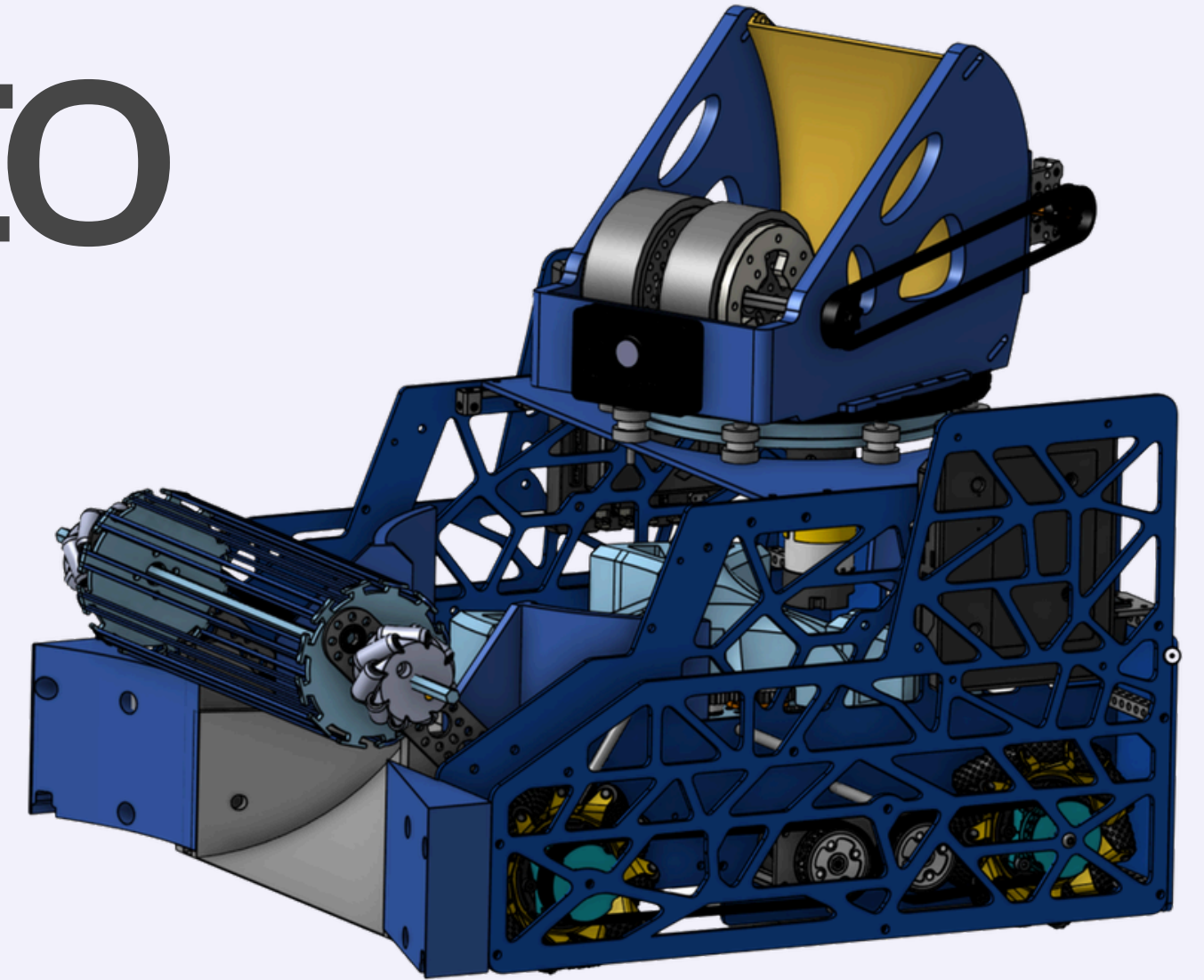
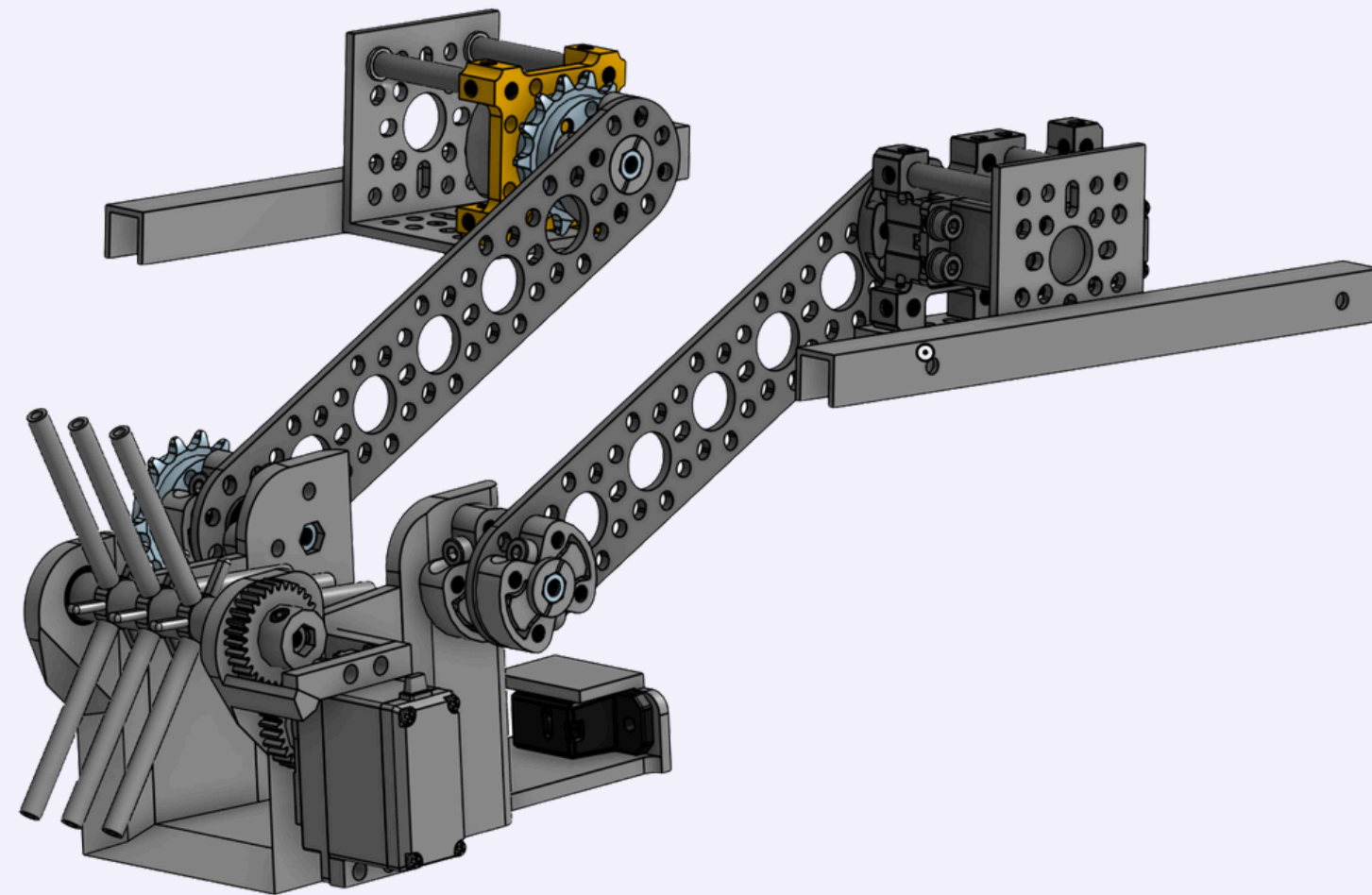

Introduction to CAD



Jason Zhu & Neil Benjamin
Engineering Co-leads
Grizzly Tech #23929



What is CAD?

