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ONSHAPE WORKSHOP

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01.

INTRODUCTION

Why CAD?

WHAT IS CAD?

C	A	D
COMPUTER	**AIDED**	**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.
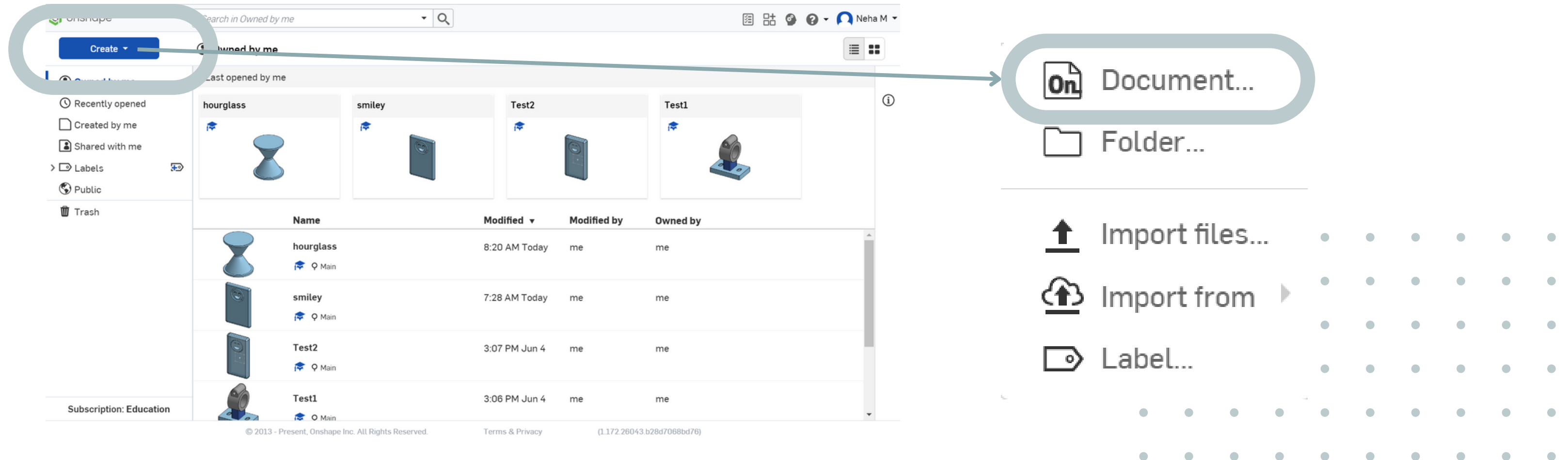
WHY?

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/** 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

PLATFORM

Make sure to NAME your document

Sketch Features

Create new sketch

Different Views
(can use to rotate)

Features (4)

Filter by name or type

∨ Default geometry
- ⊙ Origin
- Top
- Front
- Right

Sketch tree (all sketches, features, and geometry will be stored here)

∨ Parts (0)

Different Planes (can start sketches or add sketch features)

More views
(Isometric is mainly used)

Front

Top

Right

onshape ≡ **Untitled document** Main

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Sketch

Search tools... alt/~ c

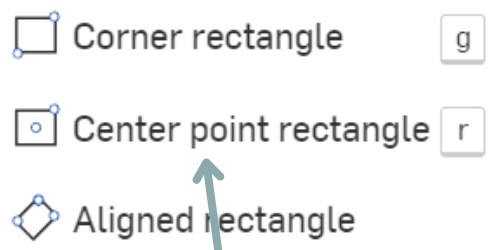Part Studio 1   Assembly 1

02.

LET'S GET STARTED!

