

Introduction to Autodesk Fusion 360

In this document, we will learn how to create a simple 3D part that looks like:



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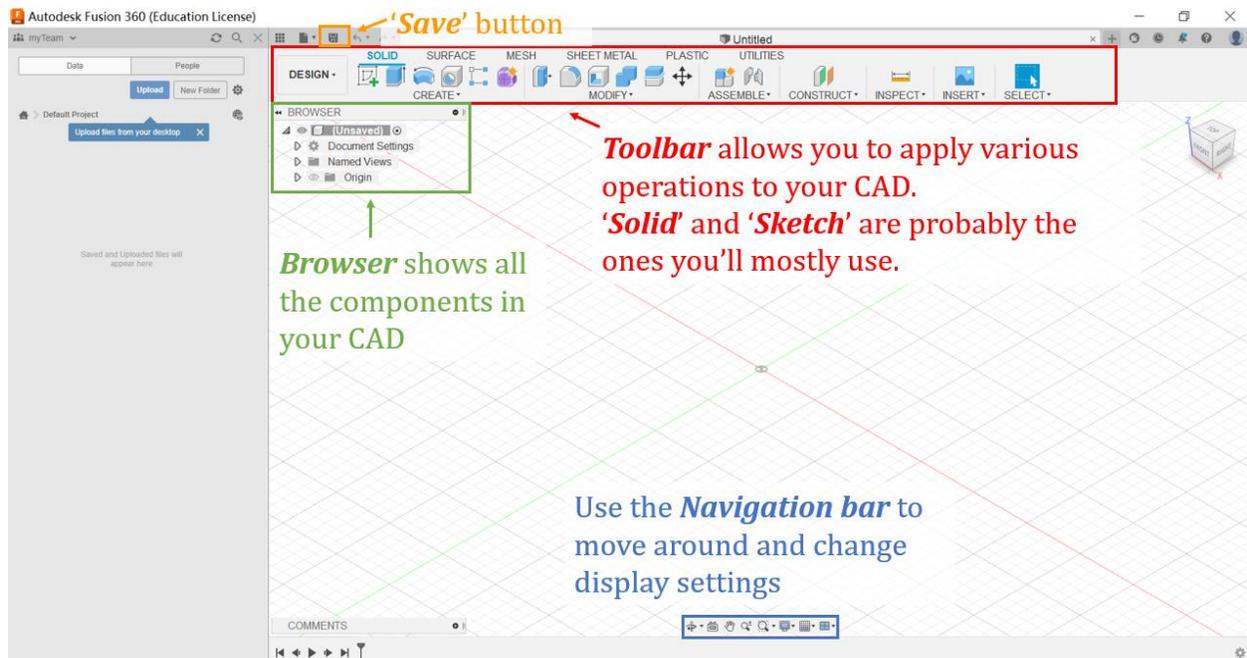
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Introduction

Welcome to Fusion 360. First, let's get familiar with the program interface:



Use your mouse to move around:

- Scroll wheel = zoom in or out
- Shift + hold scroll wheel = orbit
- Left click = select

For a more comprehensive explanation of the interface, visit:

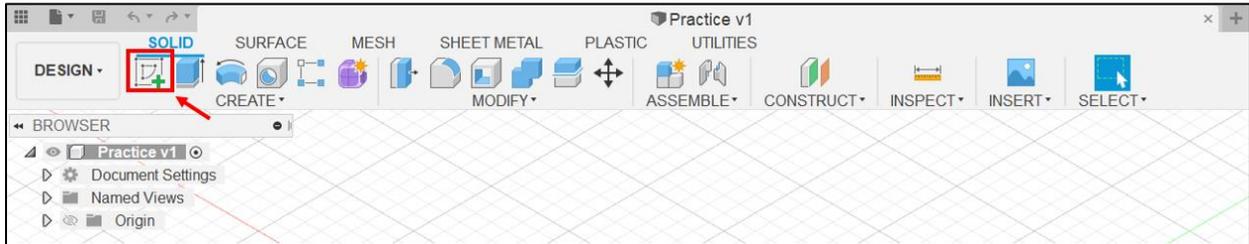
<https://help.autodesk.com/view/fusion360/ENU/?guid=GUID-E647CA56-7187-406A-ACE4-EAC59914FAE4>

Sketch

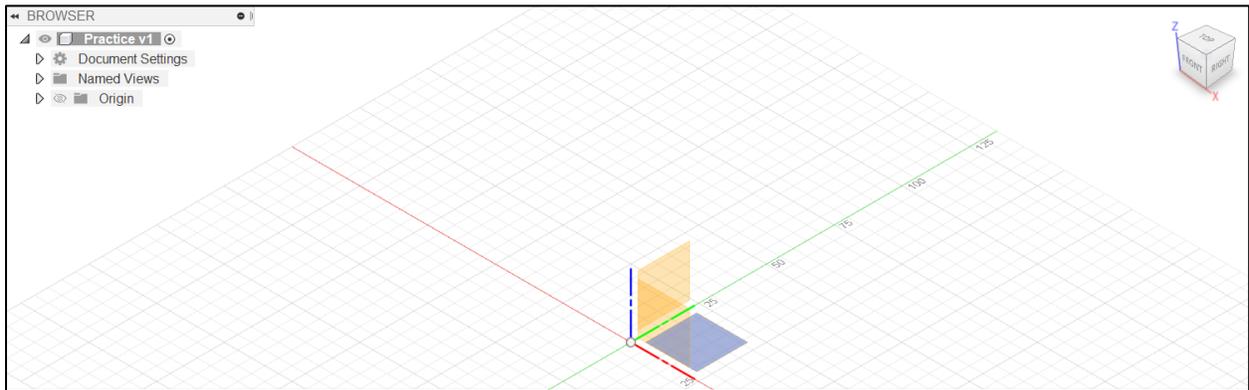
Sketch the big circle

By the end of this document, you will learn how to make a simple 3D object that looks like the one on the cover page.