- Software used to create precise 2D drawings and 3D models
- Used to design and test parts digitally before building them, helping catch problems early
- Common in engineering and robotics for designing mechanisms, assemblies, prototypes, and manufacturing-ready parts



Onshape

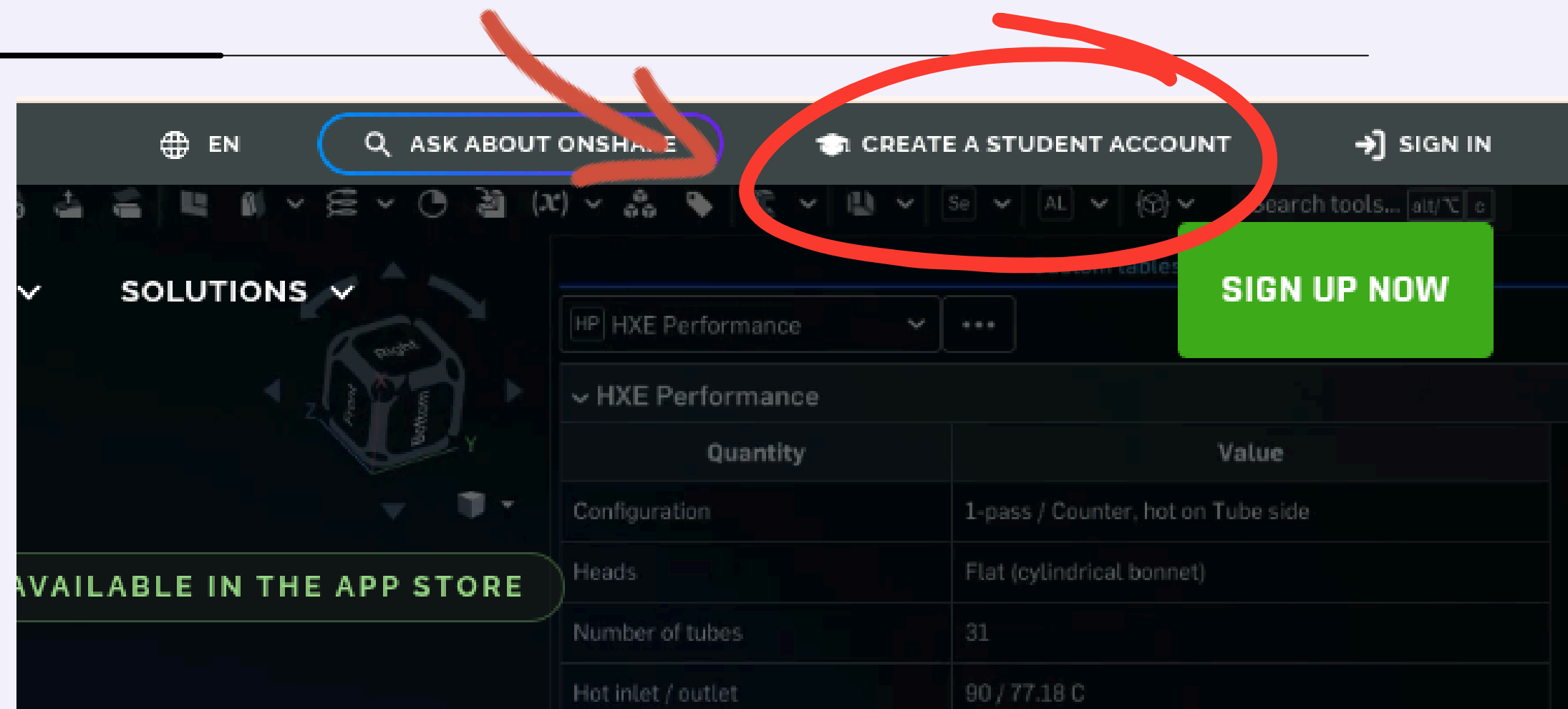


