

*Sign-In*



# ONSHAPE WORKSHOP

*Neha Muvvala*

01.

# INTRODUCTION

*Why CAD?*



# WHAT IS CAD?

C

COMPUTER

A

AIDED

D

DESIGN

A way to digitally create 2D sketches and 3D models of products before they are manufactured. It is used to increase the quality and optimize a design.

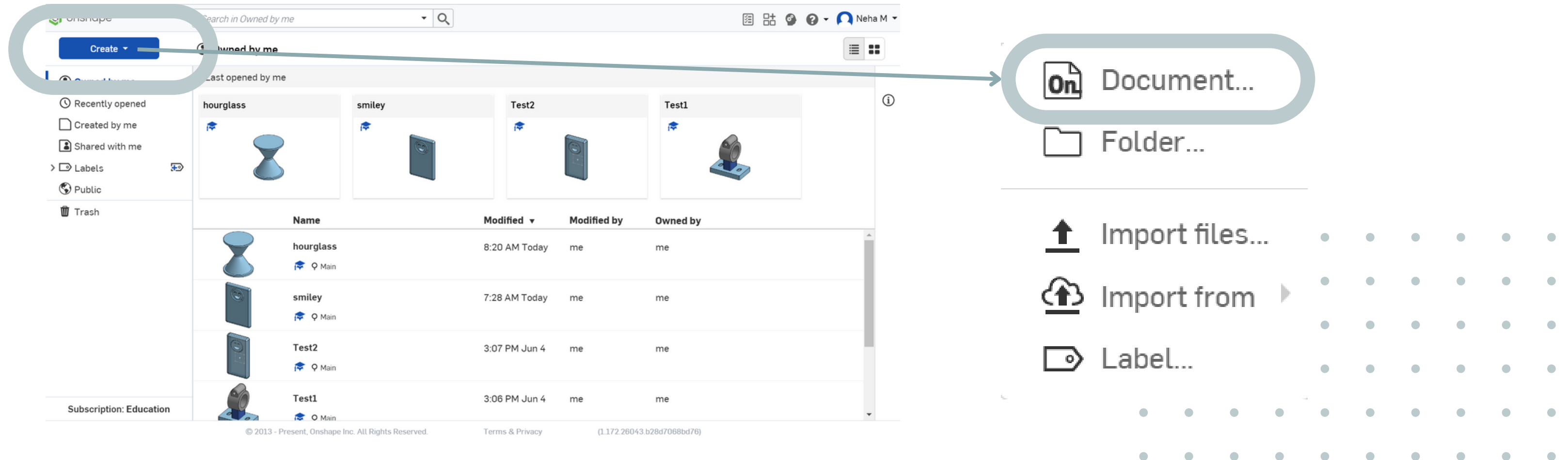
## WHY?

Medical devices are expensive to create. Manufacturing devices before testing their viability is a waste of time and resources. Using CAD to design, test (simulations), and modify products before they are actually manufactured.

# OPEN UP ONSHAPE

Go to [onshape.com/en/](https://onshape.com/en/) and sign in. If you have not made an account yet, click "Create a Student Account" and input your information to do so.

## CREATE A NEW DOCUMENT



The screenshot shows the Onshape web interface. The 'Create' button is circled in red. An arrow points from this button to a dropdown menu that is also circled in red. The dropdown menu contains the following options:

- Document...
- Folder...
- Import files...
- Import from
- Label...

Name	Modified	Modified by	Owned by
hourglass	8:20 AM Today	me	me
smiley	7:28 AM Today	me	me
Test2	3:07 PM Jun 4	me	me
Test1	3:06 PM Jun 4	me	me

Subscription: Education  
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# PLATFORM

Make sure to NAME your document

The screenshot shows the Onshape web application interface. At the top, the title bar reads "onshape" followed by "Untitled document" and "Main". A "Share" button and user profile "Neha M" are on the right. Below the title bar is a toolbar with various icons for sketching and modeling. On the left is a sidebar with a "Features (4)" section containing a filter and "Default geometry" options (Origin, Top, Front, Right). Below that is a "Parts (0)" section. The main workspace shows three intersecting planes labeled "Front", "Top", and "Right". To the right of the workspace is a "Different Views" panel showing a 3D isometric view of a cube with axes X, Y, and Z, and a vertical toolbar with icons for different view types (isometric, top, front, etc.).

Annotations with arrows point to the following elements:

- Sketch Features:** Points to the toolbar icons.
- Create new sketch:** Points to the "Sketch" icon in the toolbar.
- Sketch tree (all sketches, features, and geometry will be stored here):** Points to the "Features (4)" sidebar.
- Different Views (can use to rotate):** Points to the 3D view panel.
- More views (Isometric is mainly used):** Points to the view type icons.
- Different Planes (can start sketches or add sketch features):** Points to the "Front", "Top", and "Right" planes in the workspace.

02.