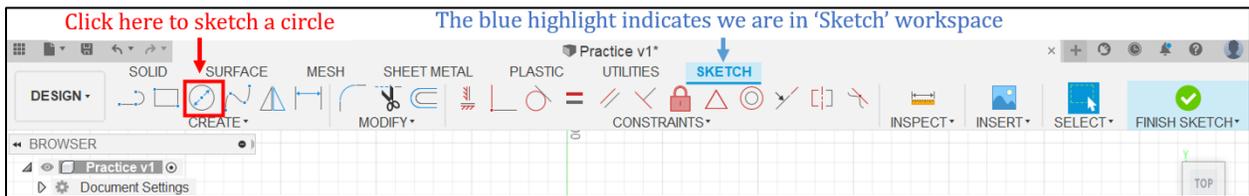
To make a 3D object, start with a 2D sketch by clicking on 'Create Sketch' icon:



Select a plane to sketch onto. In this example, I selected the XY-plane:



Now we are in the 'Sketch' workspace. To sketch a circle, click on 'Centre diameter circle' under Sketch -> Create:

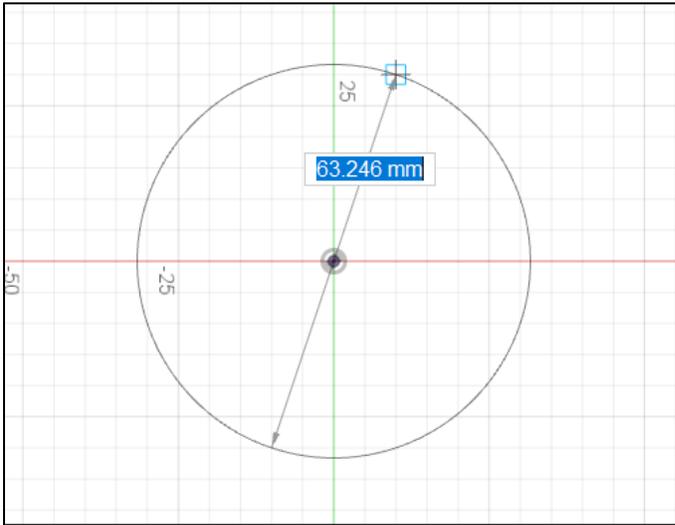


(After clicking on it, the icon gets highlighted in blue, and a small circle appears next to the cursor:

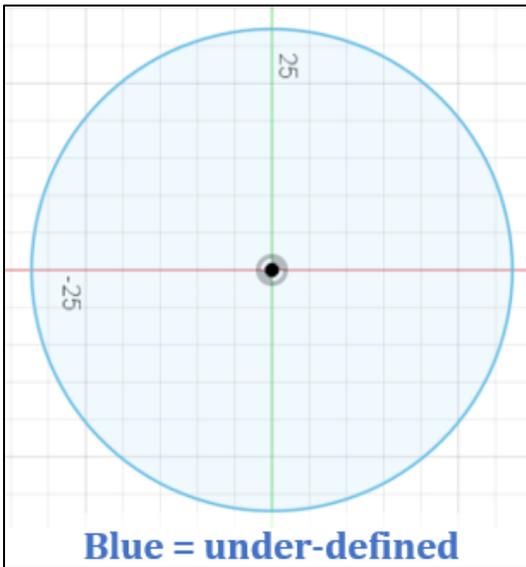


This indicates you are about to sketch a circle. If you clicked the wrong icon or wish to exit, press ESC)

Click anywhere (for example, the origin) to specify the center of the circle, and then enter its diameter:



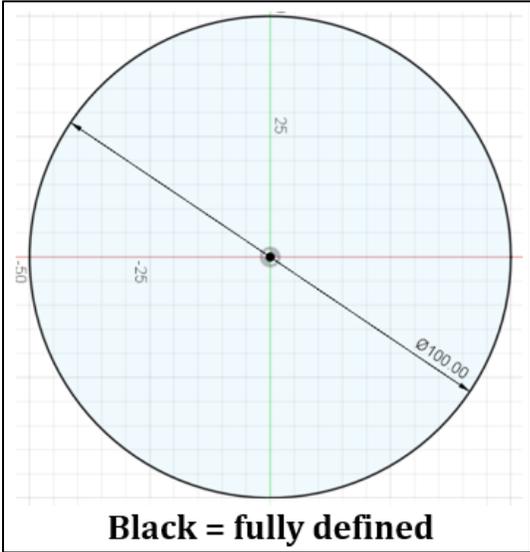
However, you will notice the boundary of the circle is blue, which means the circle is still **under-defined**. The diameter of the circle has not yet been fixed:



To fix the diameter of the circle, click on 'Sketch Dimension' icon or press D:

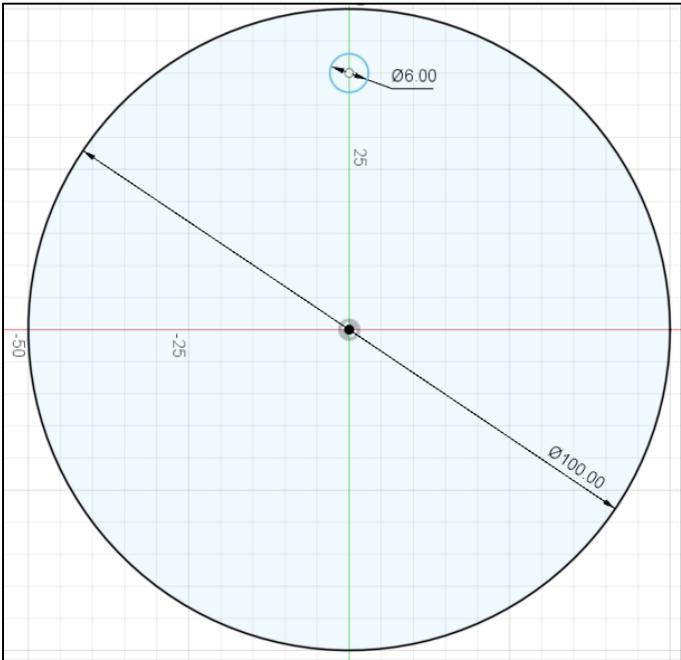


Click on the circle and enter its diameter. After finishing, you will notice the boundary of the circle now has black color, which means the circle is **fully defined**.