- Main software we use for designing the robot
- Browser-based (very convenient)
- Enables collaboration

Step 1

Create an Account!

onshape.com



Sign Up Now

First name*

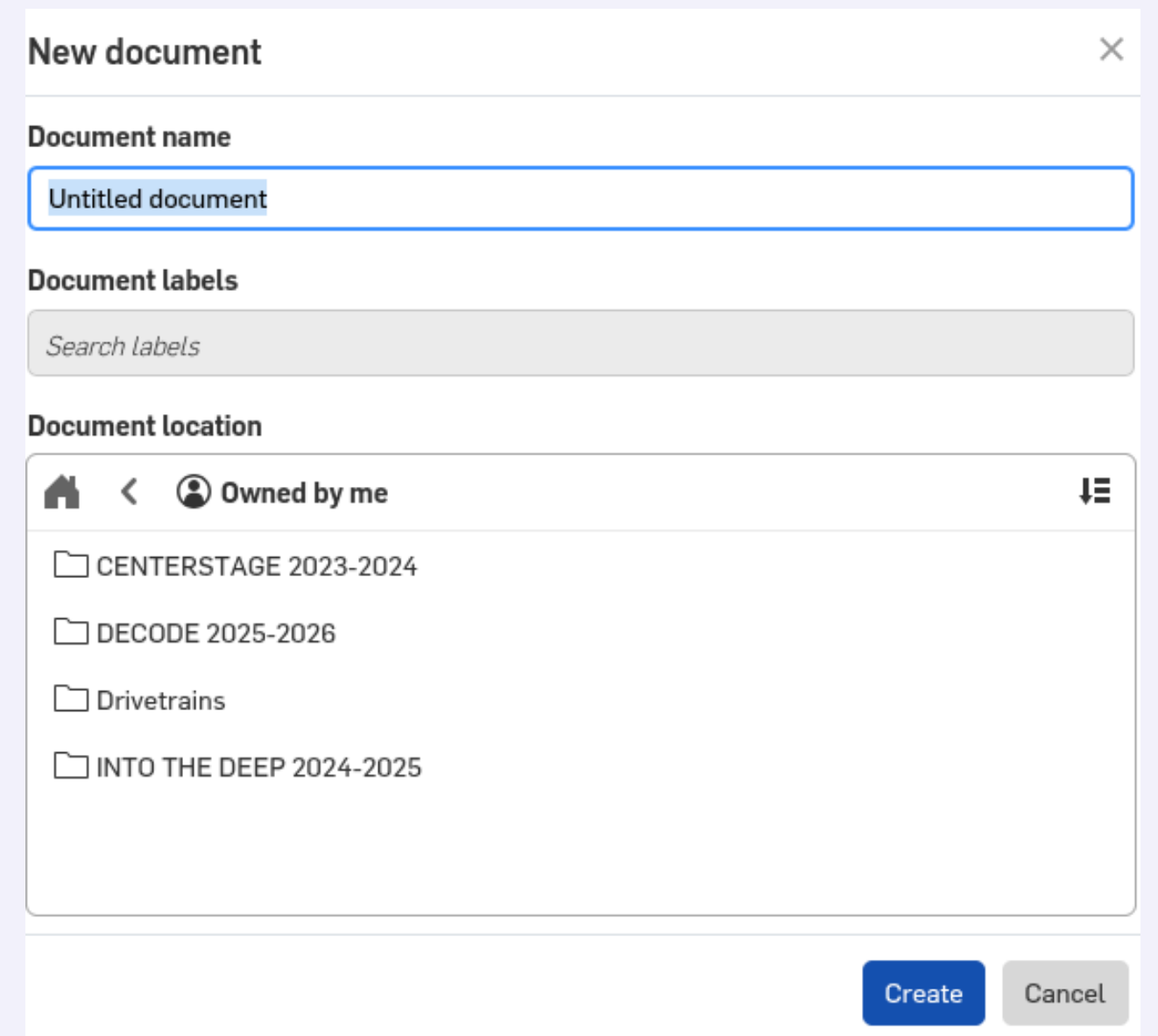
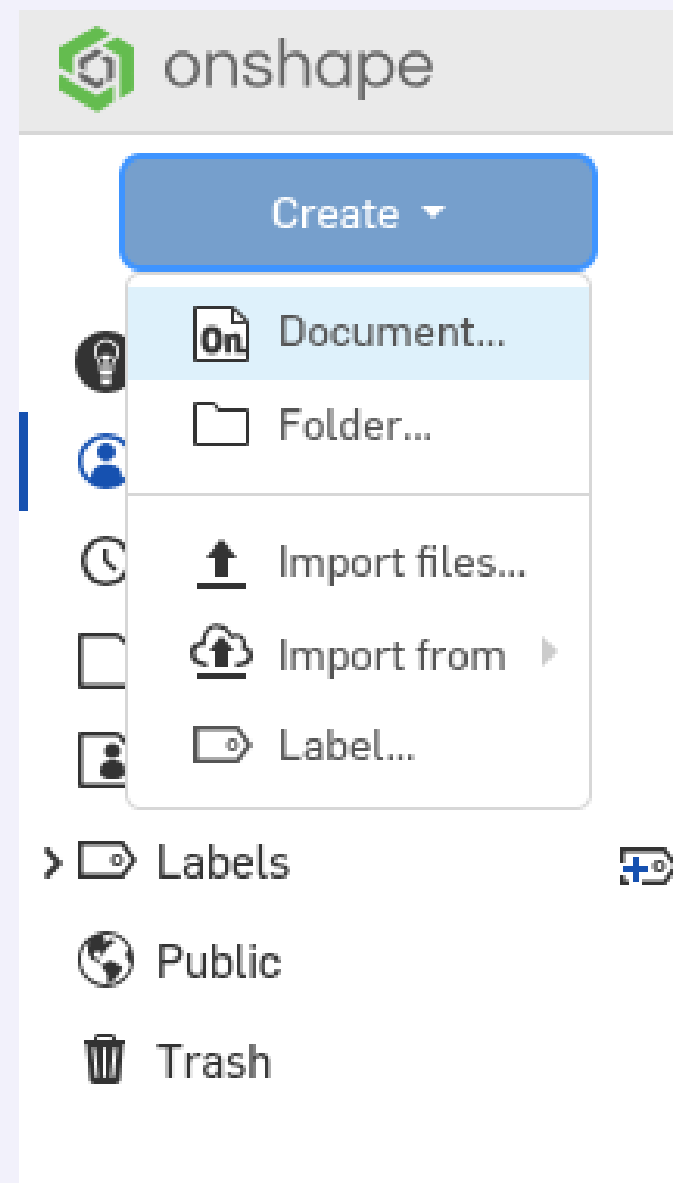
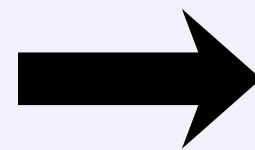
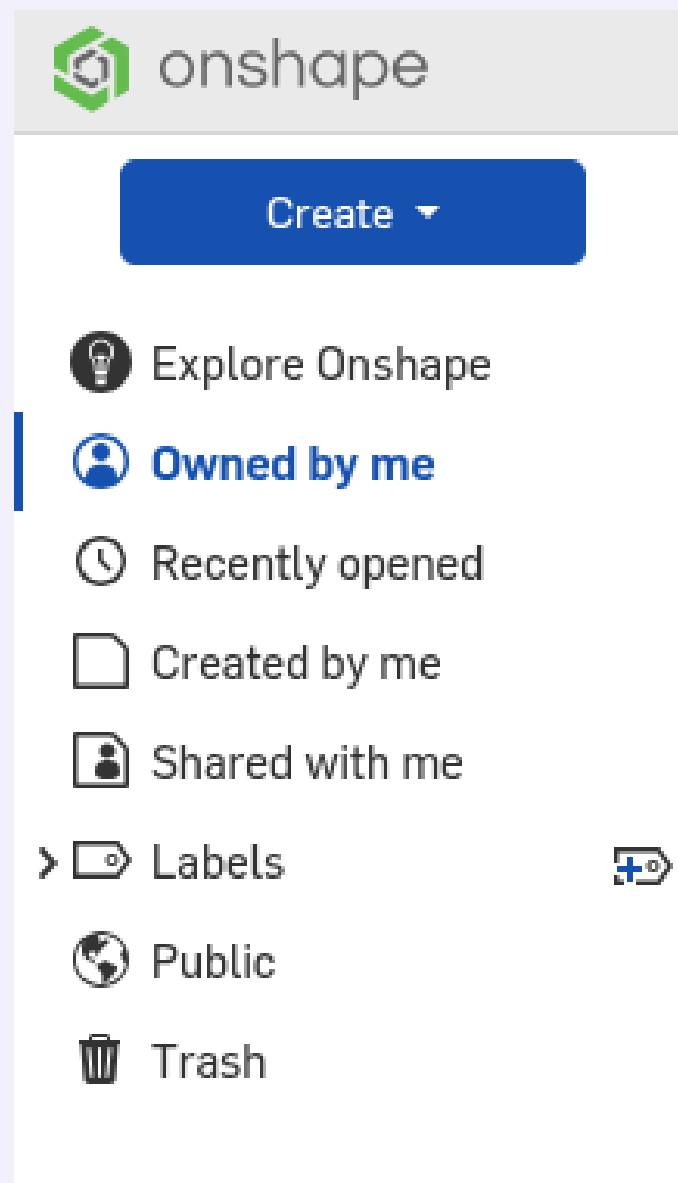
Last name*