# LET'S GET STARTED!

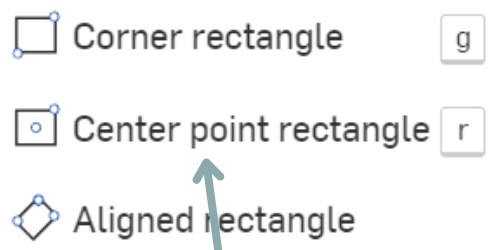
*Basic Part Live Demo*



# BASIC PART

## Step 2:

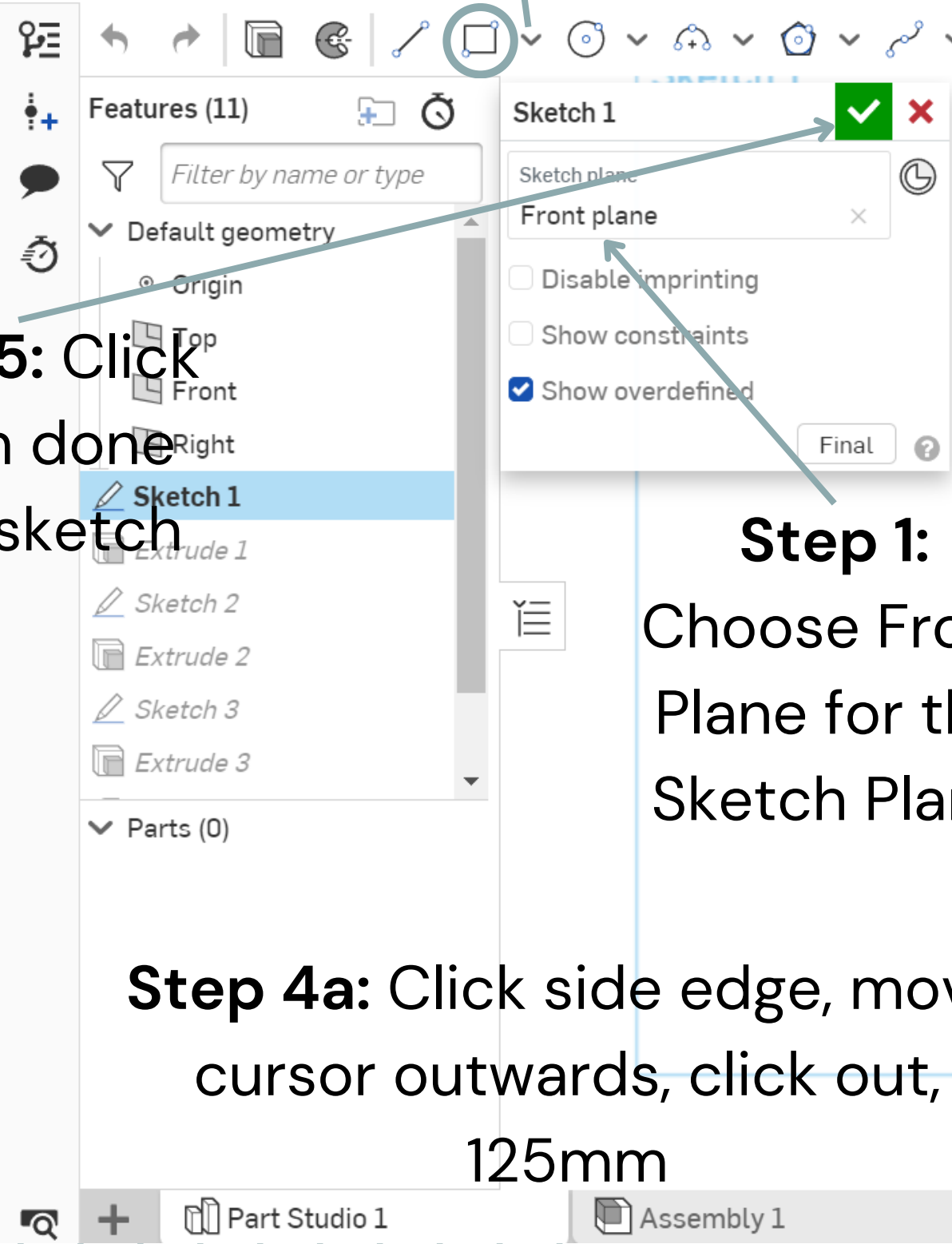
Choose center point rectangle



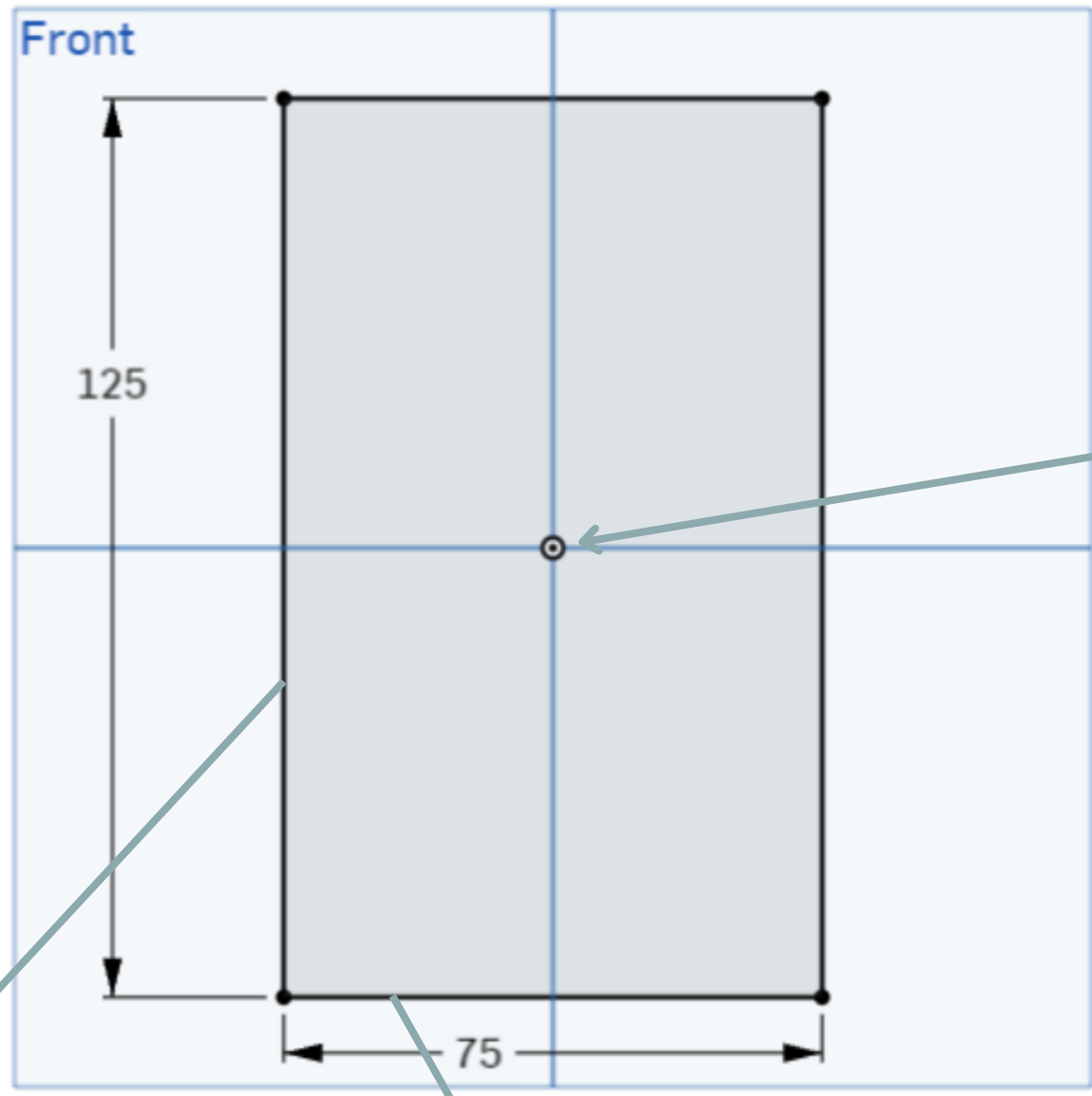
## Step 4: Select Dimension Tool



## Step 5: Click when done with sketch



**Step 1:**  
Choose Front Plane for the Sketch Plane



**Step 3:** Click the origin and drag outwards

**Step 4a:** Click side edge, move cursor outwards, click out, 125mm

**Step 4b:** Click top/bottom edge, move cursor outwards, click out, 75mm

# BASIC PART

**Step 1:** New feature:  
Extrude sketch  
(rectangle becomes box)

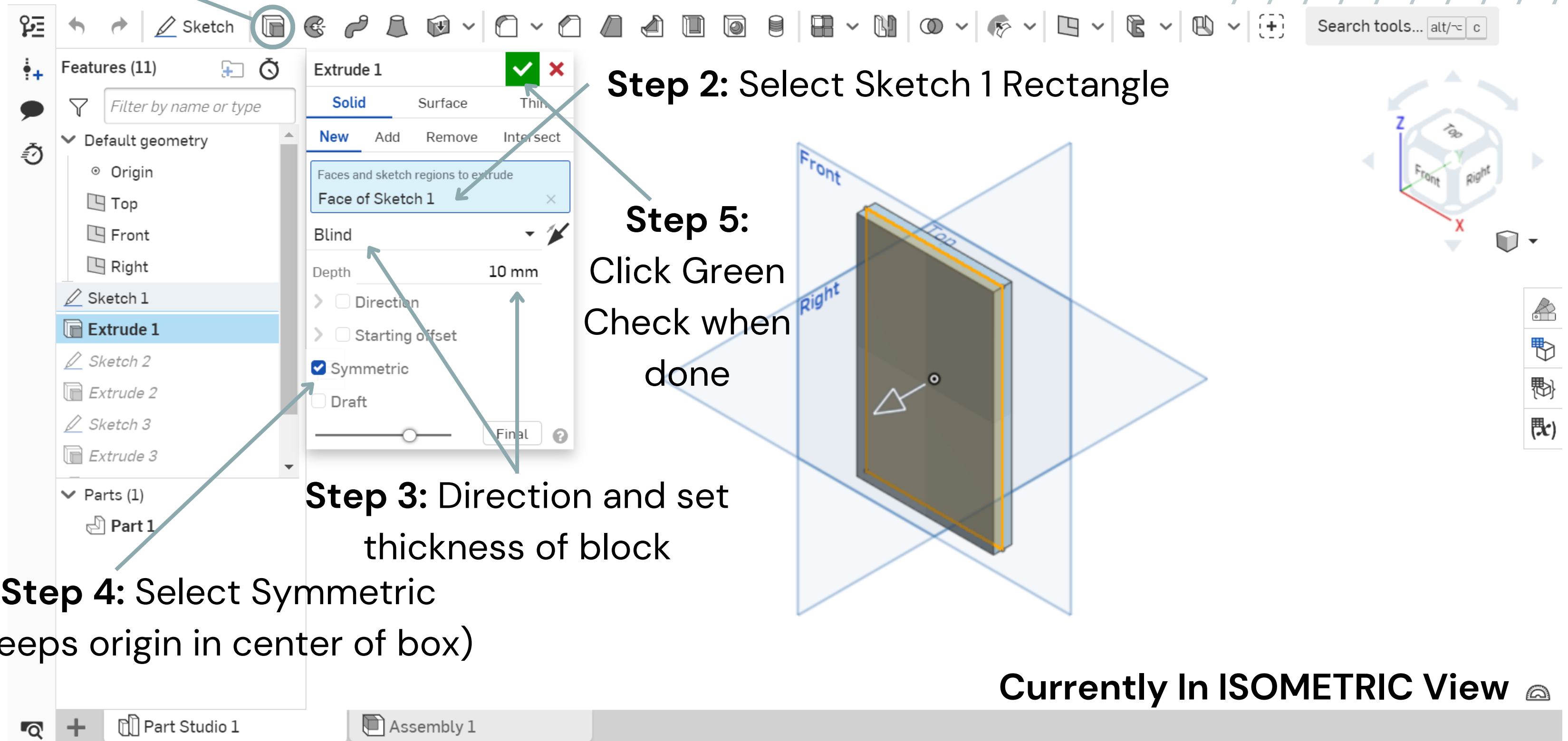