Basic Part Live Demo
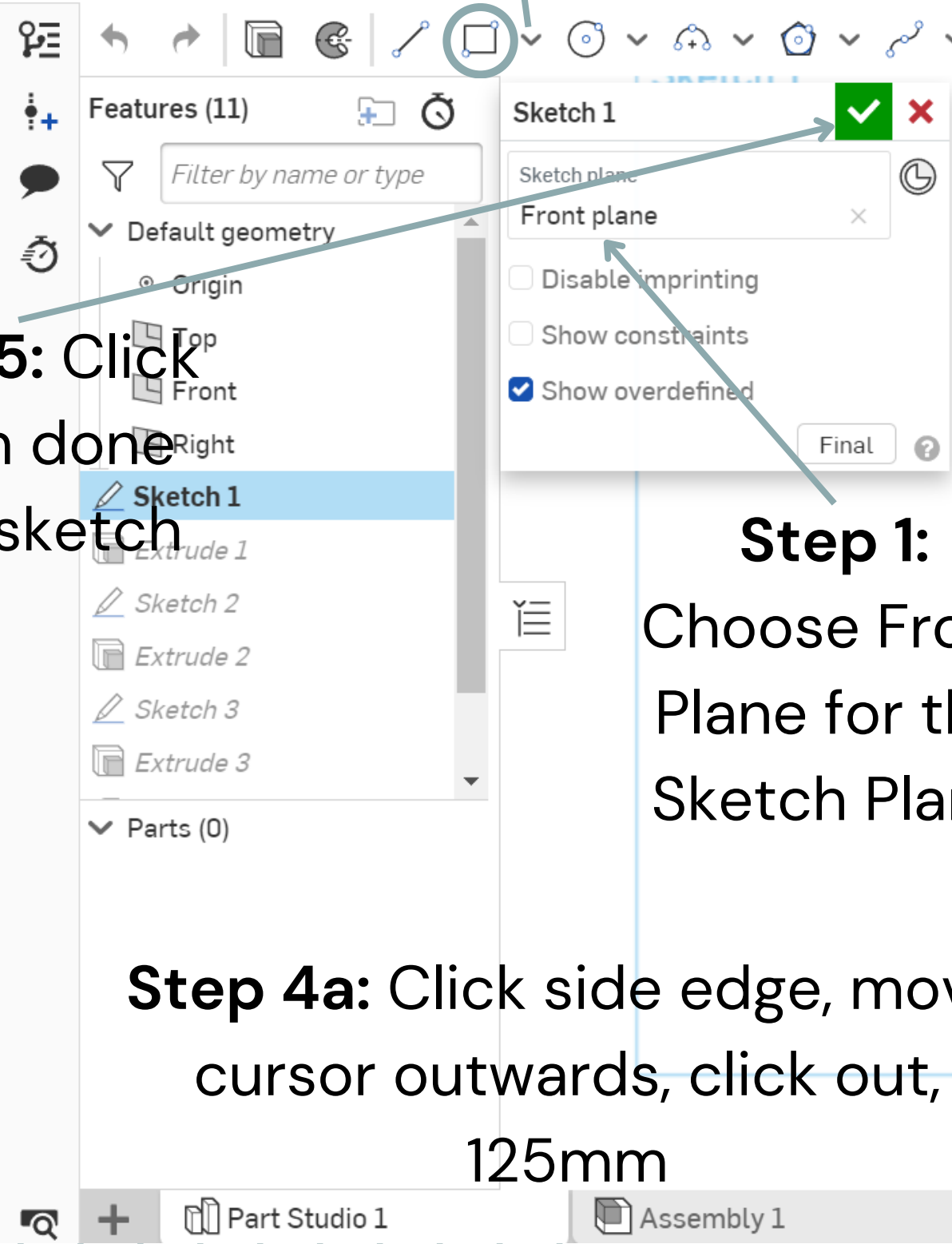
BASIC PART

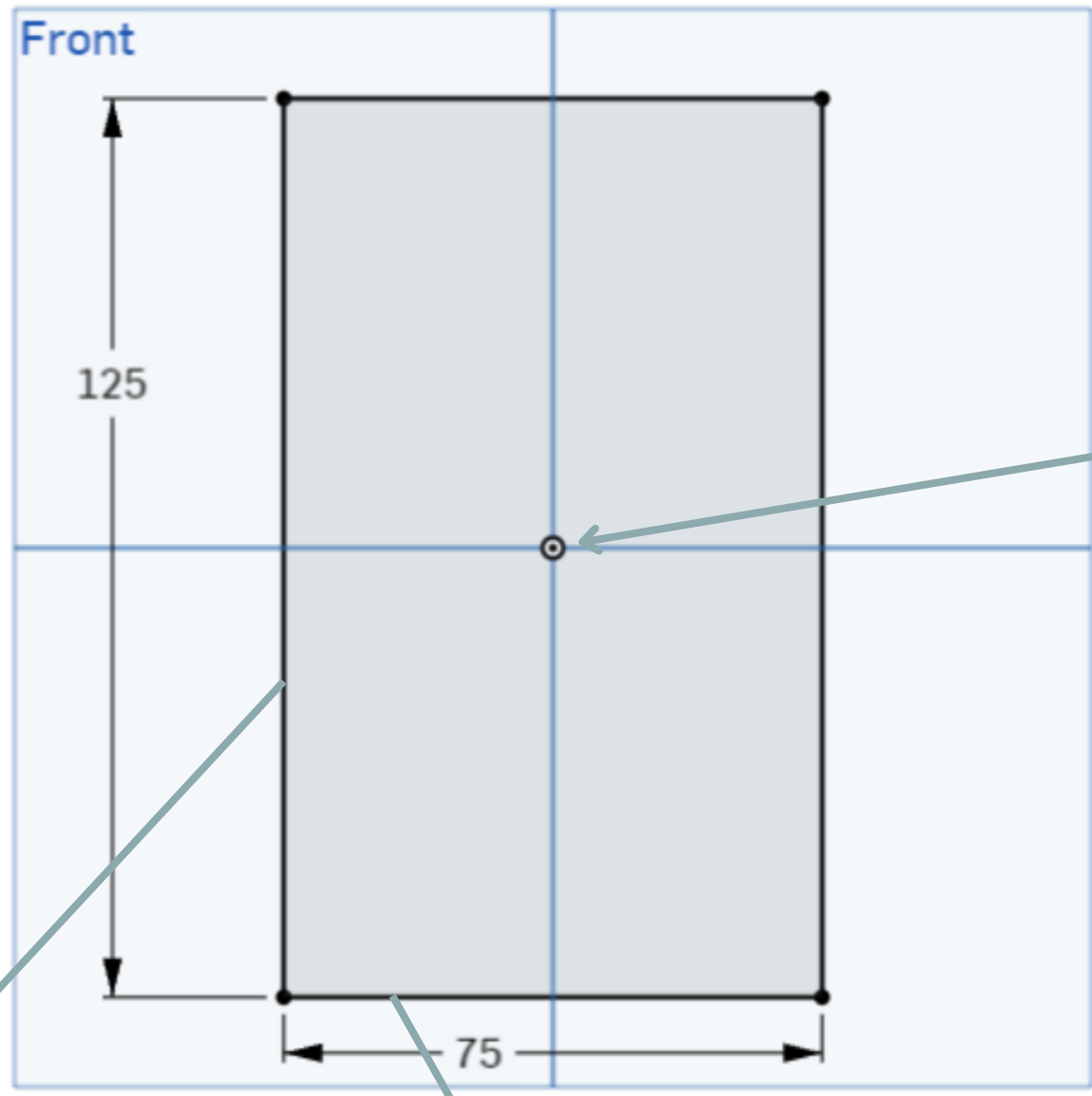**Step 2:** Choose center point rectangle

