketch.
60. On the base that just developed, select any of the two side planes by clicking it, and then in the

pop up menu click **Sketch**, then press the **Normal To** in the **View Orientation**. Now whatever we draw will be on this plane that was selected.

61. Now draw a rectangle, and define its width as **1.5 in** and height as **1.0 in**.
62. Draw two circles on top of the rectangle, both concentric with the midpoint of the top line of the rectangle. Make sure a relation of **Midpoint** is there. Otherwise, you need to add this relation. Define the diameters of these two circles as **1.5 in** and **0.5 in**, respectively.
63. Next, we want to extrude this 2D sketch into a 3D feature along the depth. Before that, we need to trim the unneeded lines using the **Power Trim**. After trimming the top horizontal line of the rectangle, we may find that the 2D sketch is **Under Defined** again. The reason is that, after trimming the top horizontal line, the **"midpoint relation"** associated with this line is also removed. Therefore, the two circles with their positions fixed by this midpoint relation become "Under Defined".
64. So to make the 2D sketch fully defined, we have to add additional relations. There are many ways to do so. One way is to add a construction **Centerline** at the position of the top horizontal line which is just trimmed. If every step is correct, we should see the following sketch on the screen.

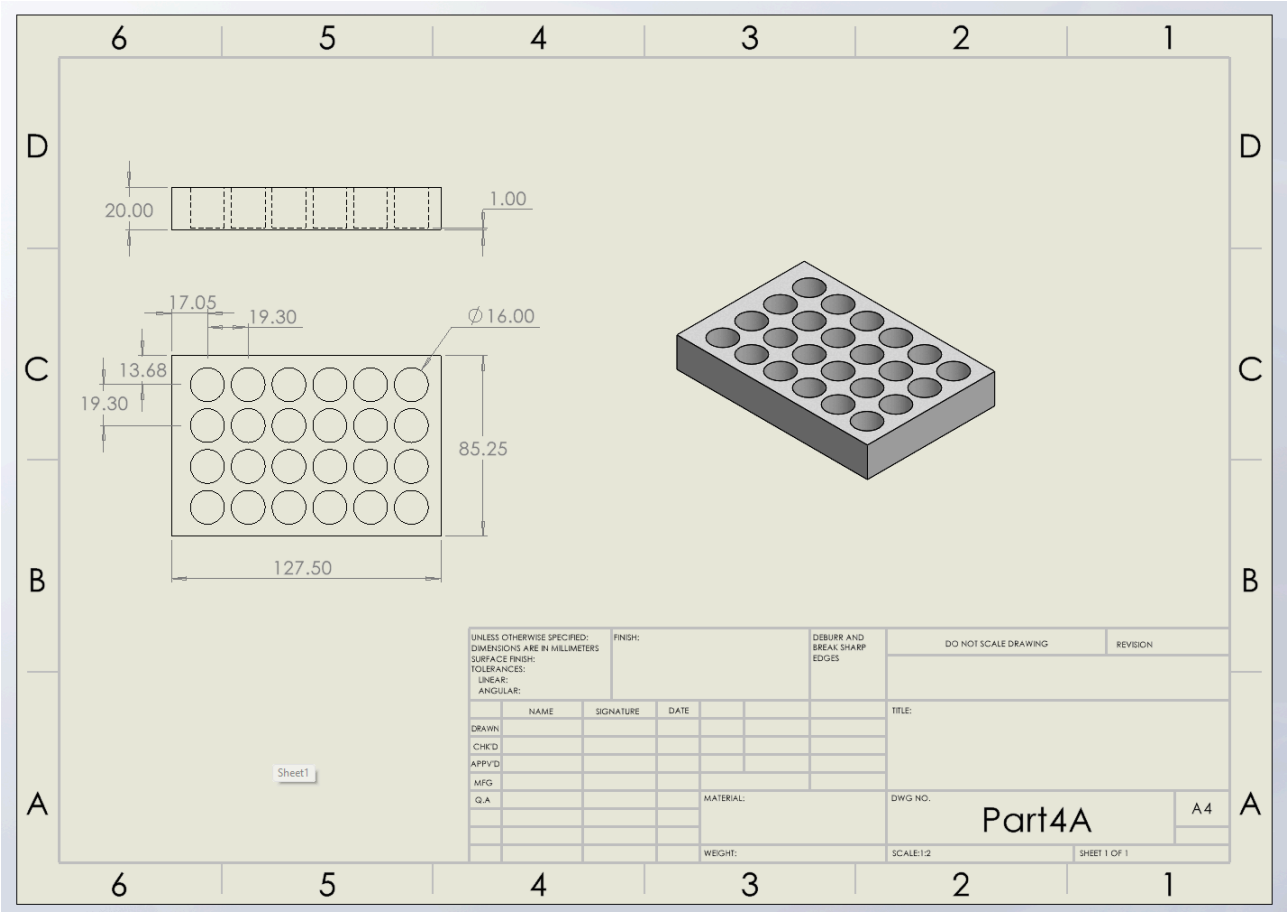


65. Select the 2D sketch we just created, click **Extrude Boss/Bass** to make a sketch from 2D to 3D by pulling the arrow, or by setting in the Boss-Extrude window (check **Reverse Direction**, and set the **Depth** as **0.3 in**). Now the first wing is made.
66. To make the second "wing", we can repeat the above steps. A much better way is to use the feature of **Mirror**.
67. Click **Mirror** at the center of the top menu. Set Mirror options:
    a. For **Mirror face/plane**, select **Front Plane** (the front plane could be found in the feature tree next to the mirror option window. We may need to open the feature tree to see it.) For **Features to Mirror**, make sure we select **Boss-Extrude2** which can also be found in the feature tree.