Email Address*

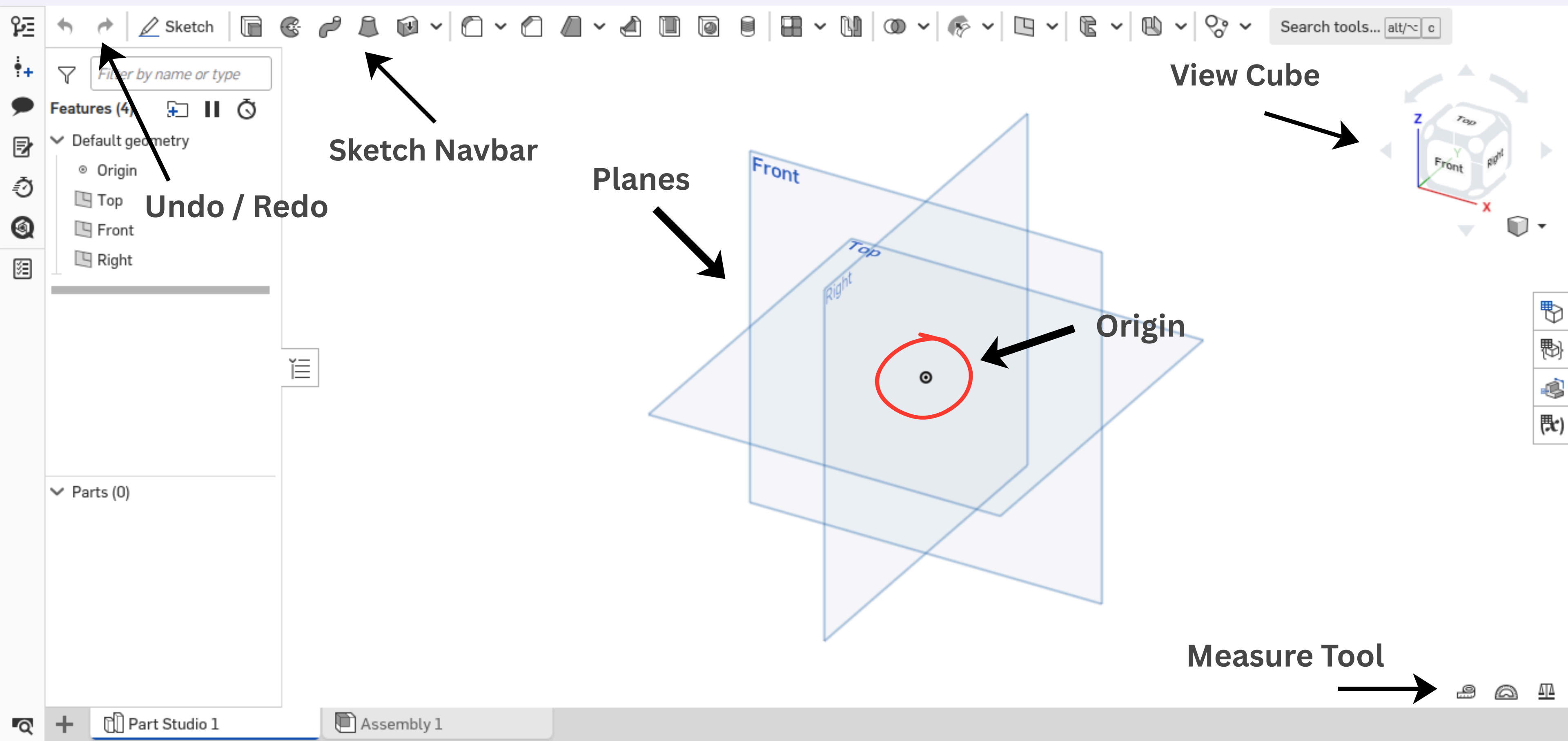
SUBMIT

Step 2

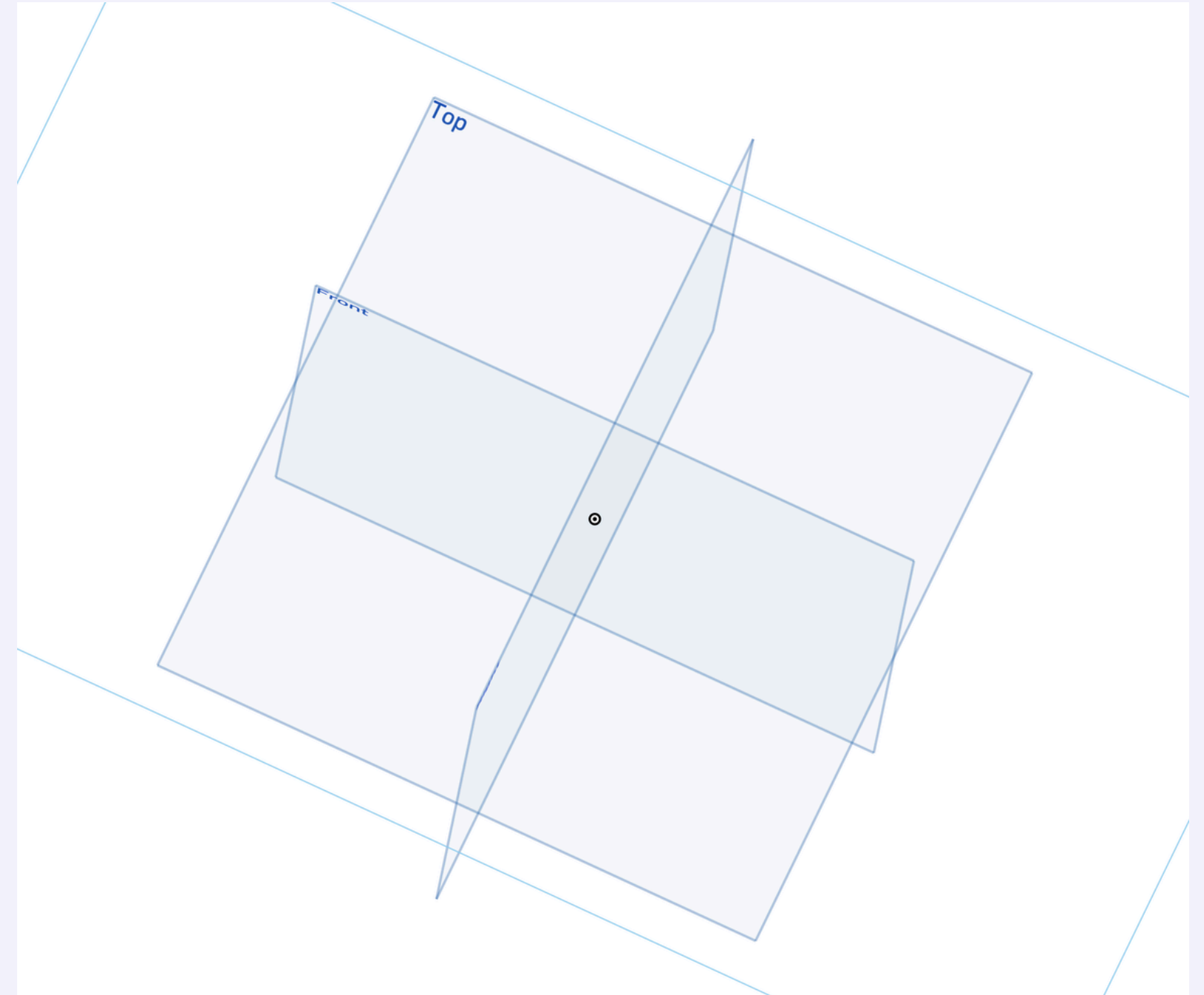
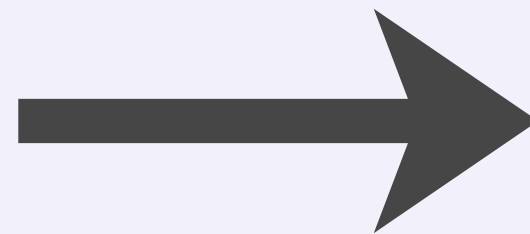
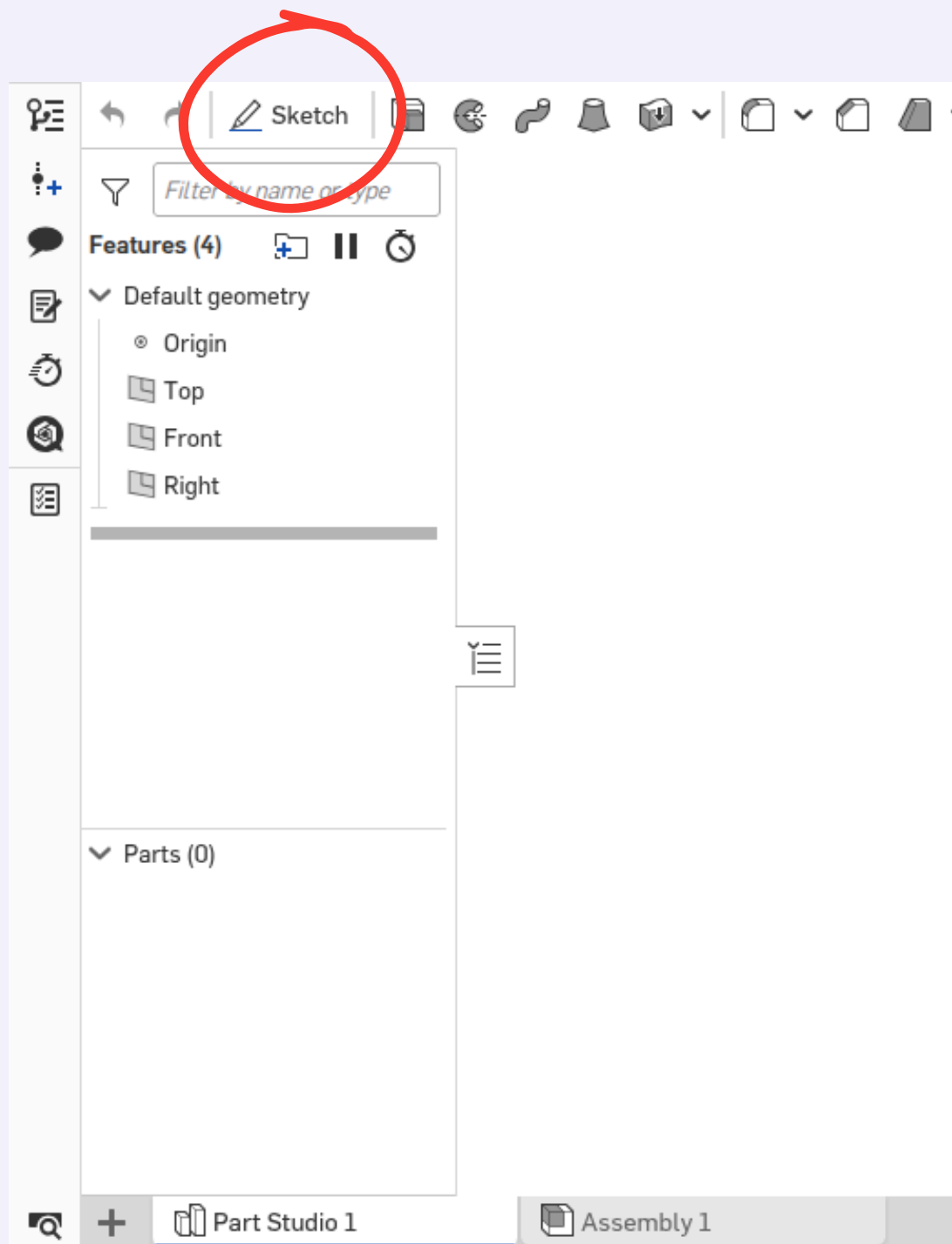
Open a new sketch!