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)
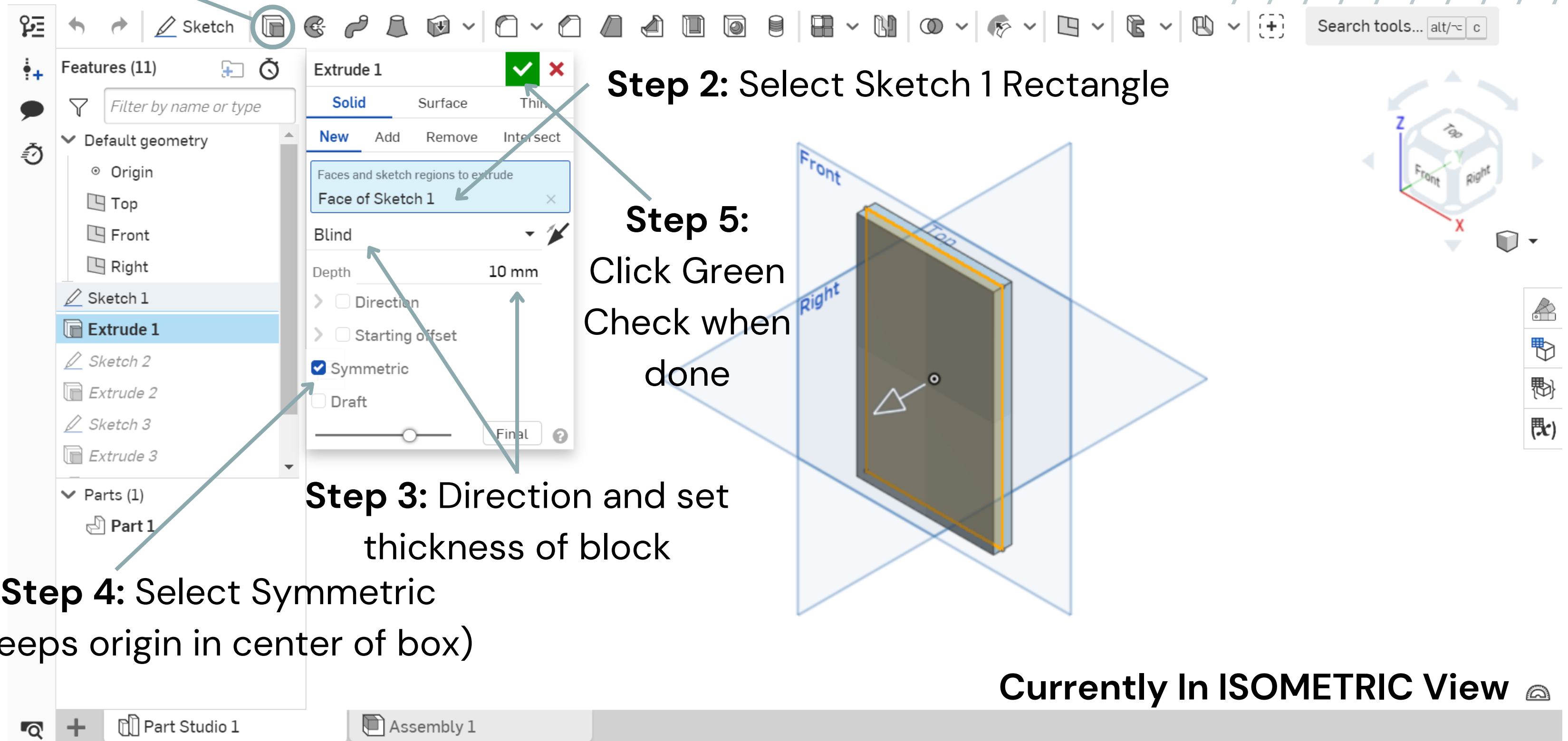
Step 2: Select Sketch 1 Rectangle

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

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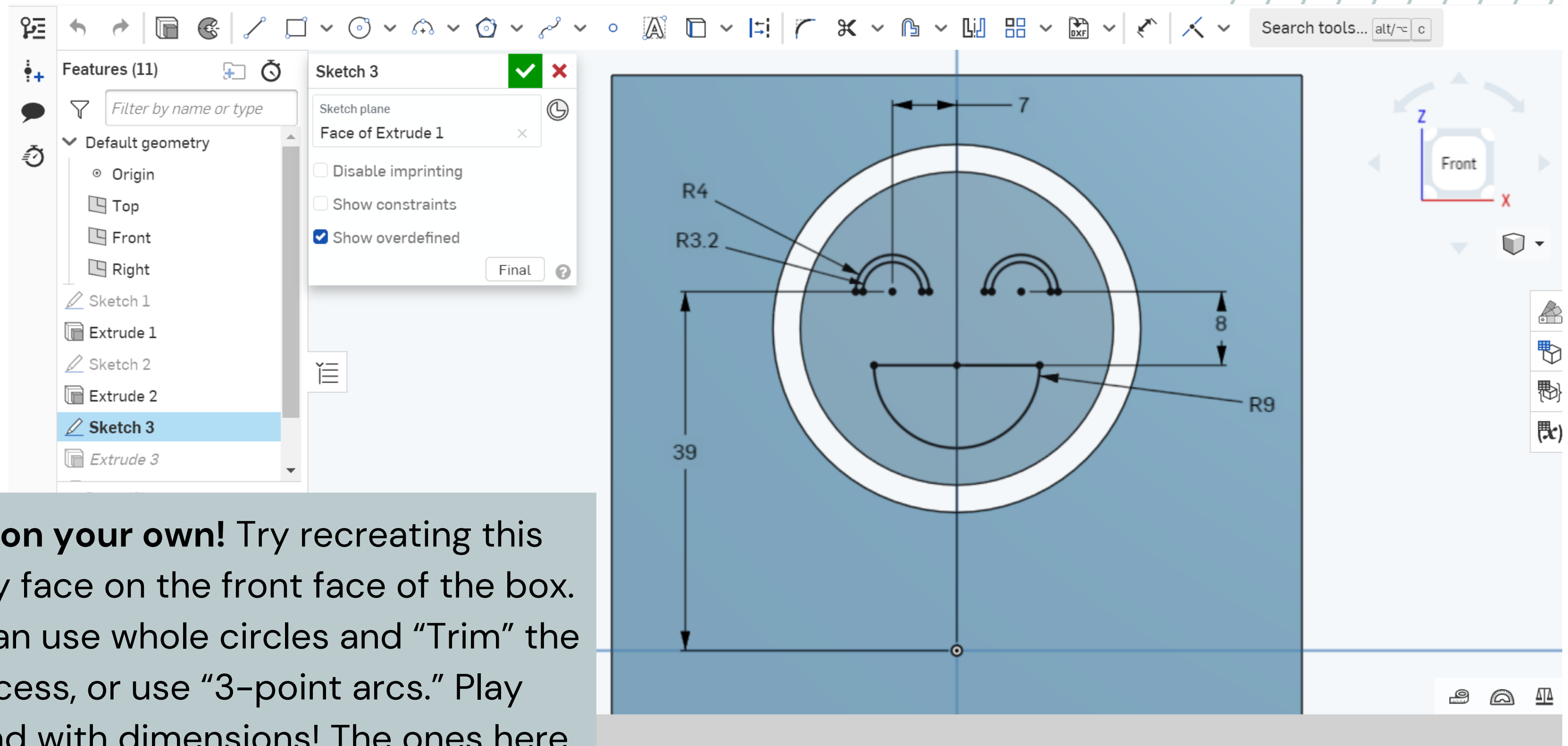