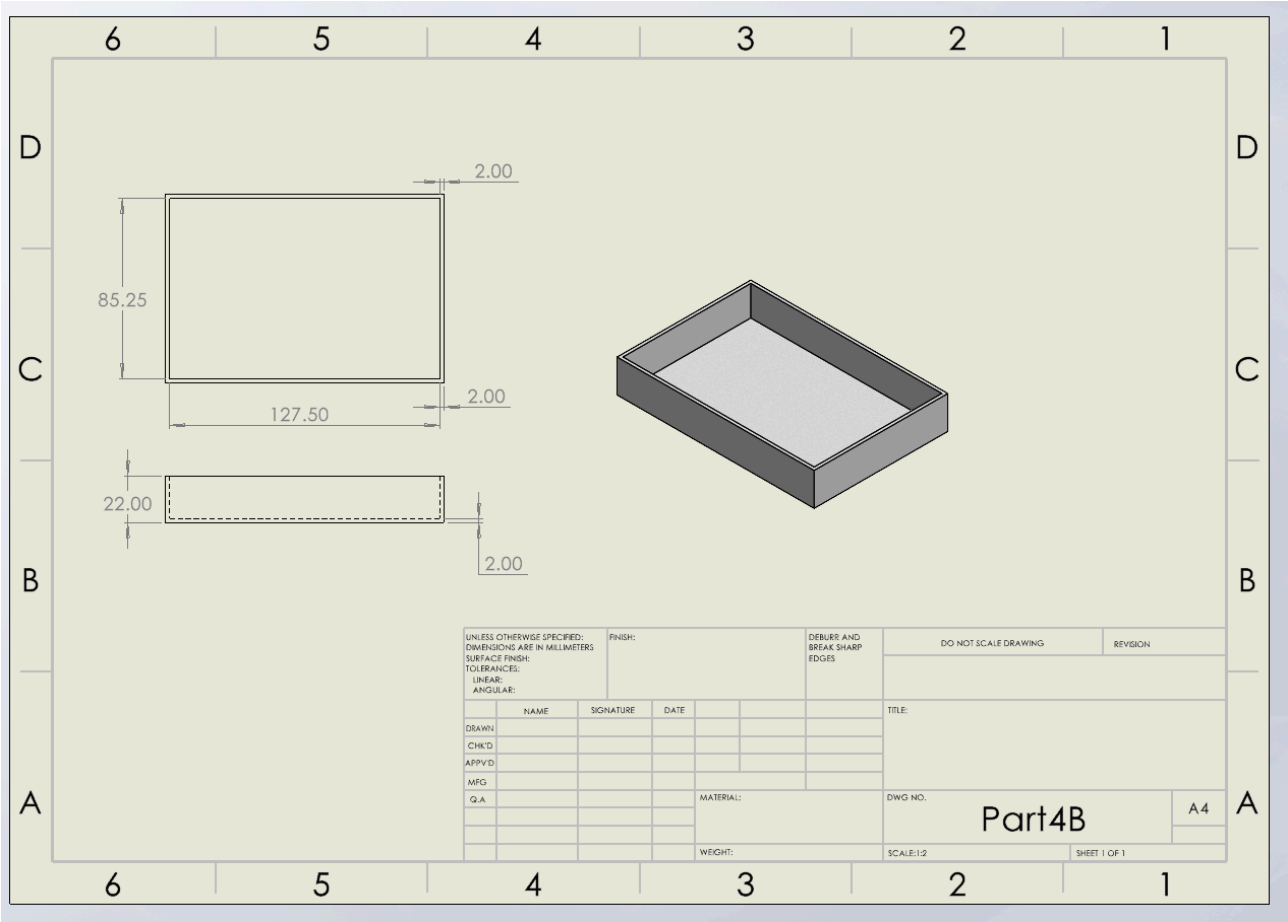
    b. Then click **OK**. Now the second wing is made.
    c. The advantage of using **Mirror** is that when we change the sketch of the first wing, the second wing will change accordingly.
    d. Next we need to develop the 4 holes on the base. The 4 holes are located on a circle with a diameter of **2 inch**, and the circle is concentric to the origin point.
68. Select the top surface of the base which will be the plane for drawing the 4 holes.
69. Draw a circle concentric to the origin point. Define its diameter as **2 in**. In the Circle option window on the left, check the option of "**For construction**" to make this circle a construction sketch (a construction line is a virtual dashed line for helping construction only).
70. Draw a vertical reference line and a horizontal reference line by adding one **vertical Centerline** and one **horizontal Centerline**, both passing through the origin point.
    a. The above two steps give us the positions for the centers of the 4 holes.
71. **Exit Sketch**, and select **Features**. On the top menu, select **Hole Wizard**. All options for holes are listed in the **Hole Specification**.
    a. In the **Hole Type**, select **Straight Tap**
    b. In the **Size**, select **1/4-20**.
    c. Set **End Condition** as **Through All**
    d. Uncheck the option of "**Near side countersink**"
72. Click **Position** and click the top surface of the base where we want to place the holes. Then we will see our mouse cursor become a yellow circle.
73. Select the 4 points where we want to drill the holes.
74. We may find that the cosmetic threads in the holes are not visible. To make the threads visible, click **Options**, then click **Document Properties**, then click **Detailing**, then check **Shaded cosmetic threads**. Then click **OK**.
75. Select **Materials** in the Feature Tree. After right-clicking **Materials**, click **Edit Materials**. Select "**Copper**" in "**Copper Alloys**".
76. Click **OK**, then we have our third part. Save it as **Part3.sldprt.**
77. We have just built a part with many layers of features. We can see each feature by sorting the feature tree which includes Boss-Extrude1, Boss-Extrude2, Mirror1, and 1/4-20 Tapped Holes. We can change each feature by selecting **Edit Feature** or **Edit Sketch**.
78. **Continue to Task 4 on the next page.**

Task #4 (Parts 4A & 4B)

79. In this task, we will build two parts (Part 4A & Part 4B), which are a standard 24-well plate and a matching cover. After the two parts are developed, we will then assemble them together.