**Step 2:** Select Sketch 1 Rectangle

**Step 5:**  
Click Green  
Check when  
done

**Step 3:** Direction and set  
thickness of block

**Step 4:** Select Symmetric  
(keeps origin in center of box)

**Currently In ISOMETRIC View**





# BASIC PART

Create new sketch

Click Green Check Button when done

**Step 1:** Select front side of new box as sketch plane

**Step 2 (x2):** Select center circle, click on blue vertical line and drag outwards (circle)

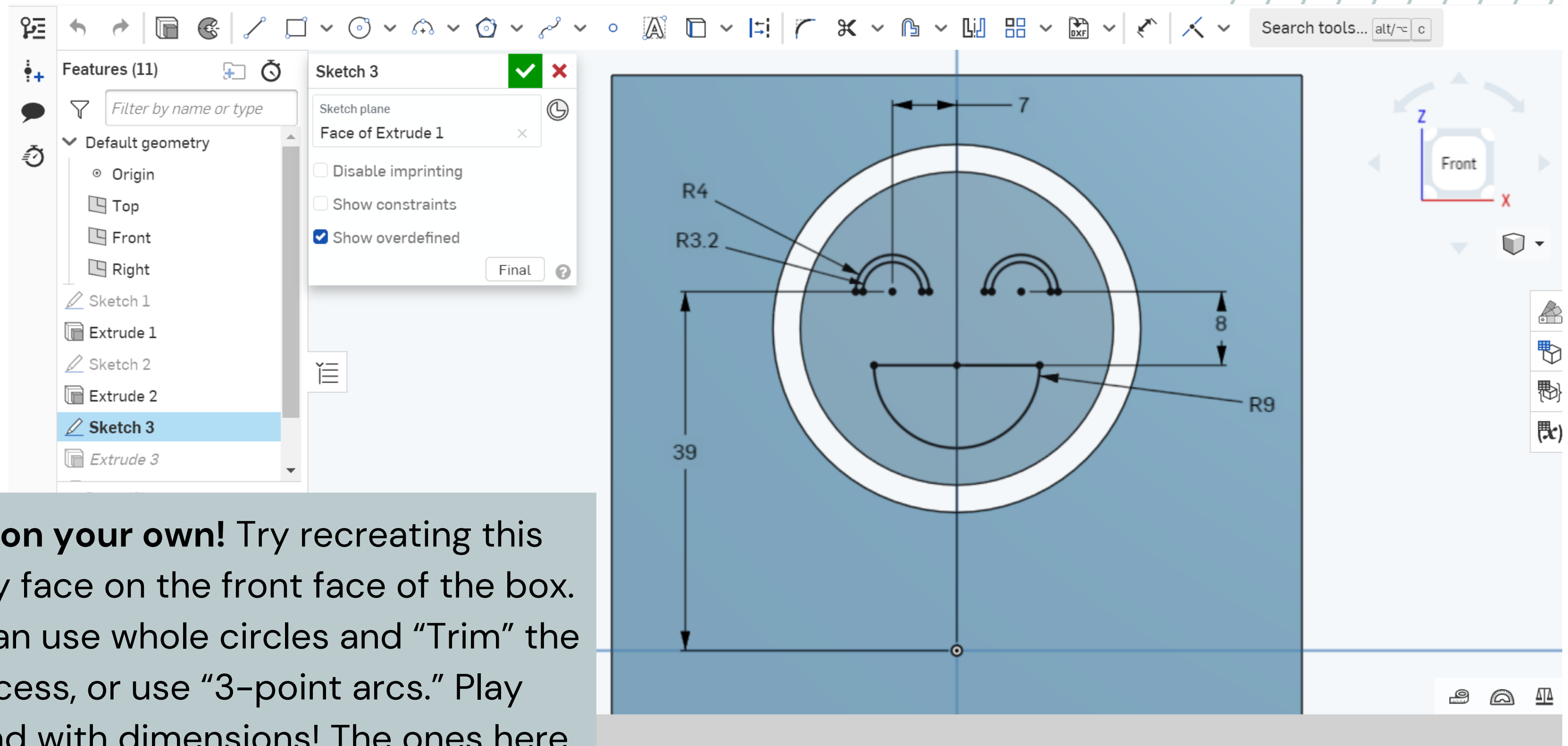
**Step 3: Dimensions**  
Outer Circle Diameter: 40mm  
Distance between inner and outer circle: 3mm  
Distance between center of circles and origin: 35mm