Step 3: Dimensions
Outer Circle Diameter: 40mm
Distance between inner and outer circle: 3mm
Distance between center of circles and origin: 35mm

BASIC PART

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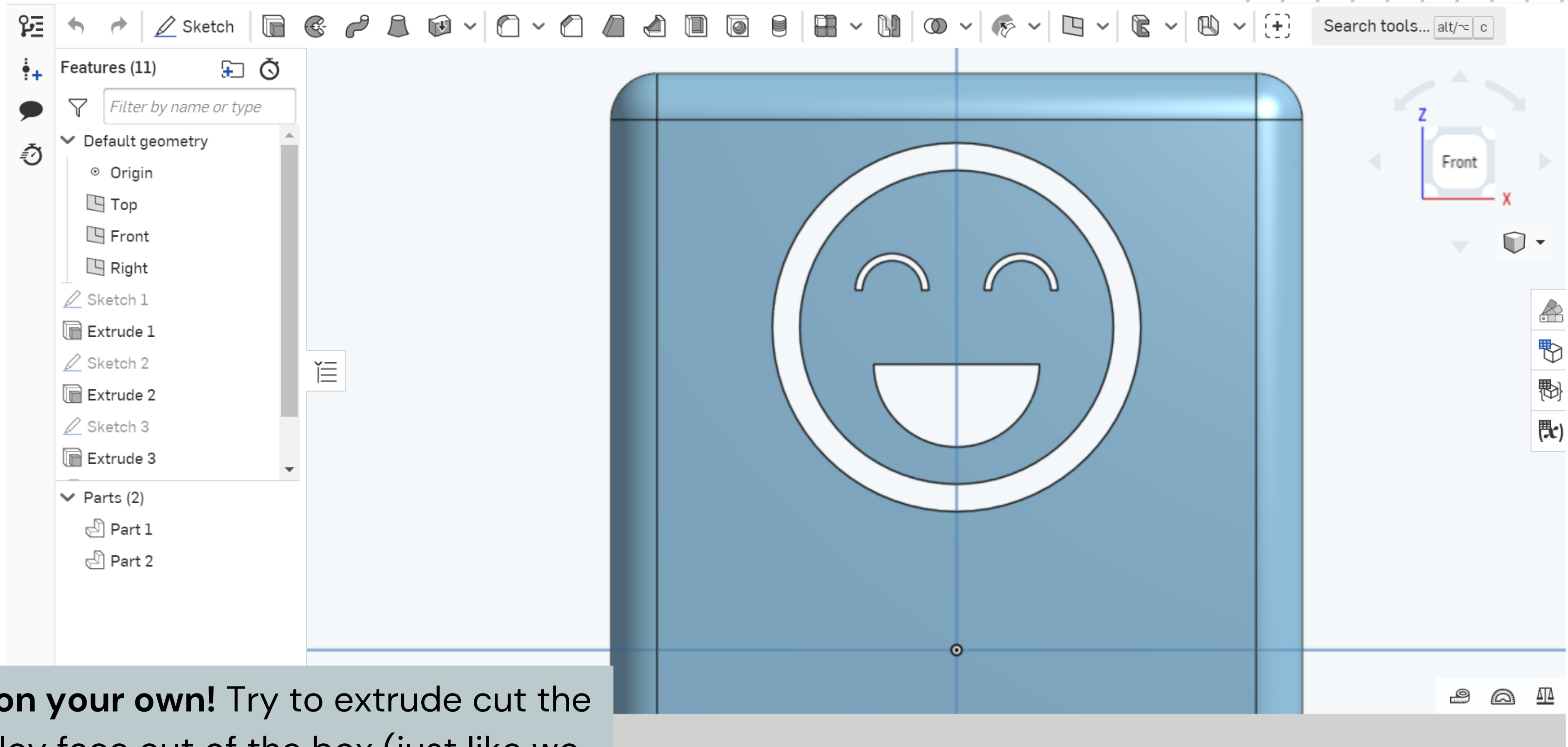
Try on your own! Try to extrude cut the smiley face out of the box (just like we did with the circle outline).
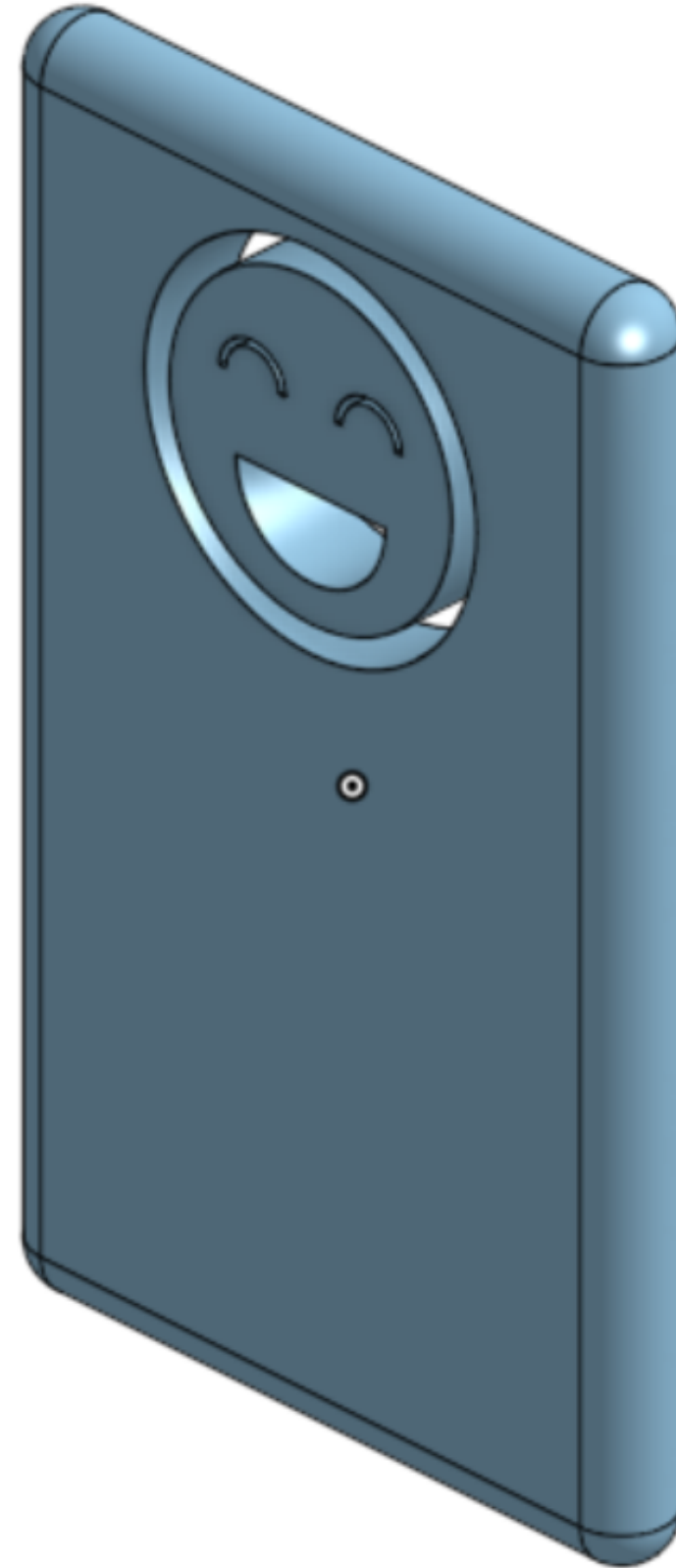
BASIC PART