UI Overview



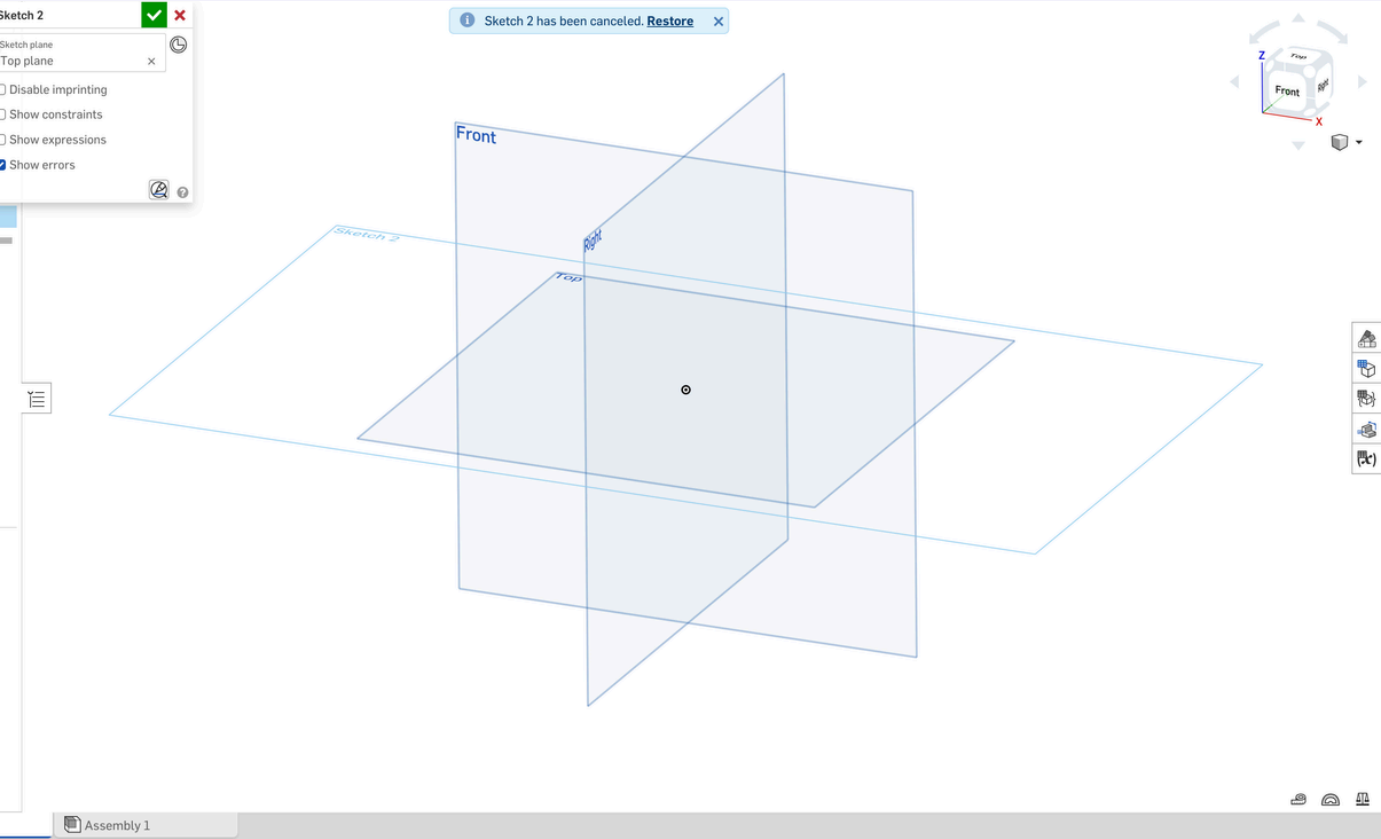
Sketching



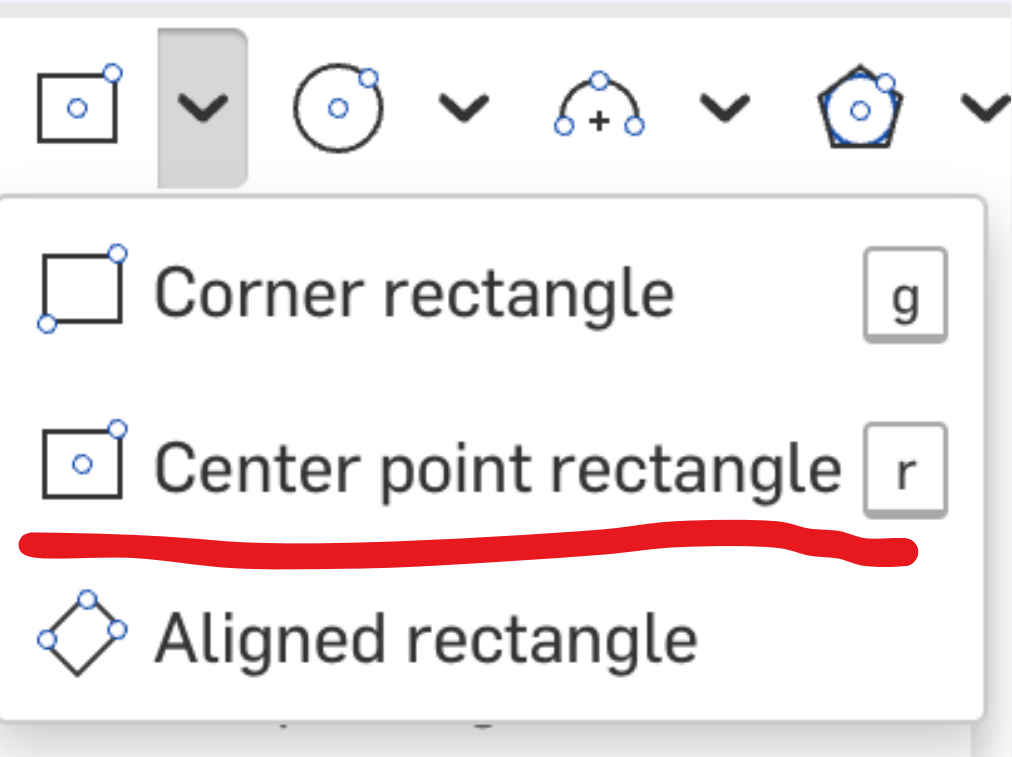
Sketch Navbar



1. After clicking sketch, click Top Plane

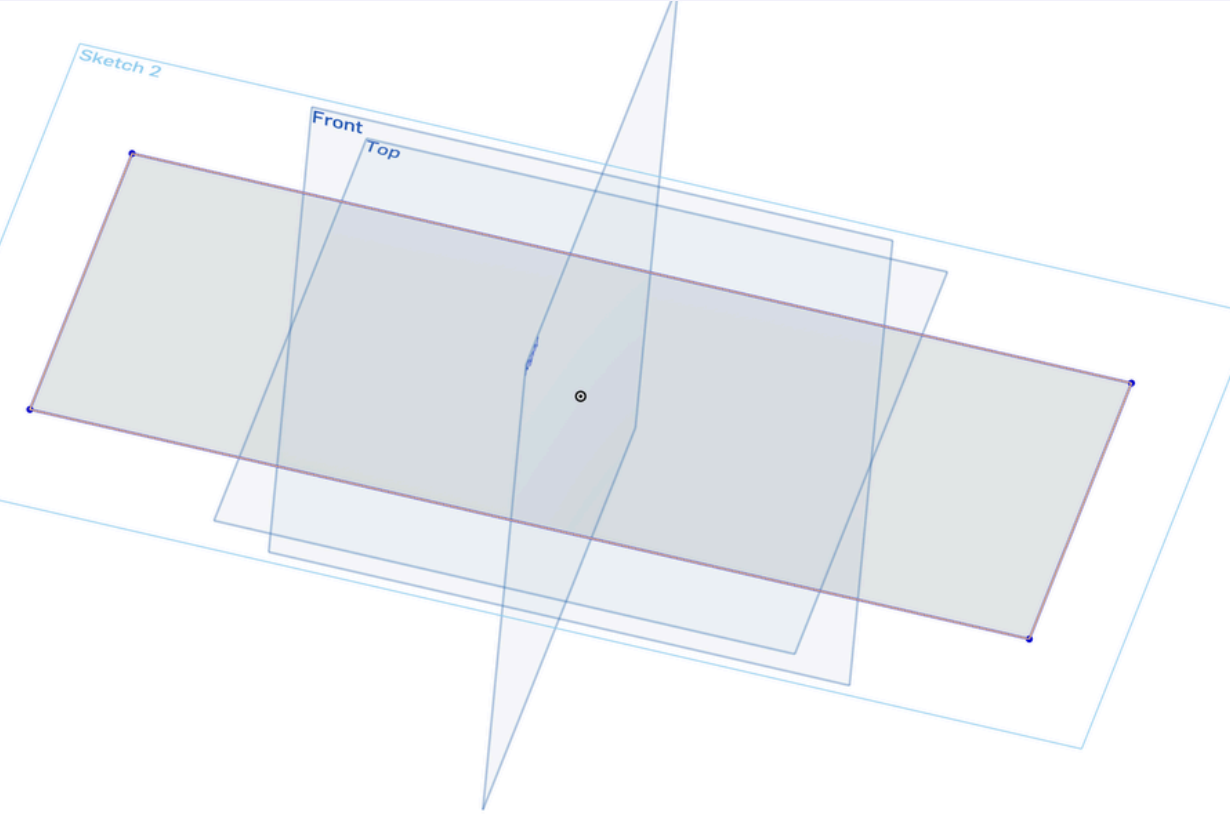


2. Click Center Point Rectangle

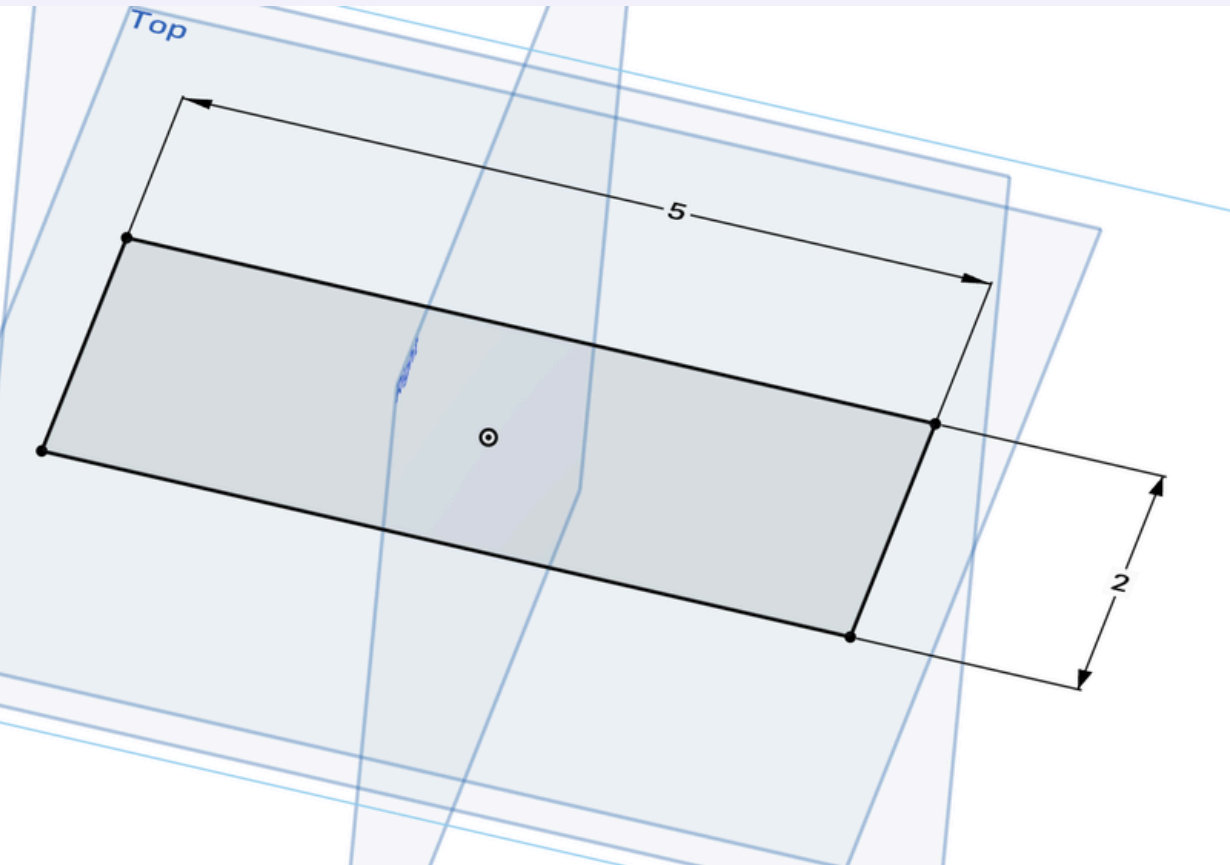


Note: You can also work with circles, polygons, etc.