# BASIC PART

The image shows a CAD software interface with the 'Extrude 2' dialog box open. The dialog has tabs for 'Solid', 'Surface', and 'Thin', with 'Solid' selected. Under the 'Remove' tab, the 'Remove' button is circled in red. Below it, the 'Through all' option is also circled in red. The 3D model shows a rectangular block with a circular cutout, and the 'Remove' button and 'Through all' option are highlighted with red circles. A callout box explains the 'Extrude Cut' process.

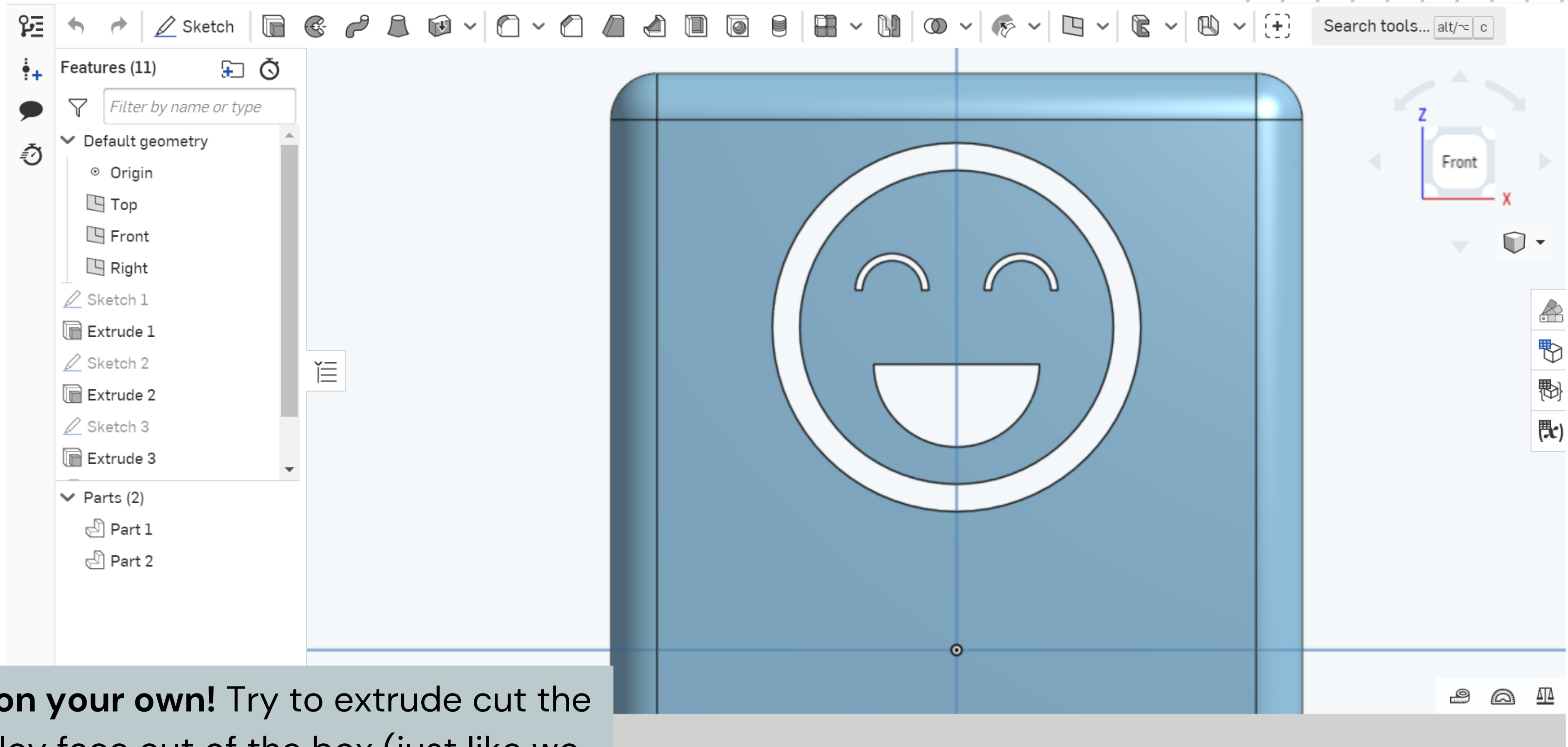
**Extrude Cut:** Same extrude as previous, select the circle outline and these new options. Once your preview looks like this, click the green check mark to finish.

# BASIC PART



**Try on your own!** Try recreating this smiley face on the front face of the box. You can use whole circles and “Trim” the excess, or use “3-point arcs.” Play around with dimensions! The ones here are just a guideline.