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

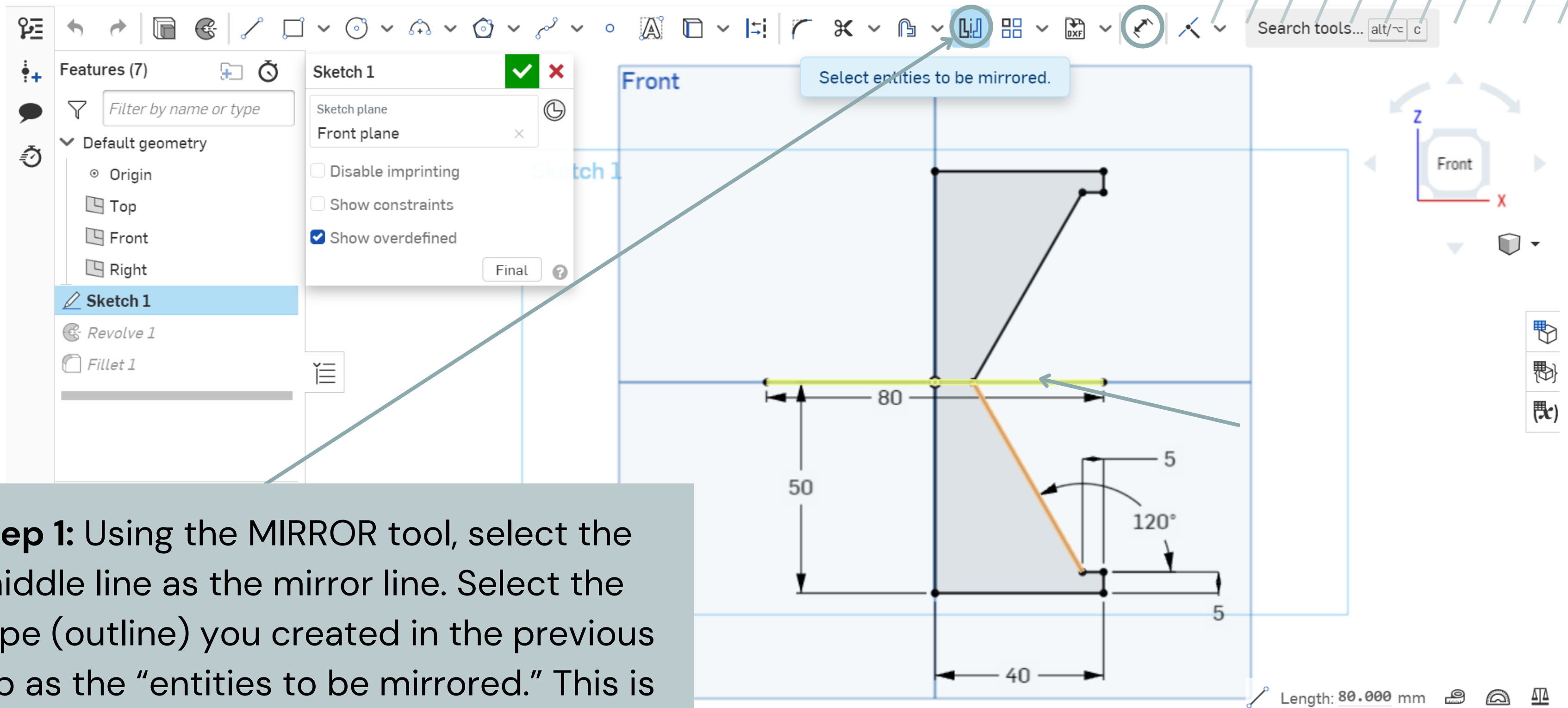**Make sure to add this line in the middle. It is important for the next step!**