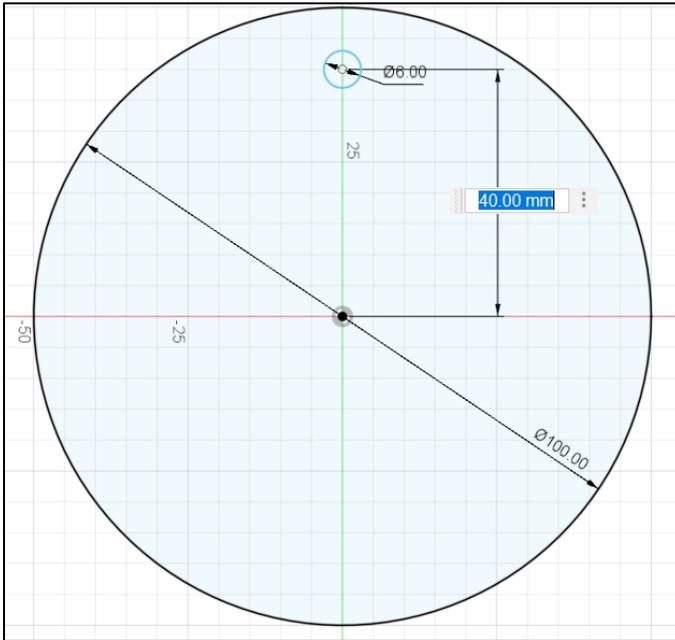


Sketch the small circles

Next, let's draw the three small circles. Again, click on 'Centre diameter circle' to sketch a circle and 'Sketch Dimension' to define its diameter (say for example, 6 mm):



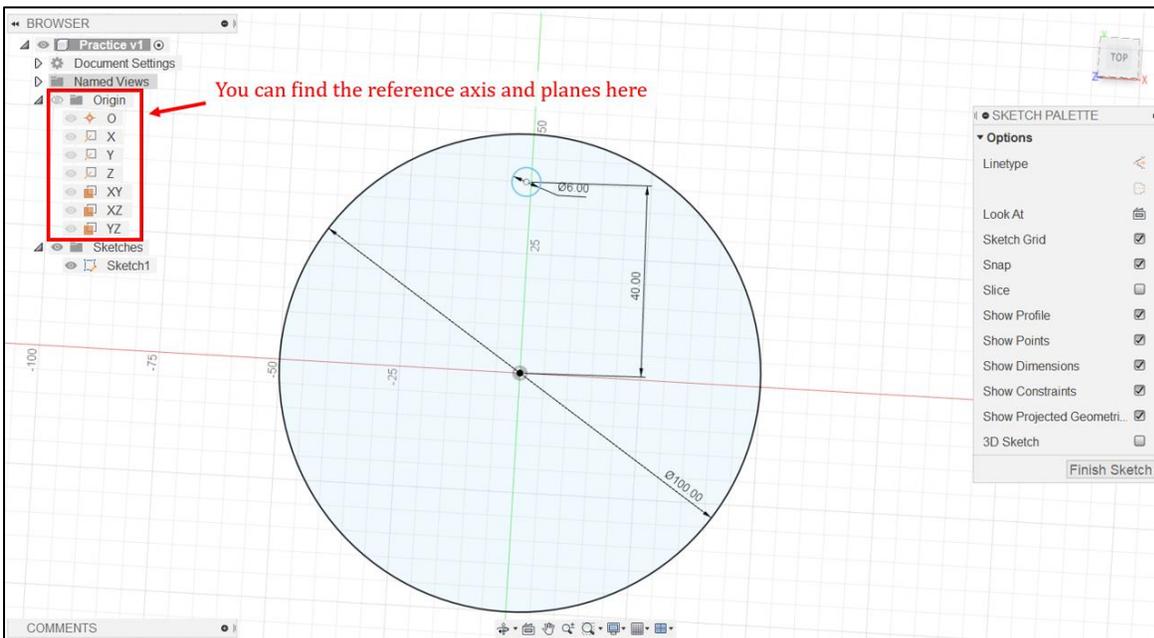
Use 'Sketch Dimension' again. Click on the center of the small circle and the large circle to define the distance in between (say, 40 mm, for example):



Finally, fully define the location of the small circle by using 'Coincident' under **Constraint**:

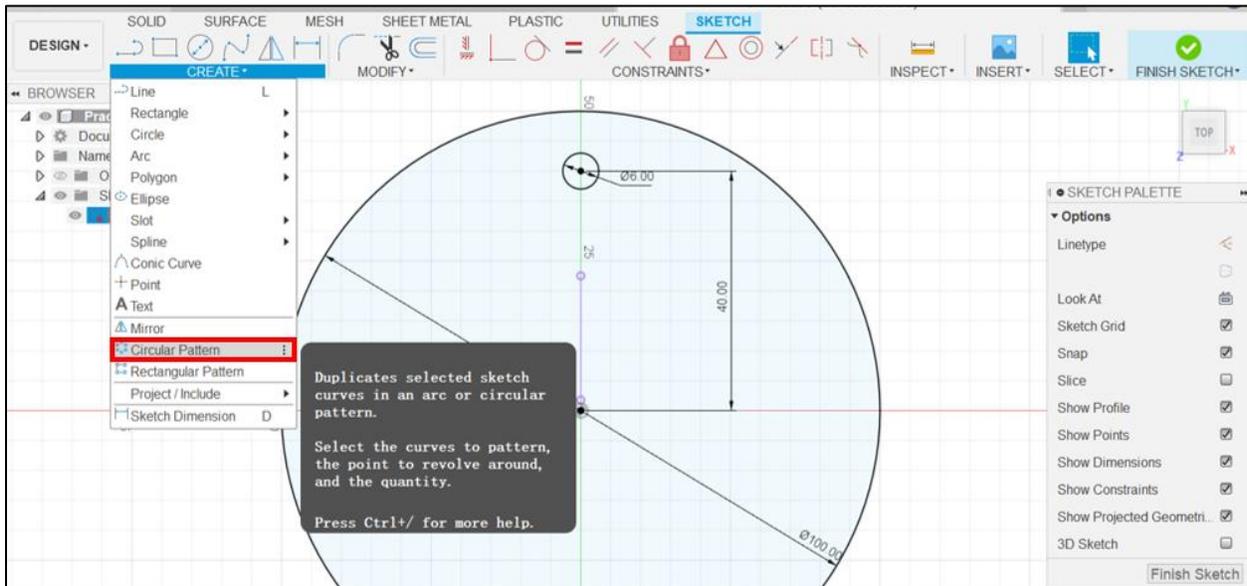


Click on the small circle and the y-axis to make them coincident:

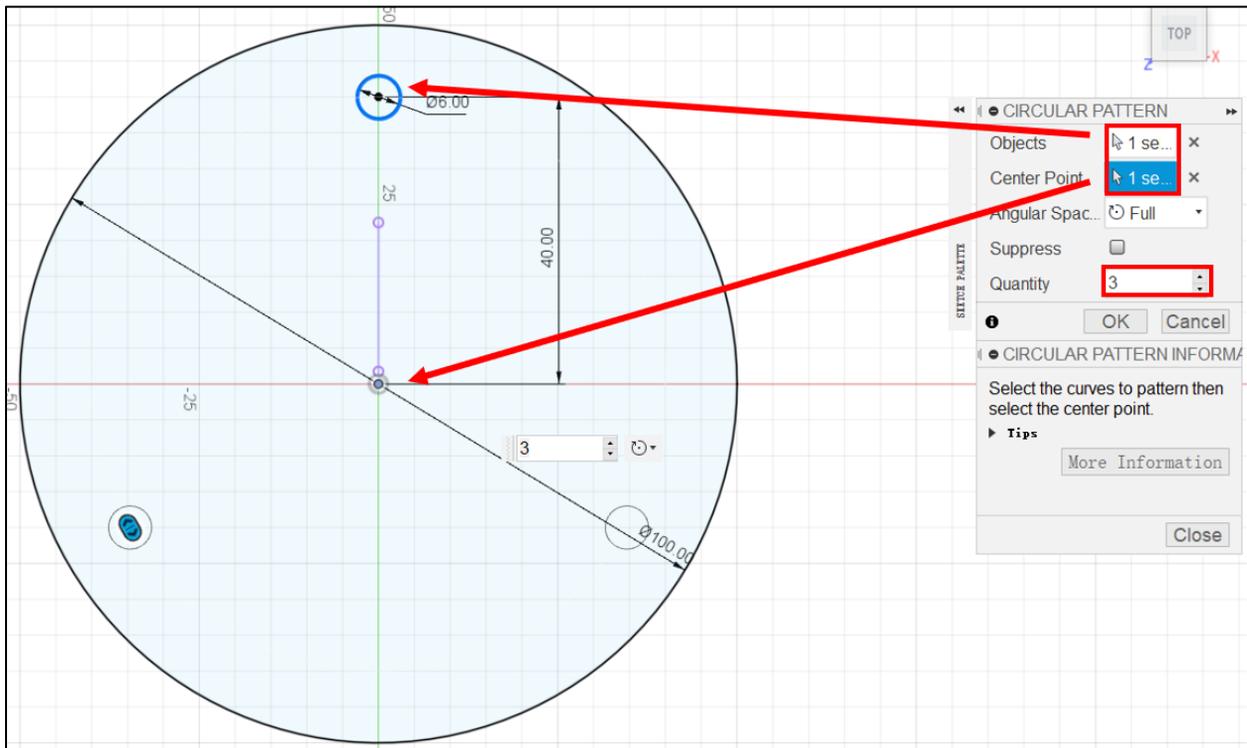


The small circle now should be fully defined.

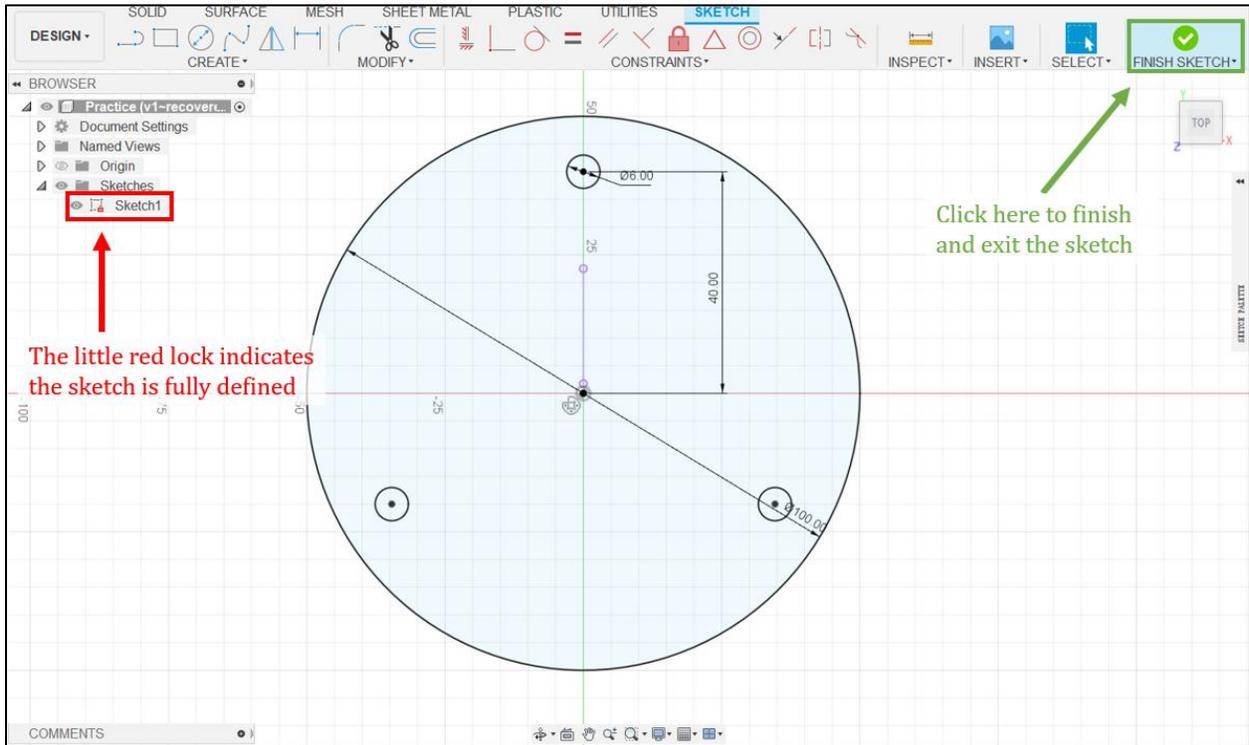
To create two more small circles 60 degrees to each other, use 'Circular Pattern' under Create:



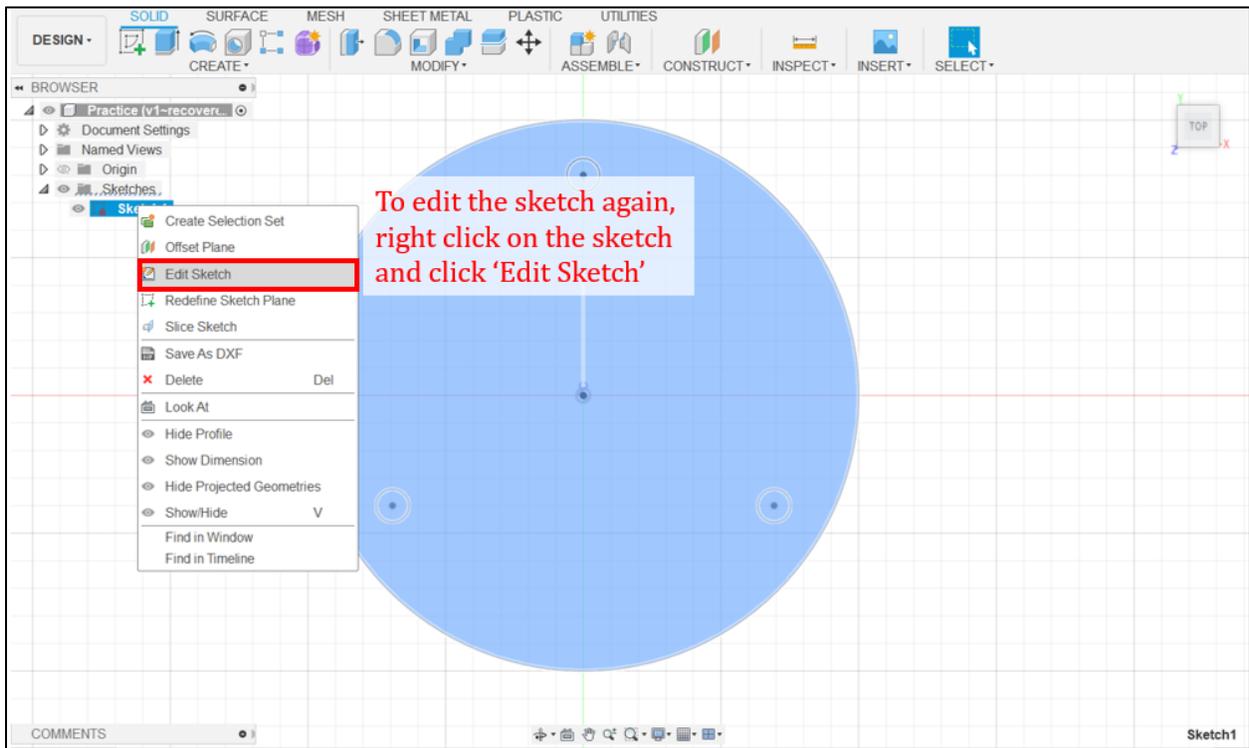
On the 'Circular Pattern' menu, select the small circle as 'Objects' to be duplicated, and select the centre of the big circle as 'Centre Points'. Enter '3' for 'Quantities'. Click on 'OK' to finish.



Now the 2D sketch is done! This sketch will be the foundation of the 3D part. Click on 'Finish Sketch' to finish and exit the sketch:



You can always go back and re-edit the sketch:

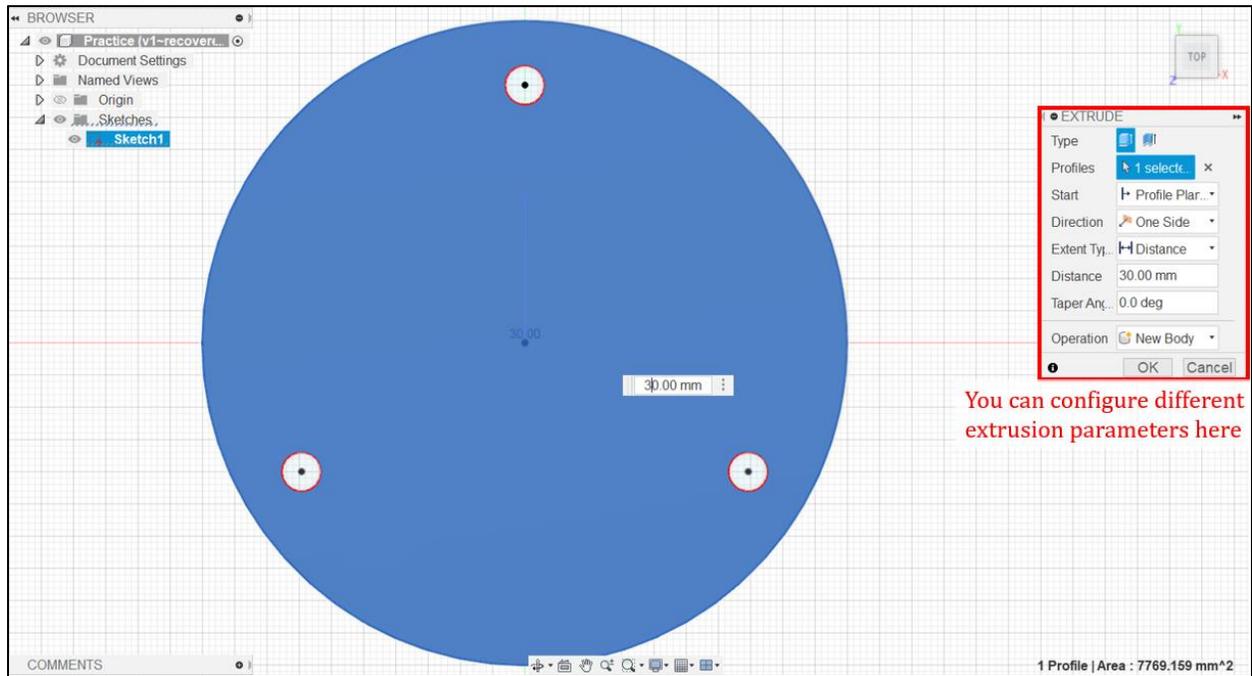


Extrude (Make a New Body)

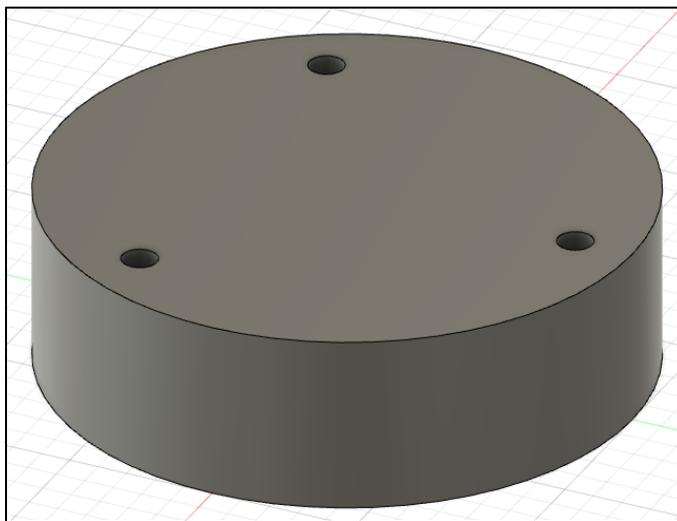
Now let's make it 3D. Click 'Extrude' under Solid tab:



Select the sketch we just made, and specify a thickness (for example, 30 mm):



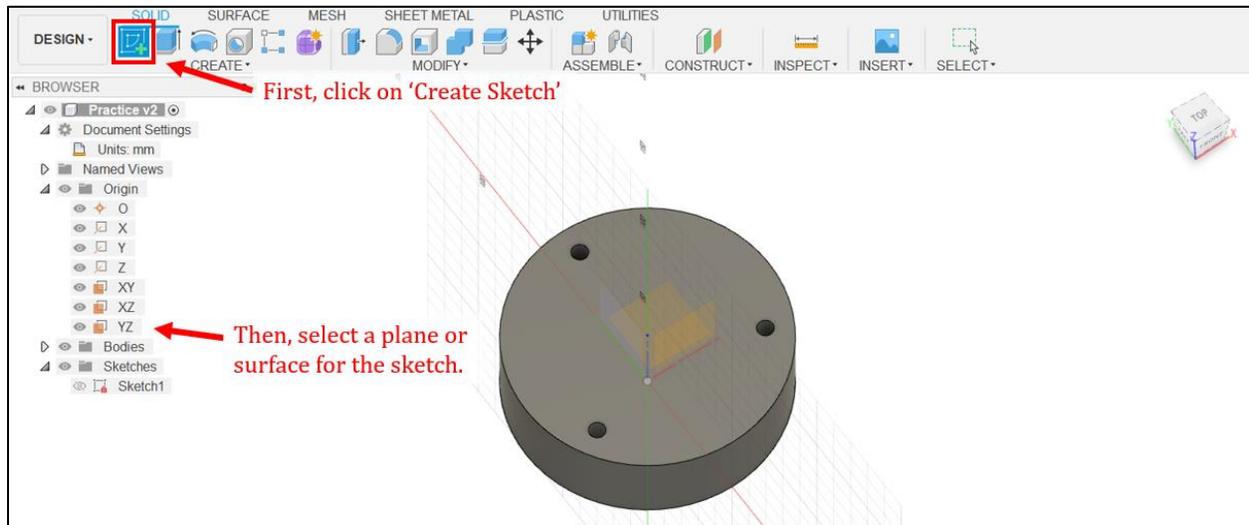
Click 'OK'. Now we have a 3D part extruded from a 2D base!



Extrude (Cut)

In this section, we are going to remove materials from the 3D part. It is very similar to what we have done.

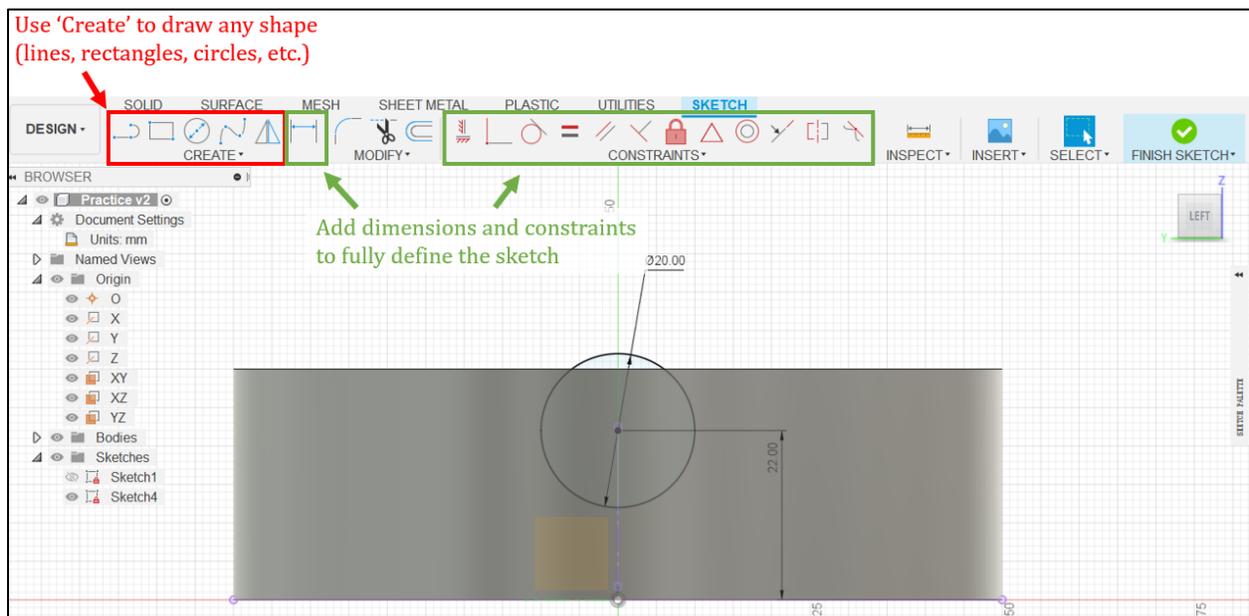
First, create a new sketch by clicking on **'Create Sketch'** and select the YZ plane.