# BASIC PART



**Try on your own!** Try to extrude cut the smiley face out of the box (just like we did with the circle outline).

# BASIC PART

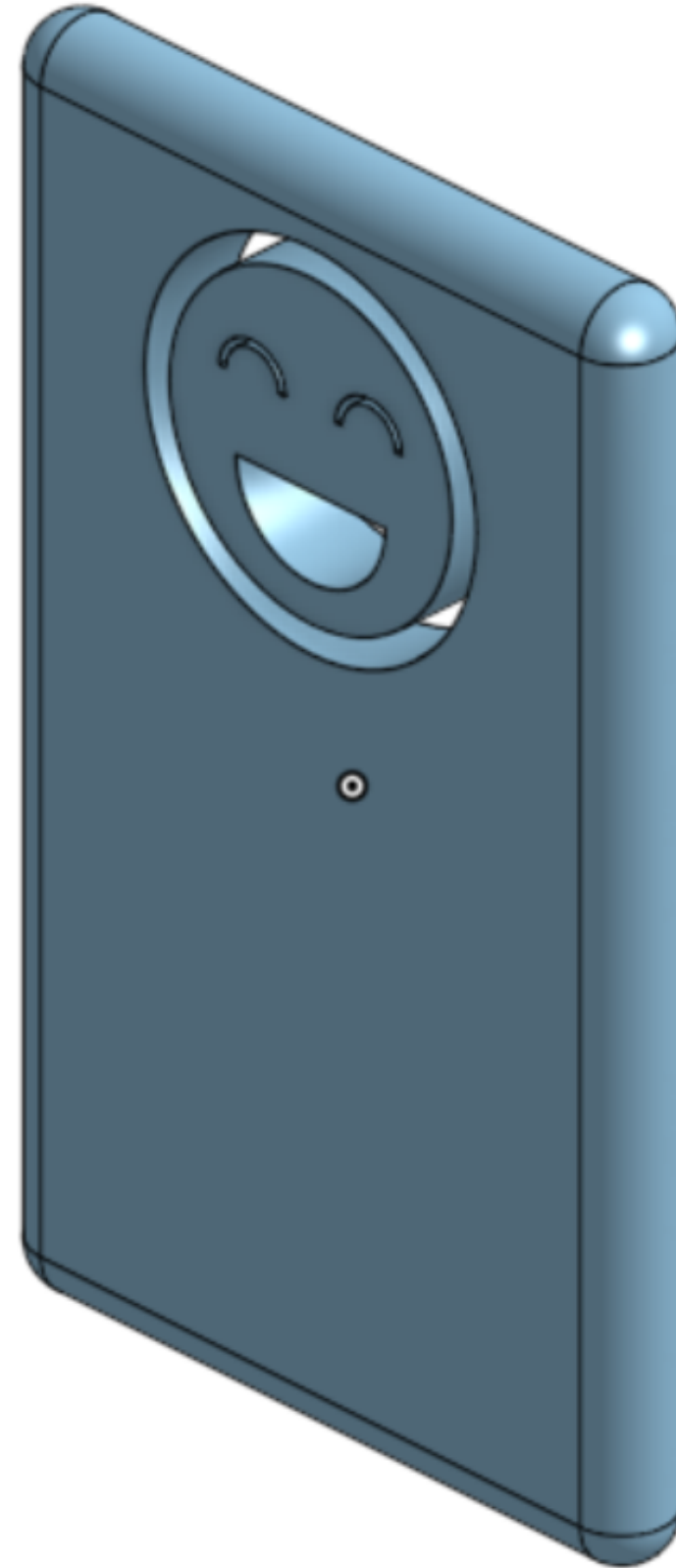
The screenshot displays the CAD software interface with the Fillet tool active. The Fillet 1 dialog box is open, showing the following settings:

- Edge** (Selected)
- Full round** (Selected)
- Entities to fillet:** Edge of Extrude 1, Edge of Extrude 1, Edge of Extrude 1, Edge of Extrude 1
- Tangent propagation**
- Measurement:** Radius
- Control:** Distance
- Radius:** 5 mm
- Asymmetric**
- Partial fillet**
- Variable fillet**
- Allow edge overflow**
- Smooth fillet corners**

The 3D model shows a box with a smiley face on its front face. The edges of the box are highlighted in orange, indicating they are selected for the fillet operation. The Fillet tool icon is visible in the top right corner of the software interface. A 3D coordinate system is also shown in the top right corner.

**New tool: Fillet**  
Select Fillet, select all the edges of the box, set the radius to 5mm, and click the green check. EDGES SHOULD CURVE!

# BASIC PART COMPLETE!



03.

# LET'S CONTINUE?

*Intermediate Part Live Demo*



# INTERMEDIATE PART

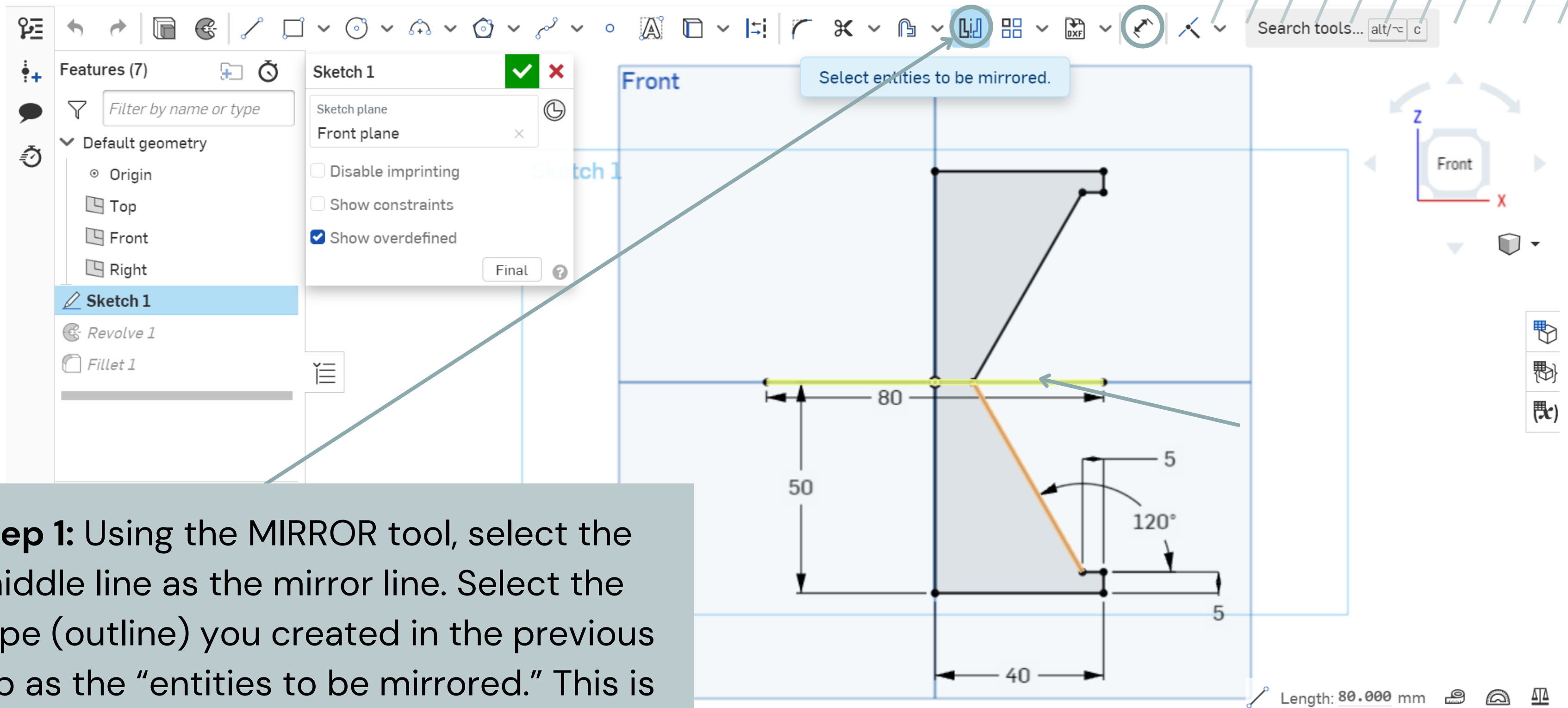
**Step 1:** Using the line tool, create this shape. Make sure the lines are straight (parallel and perpendicular) because these relations matter.