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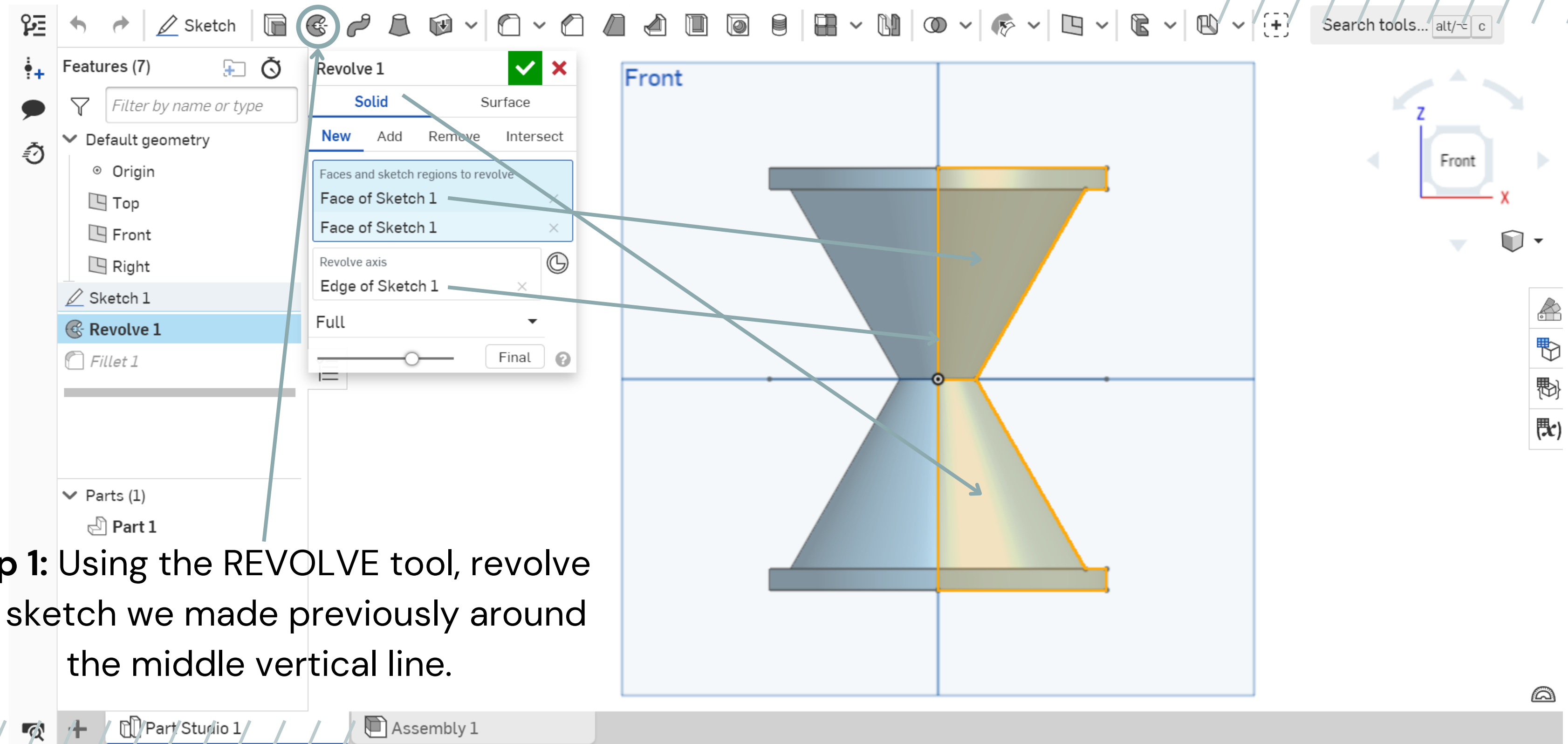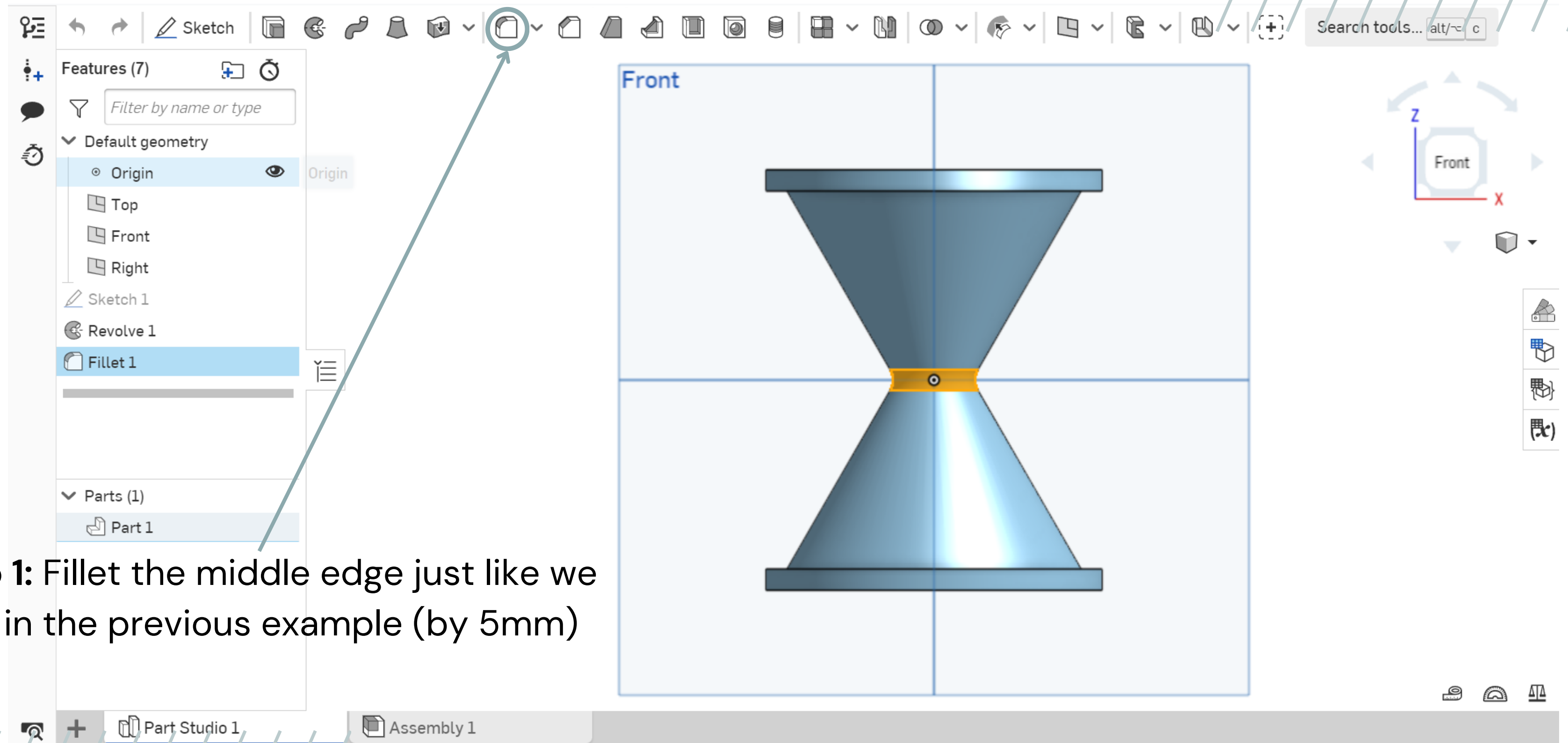
INTERMEDIATE PART