3. Dimensioning (D for keyboard shortcut)

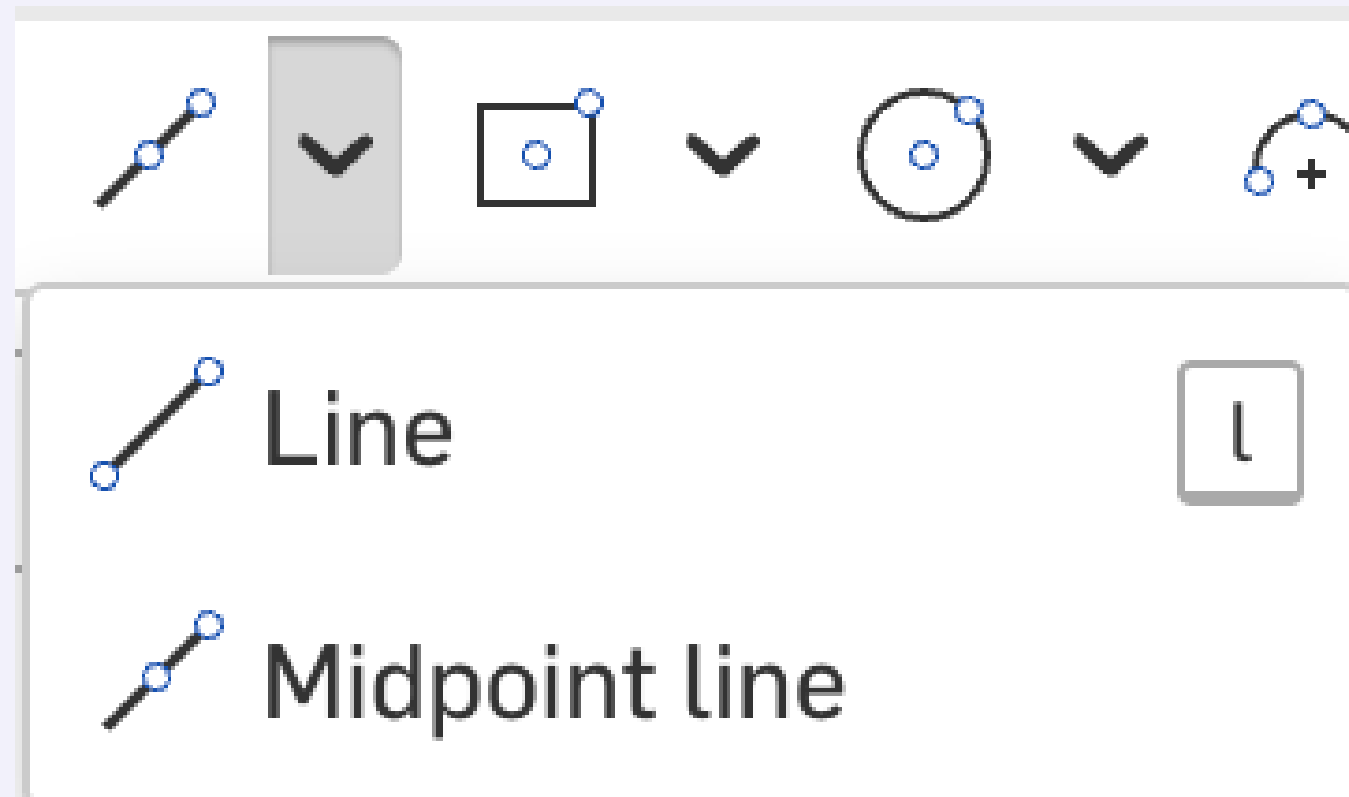


4. Press green checkmark once done. Sketch Result!



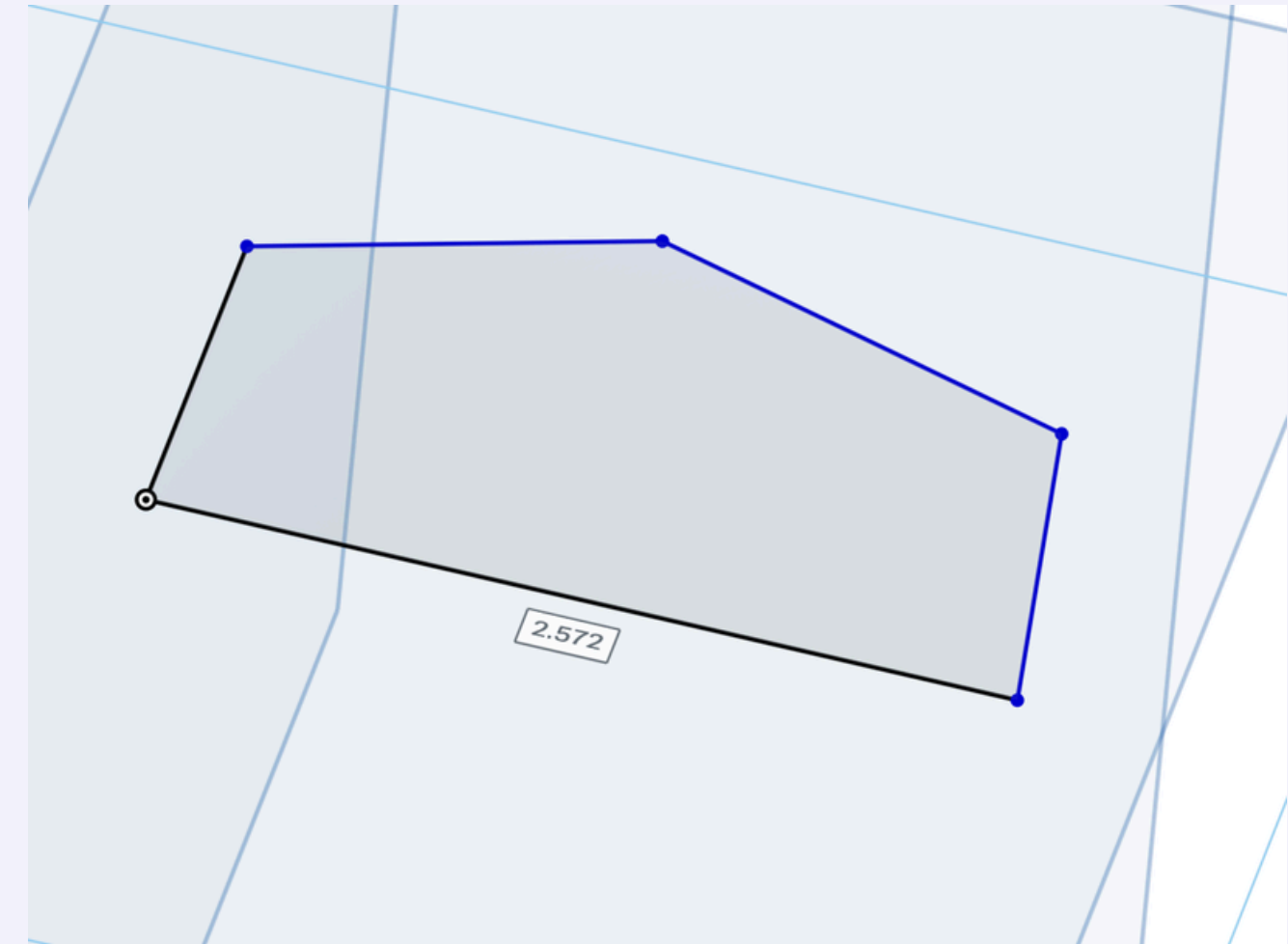
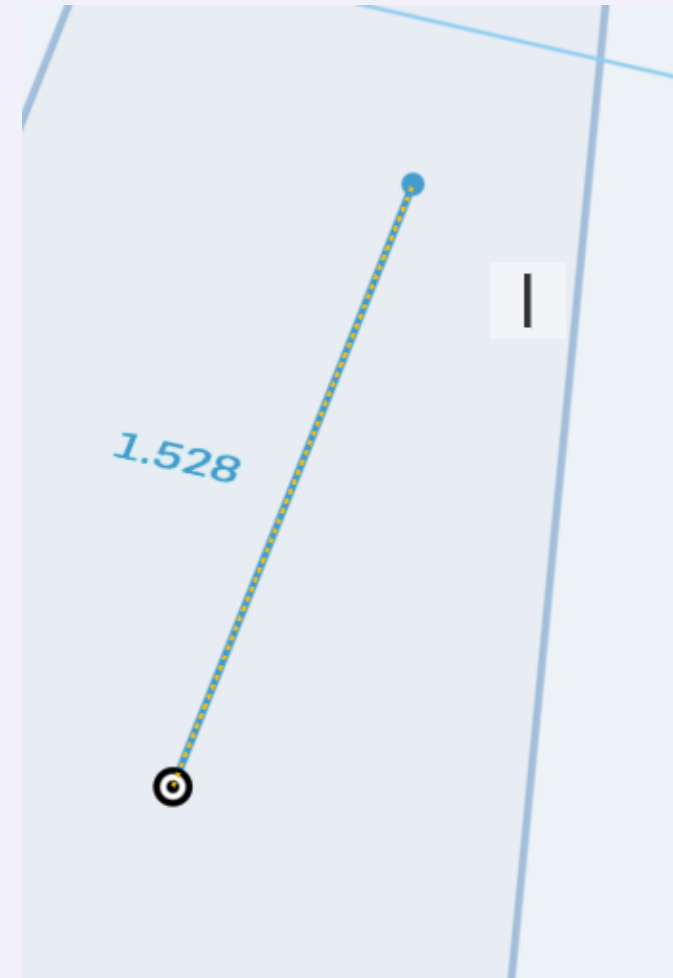
Sketching – Lines

1. Click Line dropdown, and then Line