Drawing for Part 4A (unit: MMGS)



Drawing for Part 4B (**unit: MMGS**)

Develop Part 4A:

80. Start a New part.
81. Make sure the unit is in **MMGS**
82. Start **Sketch** and select the **Top Plane**.
83. Draw a **Center Rectangle**, centered at origin. Define its dimension as **127.5 mm** by **85.25 mm**
84. Click **Features**. Select **Extruded Boss/Base**. Set extrude **Depth** as **20 mm**. Click **OK**
85. Select the top surface of the part, and click **Sketch** to work on this surface. Click **Normal To** so that we can better sketch on this surface.
86. Draw the first circle close to the top left corner. Define its diameter as **16 mm**, the distance from the left edge as **17.05 mm**, and the distance from the top edge as **13.68 mm**.
87. Click **Linear Sketch Pattern** in the top menu, then work in the Linear Pattern option window: First, in **Entities to Pattern**, select the circle by clicking it.
    a. In **Direction 1**, set **Spacing** as **19.3 mm**, set **Number** as **6**, check **Dimension X spacing**. In **Direction 2**, change the **Number** to **4** first, then set **Spacing** as **19.3 mm**, also check the Reverse **Direction**, and check **Dimension Y spacing**, and check **Dimension angle between axes**. Click **OK**
88. Above steps will duplicate the circle into a 4 by 6 pattern, with a constant distance of 19.3 mm between the adjacent circles. Please note that the checking of **Dimension X/Y spacing** and **Dimension angle between axes** are very important; otherwise, the pattern will be left as "Under Defined".

89. Click **Features**, then click **Extruded Cut** in the top menu
90. In the Cut-Extrude option window, set **Depth** as **19 mm** (note: it is not a Through All cut)
91. Select **Materials** in the Feature Tree. After right-clicking **Materials**, click **Edit Materials**. Select "**ABS**" in "**Plastics**".
92. Save it as **Part4A.sldprt**

Develop Part 4B:

93. Start a **New** part
94. Make sure the unit is in **MMGS**
95. Start **Sketch** and select the **Top Plane**.
96. Draw a **Center Rectangle**, set the size as **127.5 mm** by **85.25 mm**
97. Click **Extruded Boss/Base**, set the **Depth** as **20 mm**. Click **OK**.
98. Click **Shell** at the center of the top menu. In Shell option window,
    a. Set **Thickness** as **2 mm**
    b. In **Faces to Remove** box, select the top face of the part (the top face will open)
    c. Check **Shell outward** (increase the outside dimensions of the part)
    d. Check **Show preview**
    e. Click **OK**
99. Select **Materials** in the Feature Tree. After right-clicking **Materials**, click **Edit Materials**. **Select** "**ABS**" in "**Plastics**".
100. Save it as **Part4B.sldprt**

Assemble Part 4A and Part 4B.

101. Click **New**, and then click **Assembly**
102. In **Begin Assembly** window, click **Browse** and then select both **Part 4A** and **Part 4B** for the **Part/Assembly to Insert**
103. Put both **Part 4A** and **Part 4B** on the screen, side by side. Right click **Part 4A** in the feature tree and select **Fix**.
104. Click **Mate** in the top menu
105. For assembly, the feature used most is **Mate**. In the Mate option window, we can see many options, such as Coincident, Parallel, Perpendicular, Tangent, Concentric, and Lock. The mates used most include **coincident** and **concentric. Coincident** can mate the two surfaces together. **Concentric** is usually used to mate two holes.
106. or the **1st Mate**, select the **top surface of Part 4A**, and then select the **interior top surface of Part 4B**.
107. Once we click the **interior top surface of Part 4B**, a menu will pop up. Make sure we flip Part 4B upside down by clicking **Flip Mate Alignment** in the pop up menu.
108. Click **OK** in the Mate option window to finish the **1st Mate**
109. For the **2nd Mate**, mate one **side surface of Part 4A** and one **interior side surface of Part 4B** together by selecting both surfaces, also by using **Coincident** mate. We may need to rotate Part 4B before we can see its interior side surfaces.
110. Once we click **OK**, we can see that these two surfaces are now aligned.
111. For the **3rd Mate**, mate another **side surface of Part 4A** and another **interior side surface of Part 4B** together by selecting both surfaces, again by using **Coincident** mate. Click **OK**.
112. Now Part 4A and Part 4B are perfectly assembled. Save the assembly as **Part4.sldasm**
113. The basic concept of the above steps is to mate three pairs of surfaces of Part 4A and Part 4B, so that their relative positions are fixed.
114. To better visualize each part in the assembly, we can select the parts in the **Feature Tree**, then either **Hide** them, or change their **Transparency**.