**Step 2:** Dimension all the sides with respect to the origin. *Hint:* To define an angle, select 2 adjacent sides.

**Make sure to add this line in the middle. It is important for the next step!**

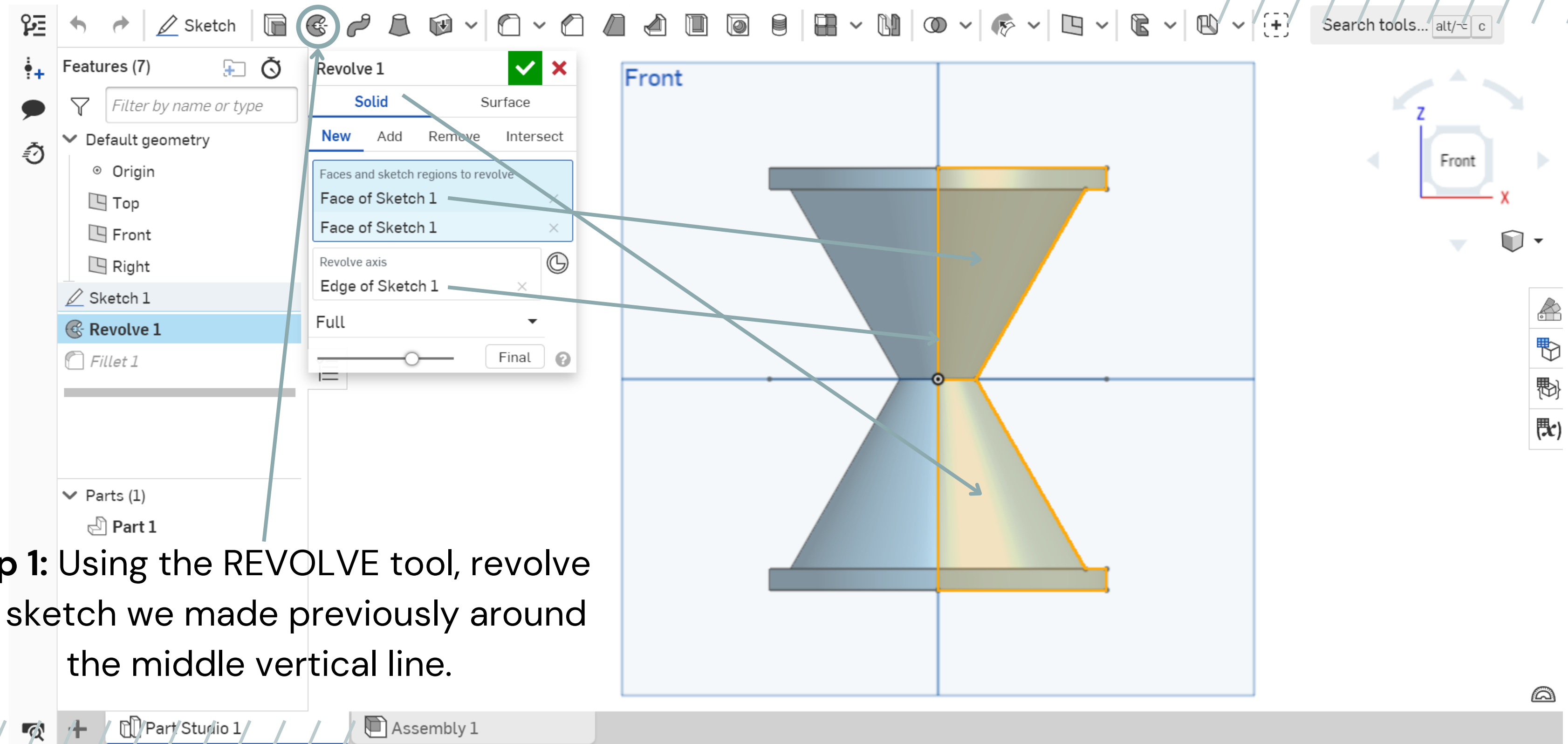


# INTERMEDIATE PART



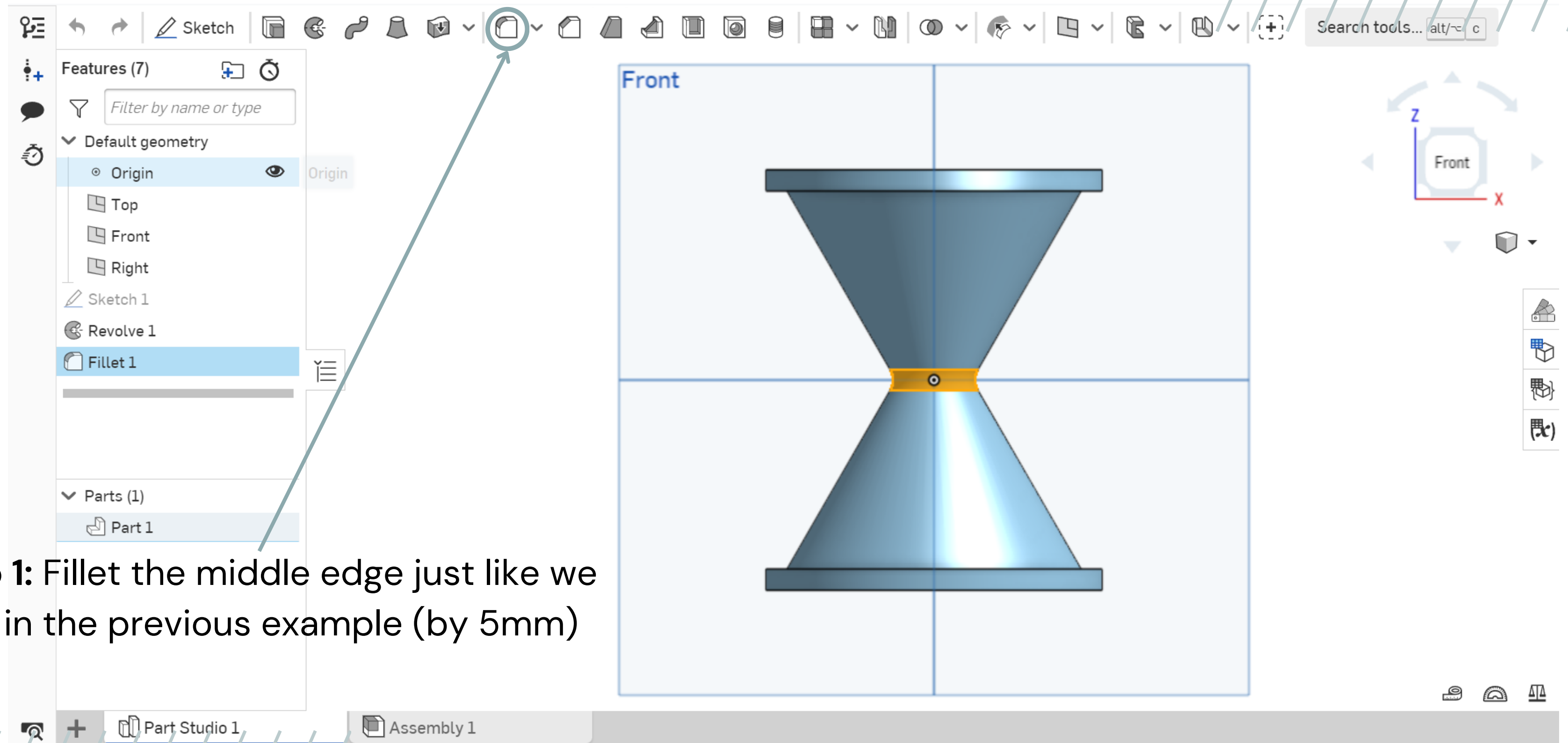
**Step 1:** Using the MIRROR tool, select the middle line as the mirror line. Select the shape (outline) you created in the previous step as the “entities to be mirrored.” This is what it should look like after.