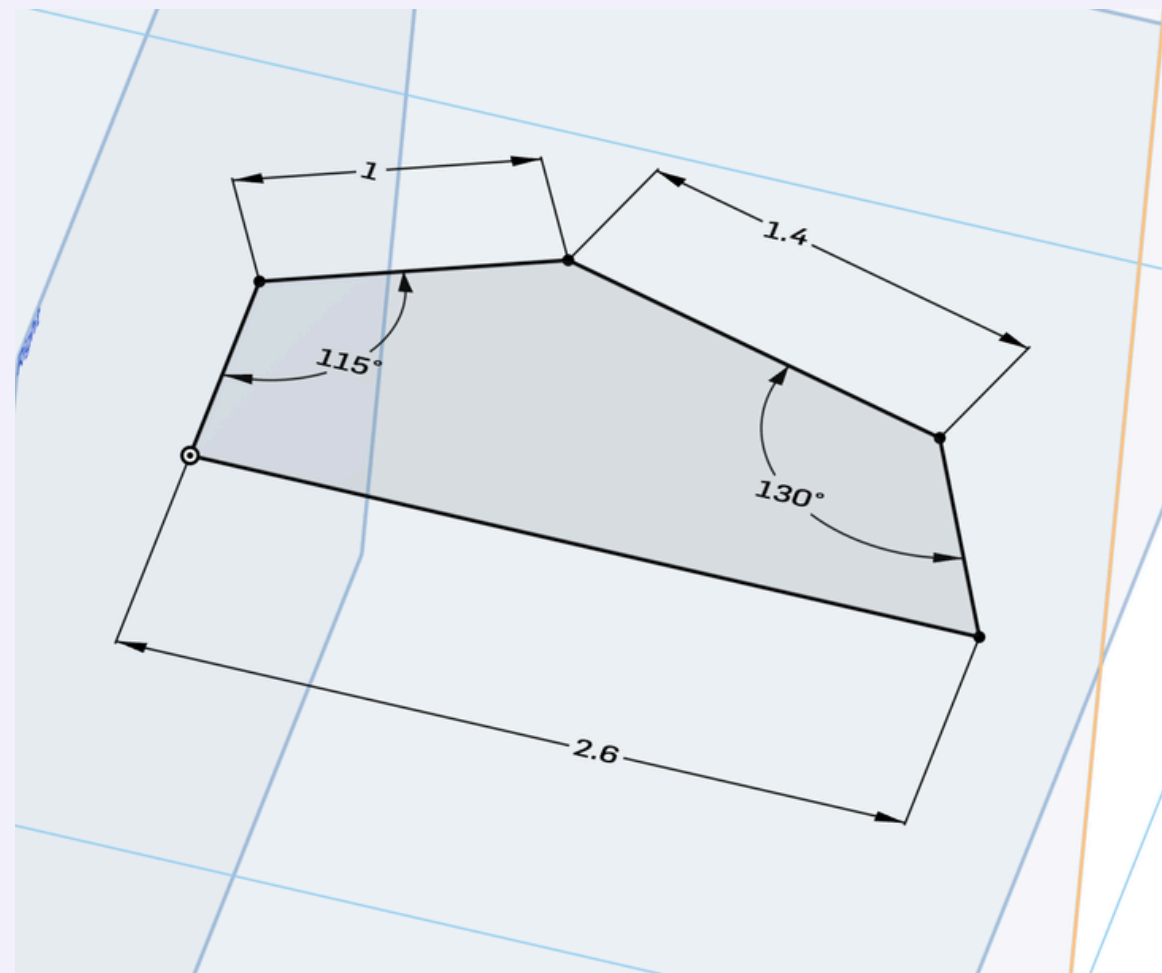


2. Click on Origin, then you can drag a line anywhere.

- Number on left shows length
 - Blue line means undefined
 - Black line means defined
- Line should turn grey and click into place if straight