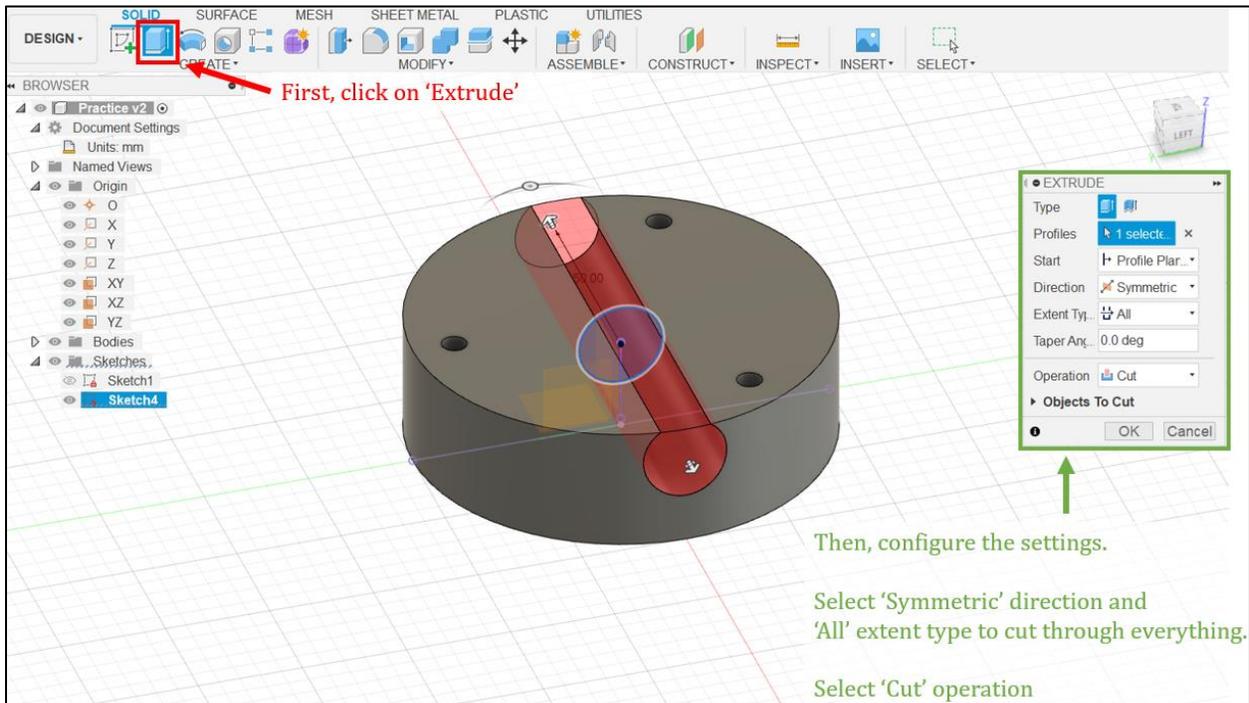
Use:

- **'Centred Diameter Circle'** to draw a circle
- **'Sketch Dimension'** to specify the cut-out circle diameter and its distance from the bottom of the 3D body
- **'Coincident'** to make the center of the circle to stay on the z-axis

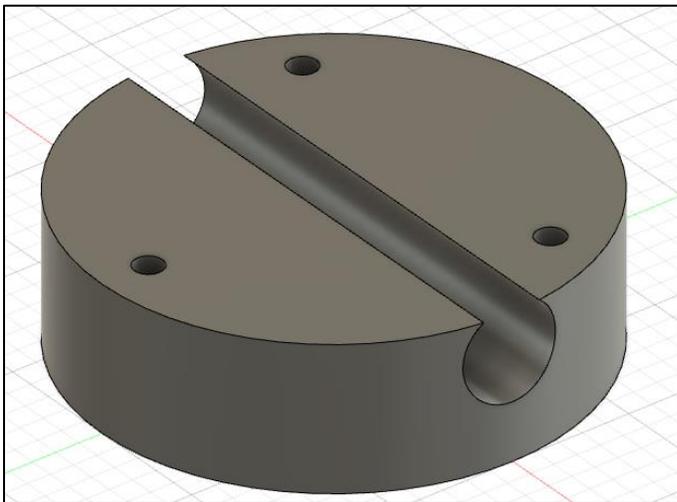
(there is no unique way of making a sketch or part, you can try different functionalities to see which works the best for you):



Finally, use 'Extrude' again. But instead of using 'Create a New Body', select 'Cut' operation:



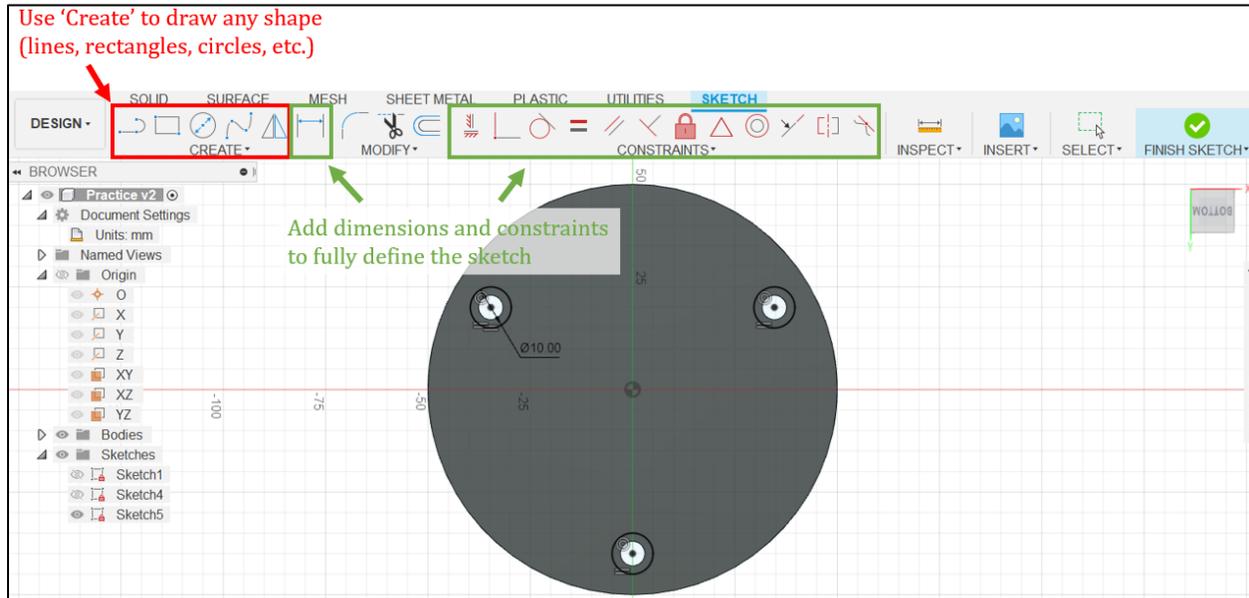
Click on 'OK' to implement the extrusion cut:



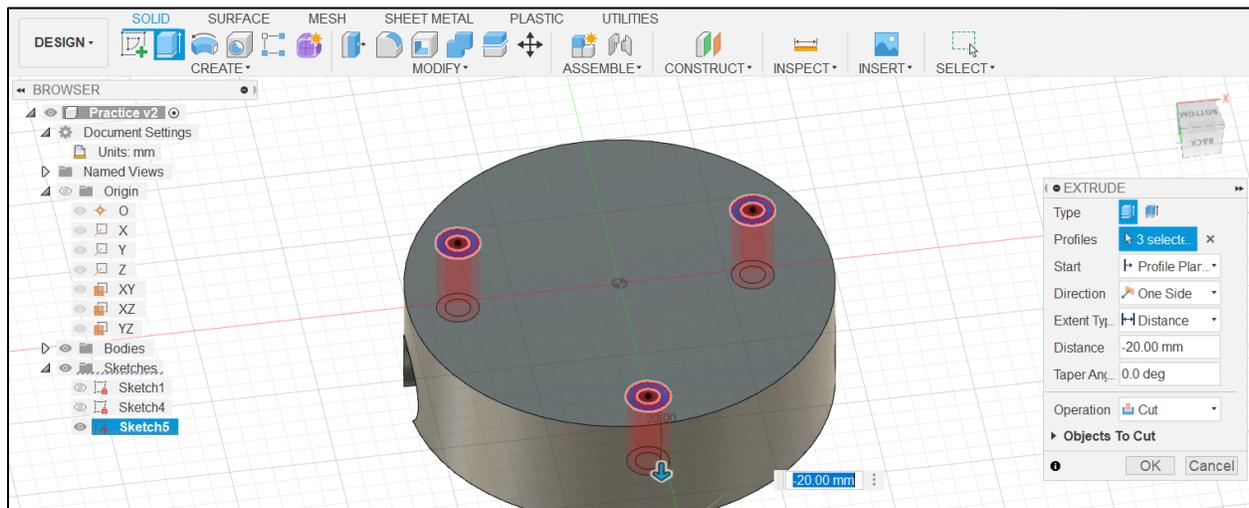
The larger three holes on the bottom can be made using extrude-cut in a similar fashion.

First, create a new sketch on the bottom surface of the 3D body.

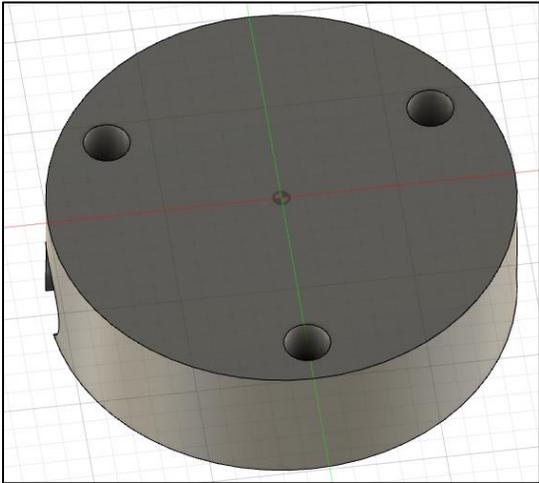
Next, sketch the three circles. You can use 'Circular Pattern' again. Alternatively, you can draw three circles separately, apply 'Concentric' to constraint the circles with the existing ones and use 'Equal' to make them have the same diameter.



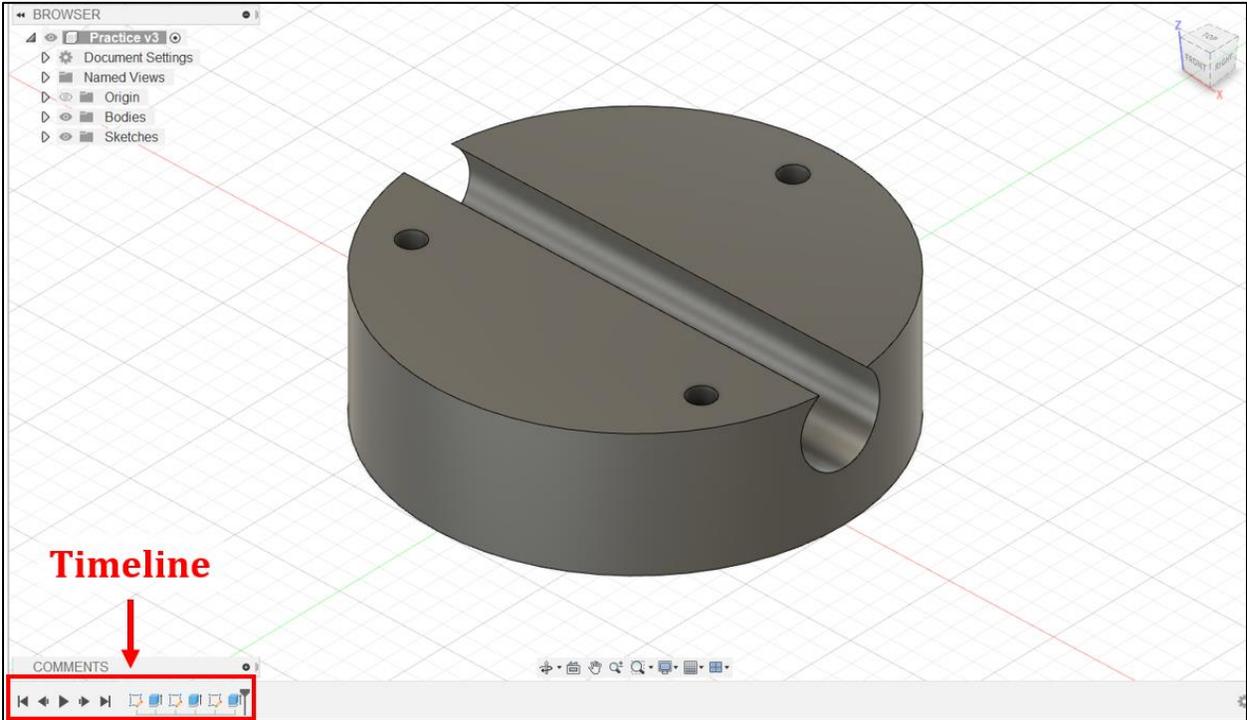
Use 'Extrude' to cut out the holes:



Click 'Ok', and we are finished!



All the operations you have done are listed as a **Timeline** on the bottom left of the screen. You can right click them to edit or delete:



Resources

For a more detailed written tutorial, visit Fusion 360's website:

<https://help.autodesk.com/view/fusion360/ENU/?guid=SKT-3D-SKETCH>