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.
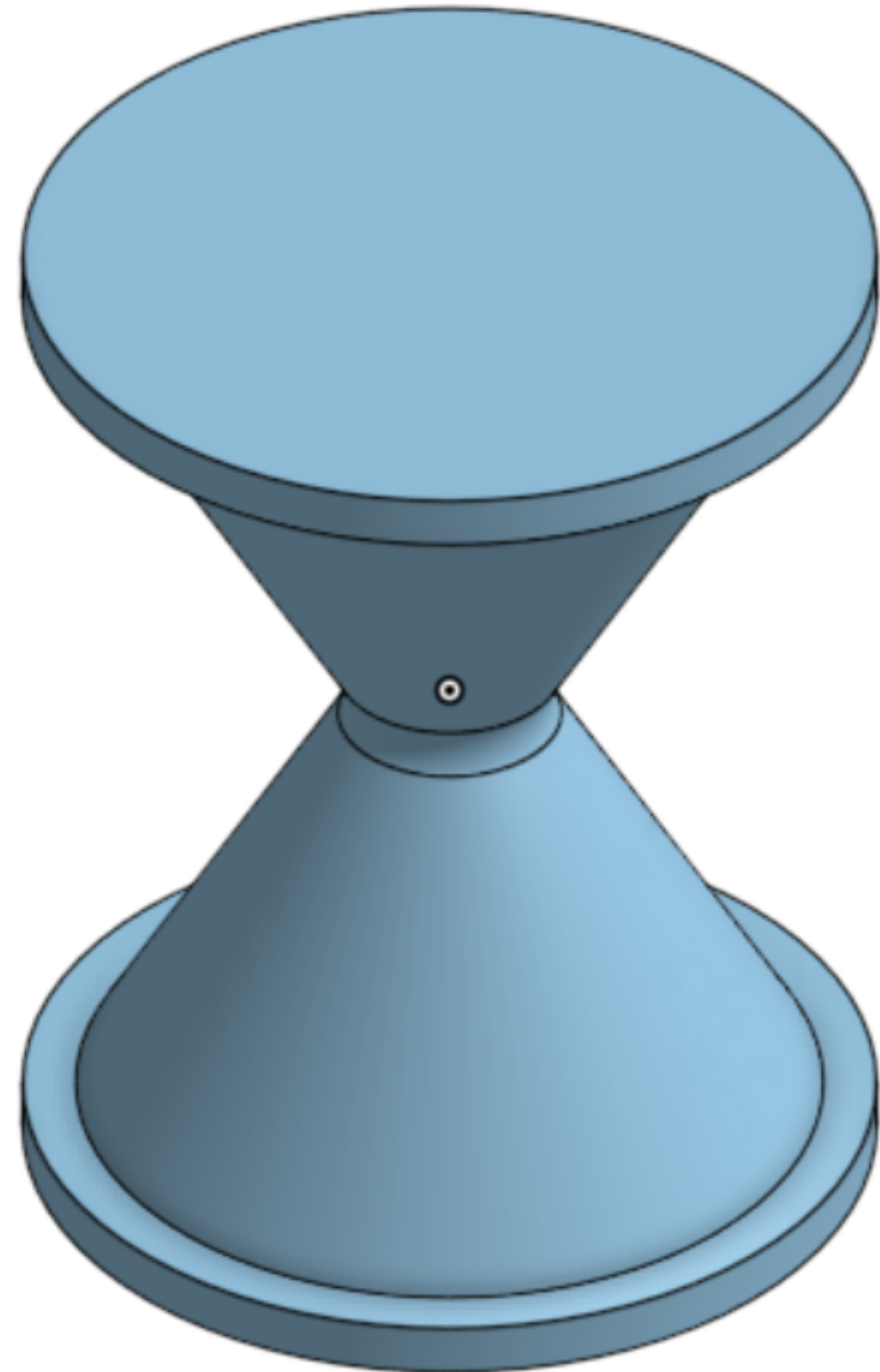
INTERMEDIATE PART