# INTERMEDIATE PART

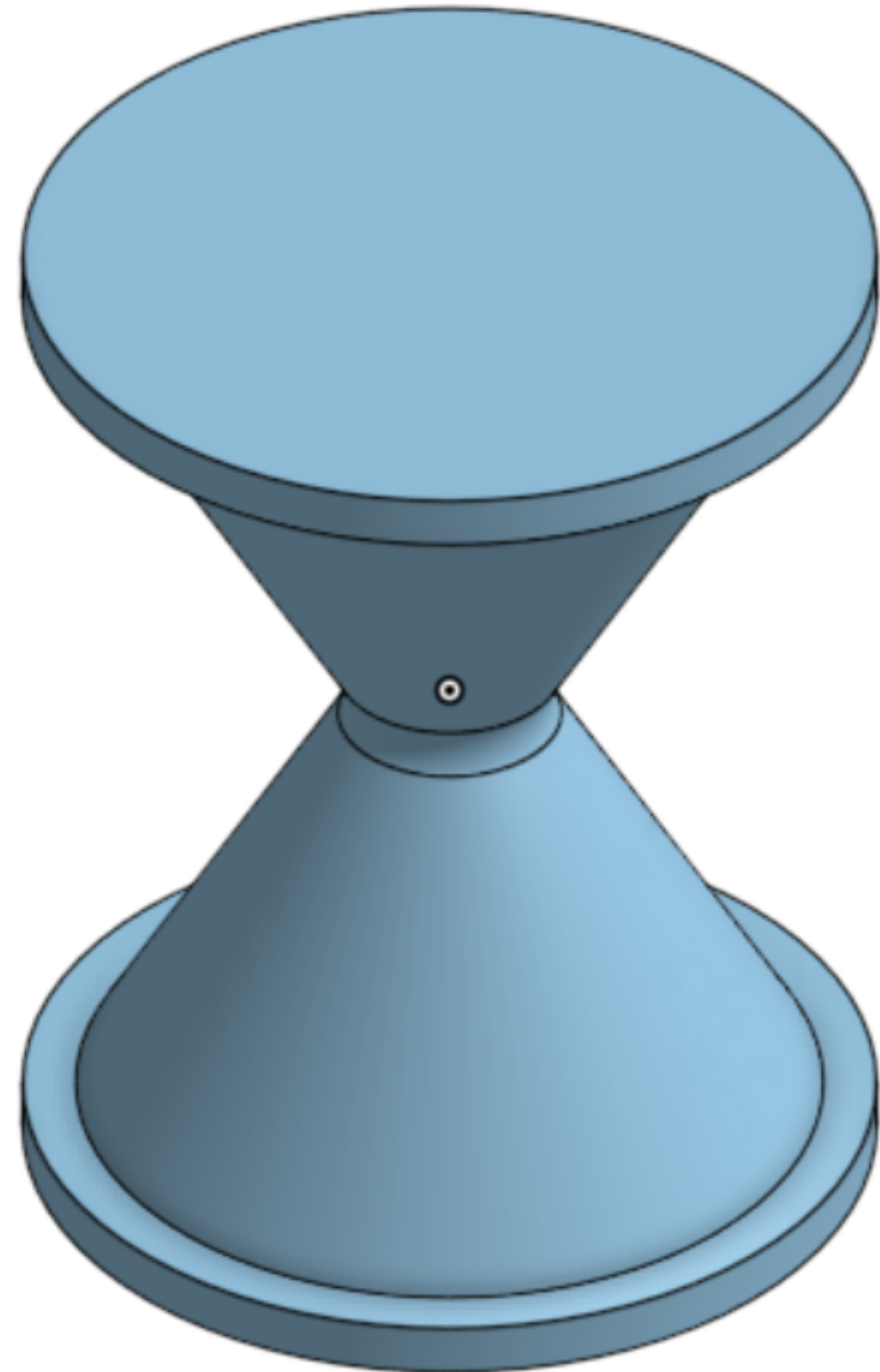


**Step 1:** Using the REVOLVE tool, revolve the sketch we made previously around the middle vertical line.

# INTERMEDIATE PART



INTERMEDIATE PART COMPLETE!



01.