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

INTERMEDIATE PART

Step 1: Fillet the middle edge just like we did in the previous example (by 5mm)
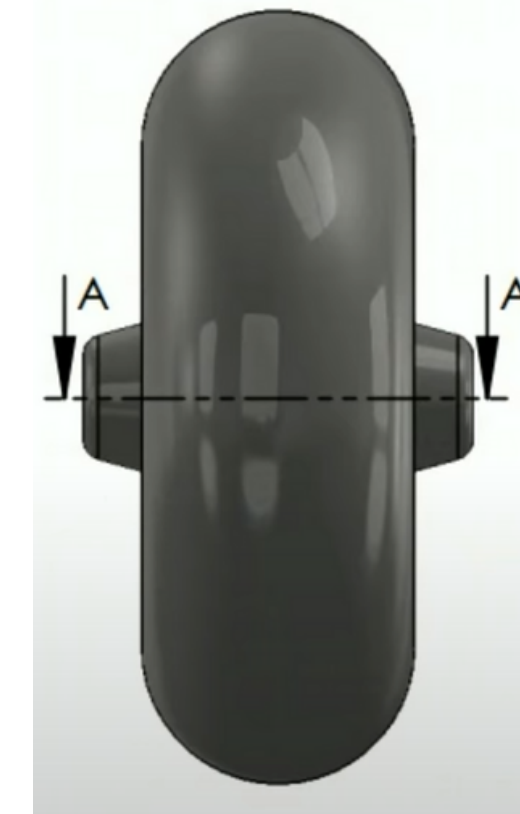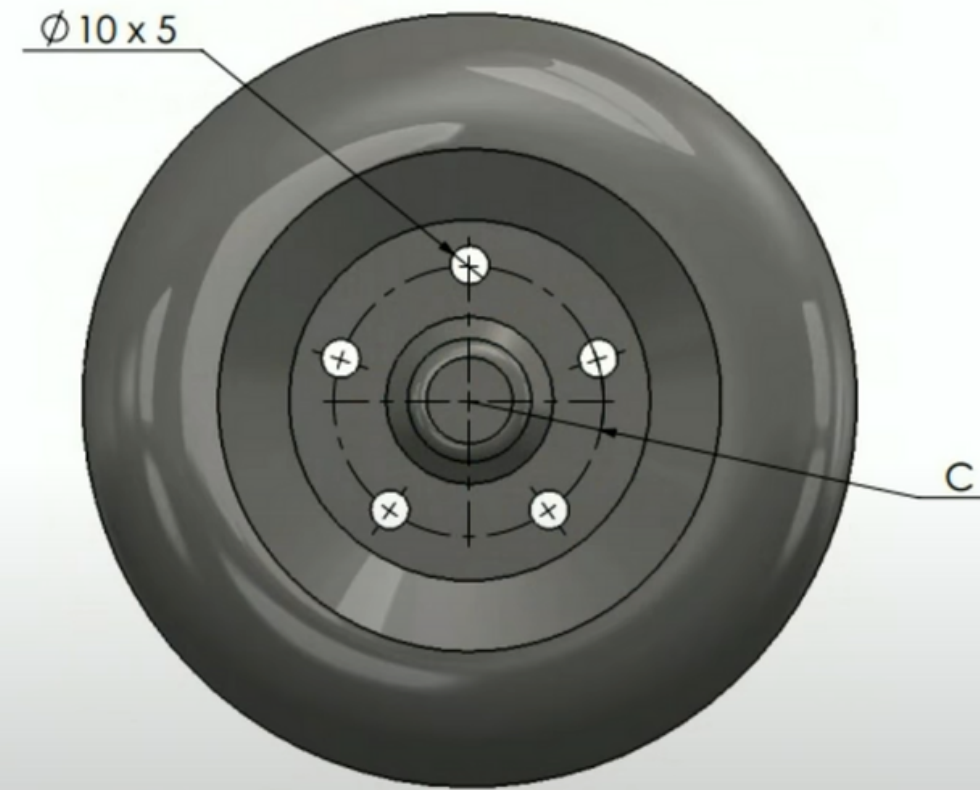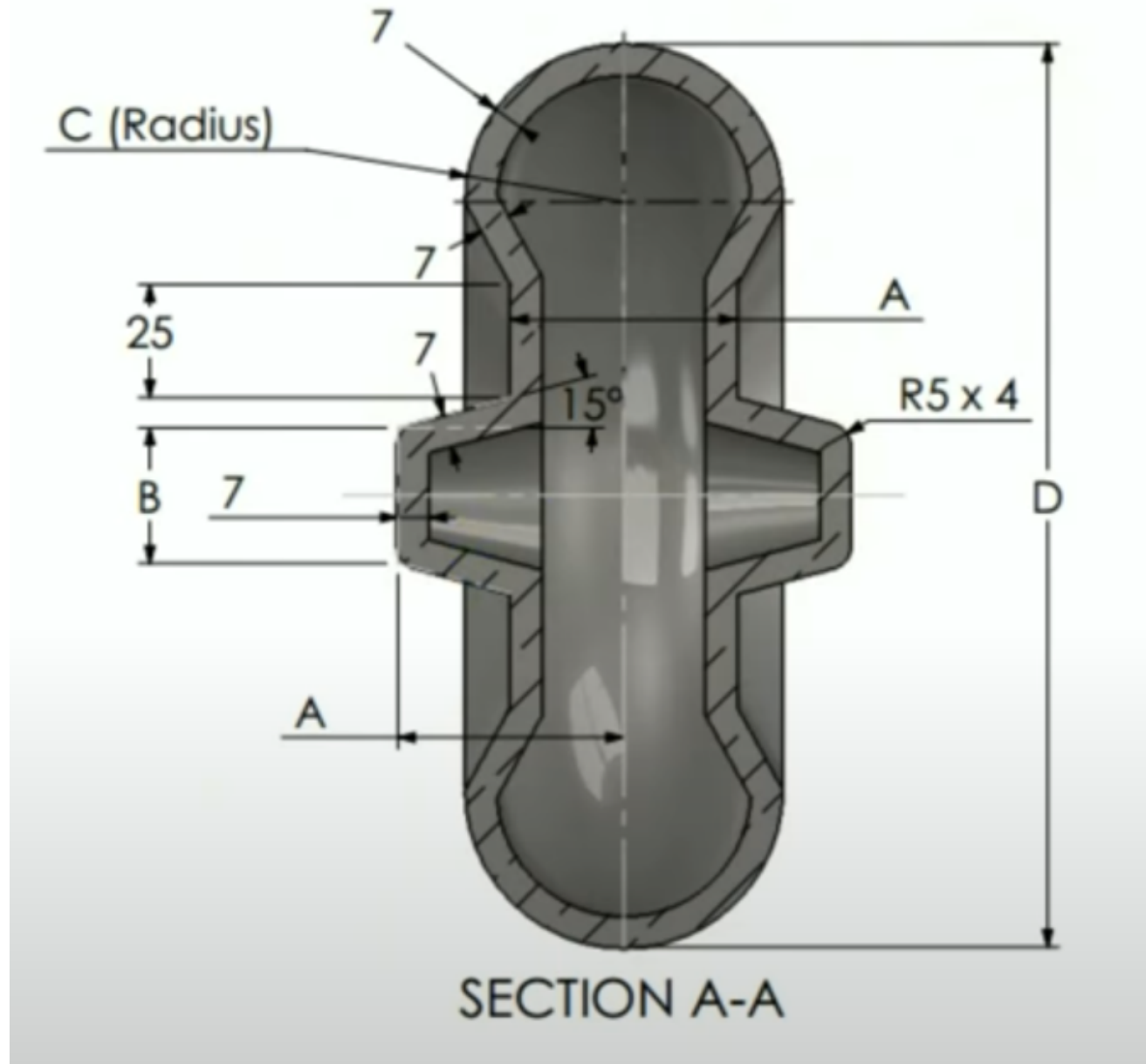
INTERMEDIATE PART COMPLETE!

01.

CHALLENGE

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

Raise your hand if you need help!

CHALLENGE PART!

SECTION A-A

C (Radius)
7
7
7
25
B
7
15°
A
R5 x 4
A
D

Ø 10 x 5
C

A | A

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)