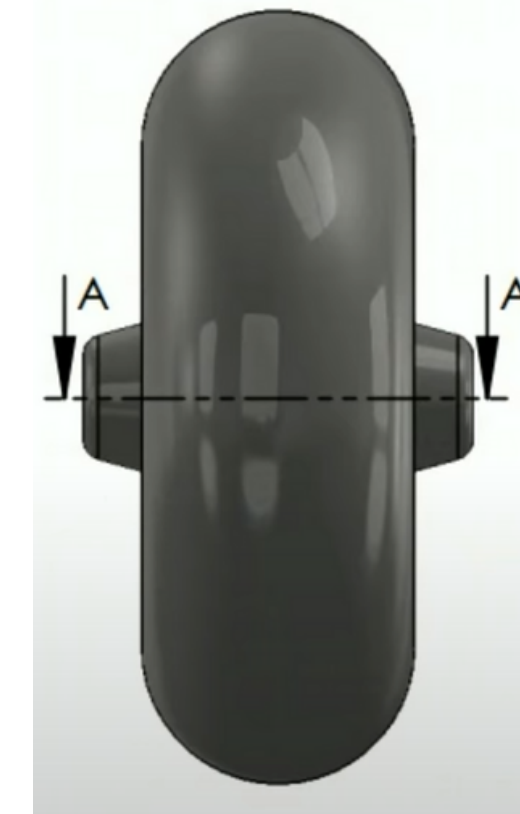
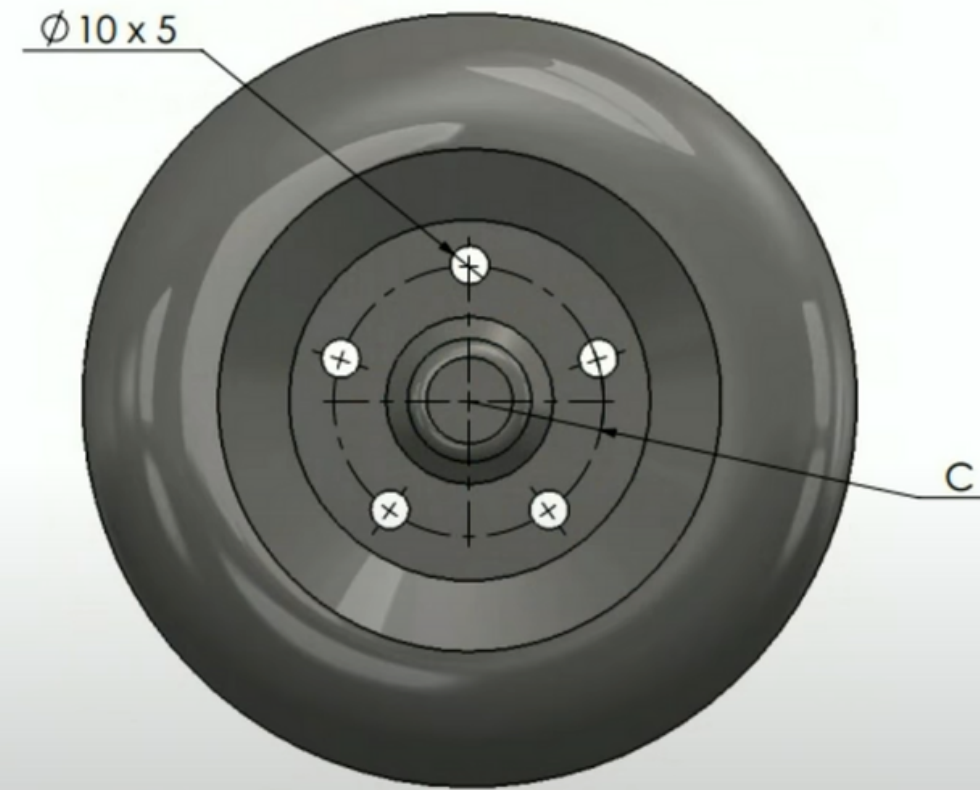
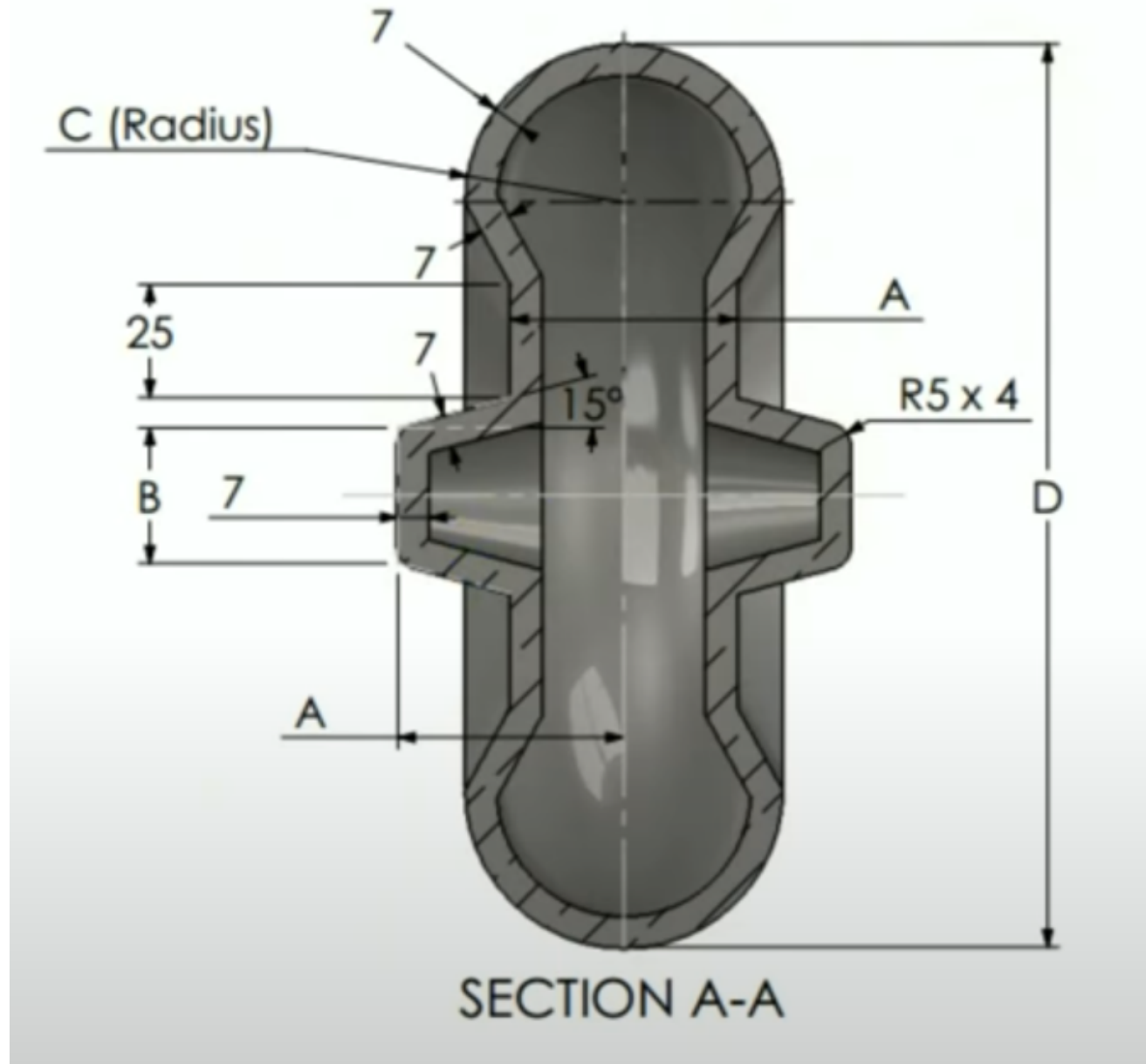
# CHALLENGE

*Can you put all the skills you learned and create this part?*



Raise your hand if you need help!

# CHALLENGE PART!



- A** = 50mm
- B** = 30mm
- C** = 35mm
- D** = 200mm

## Hints:

What sketch would you start out with? Can you sketch something smaller and mirror it?

What features can you use (extrude, extrude cut, revolve, mirror)?

For the smaller holes, can you make it easier instead of doing all 5? (Circular sketch pattern)