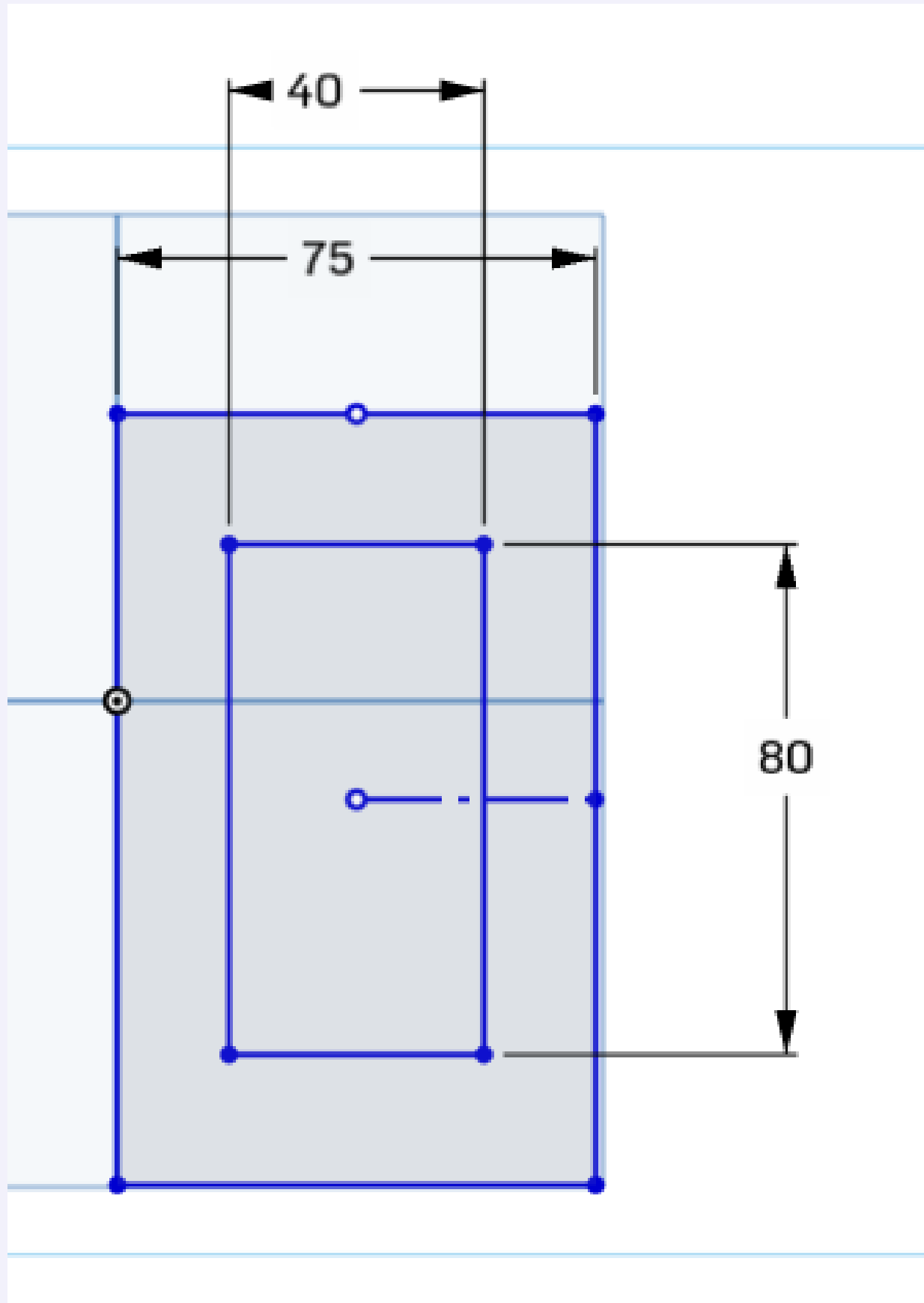


You can create any type of shape with lines!

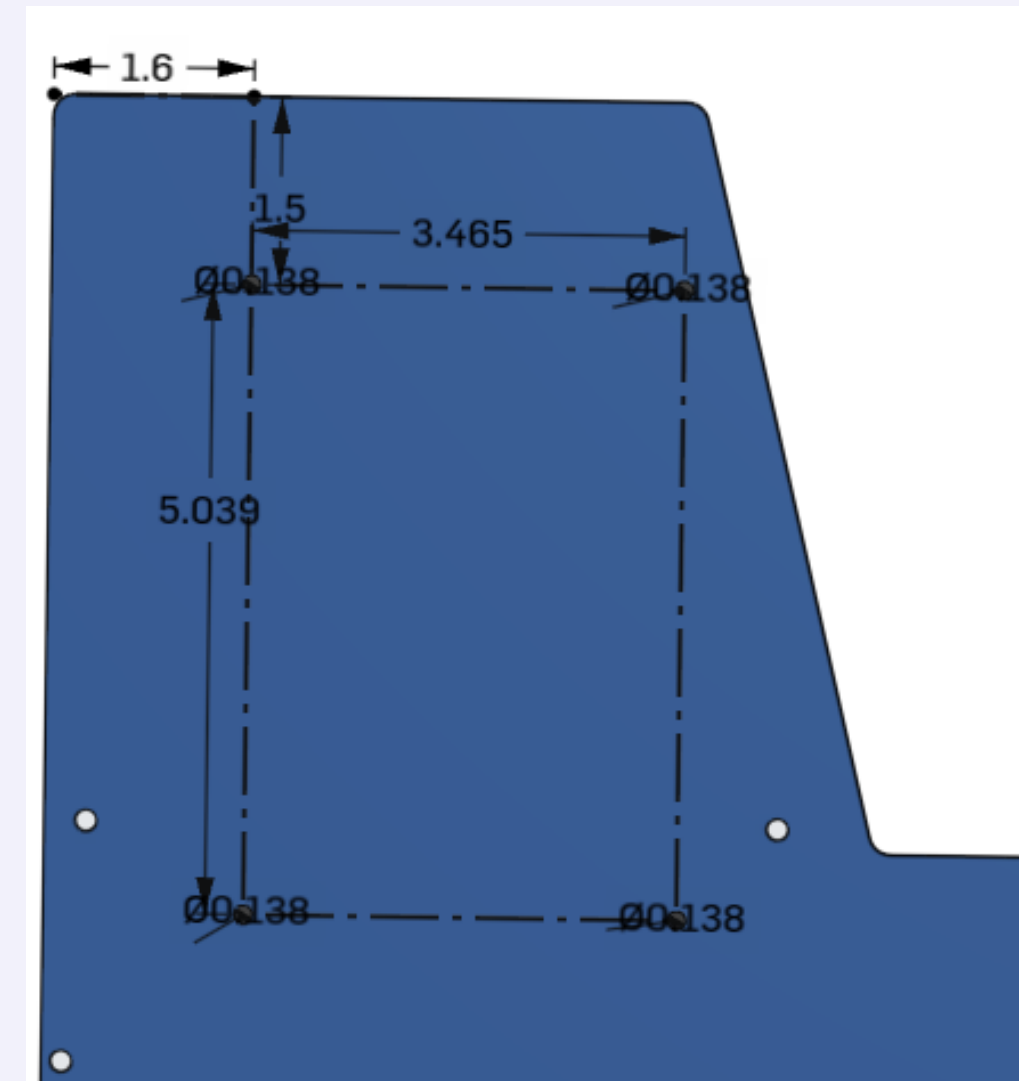


- Change the length of each line with dimension tool
- Change angle between lines by selecting two lines joined together by a vertex

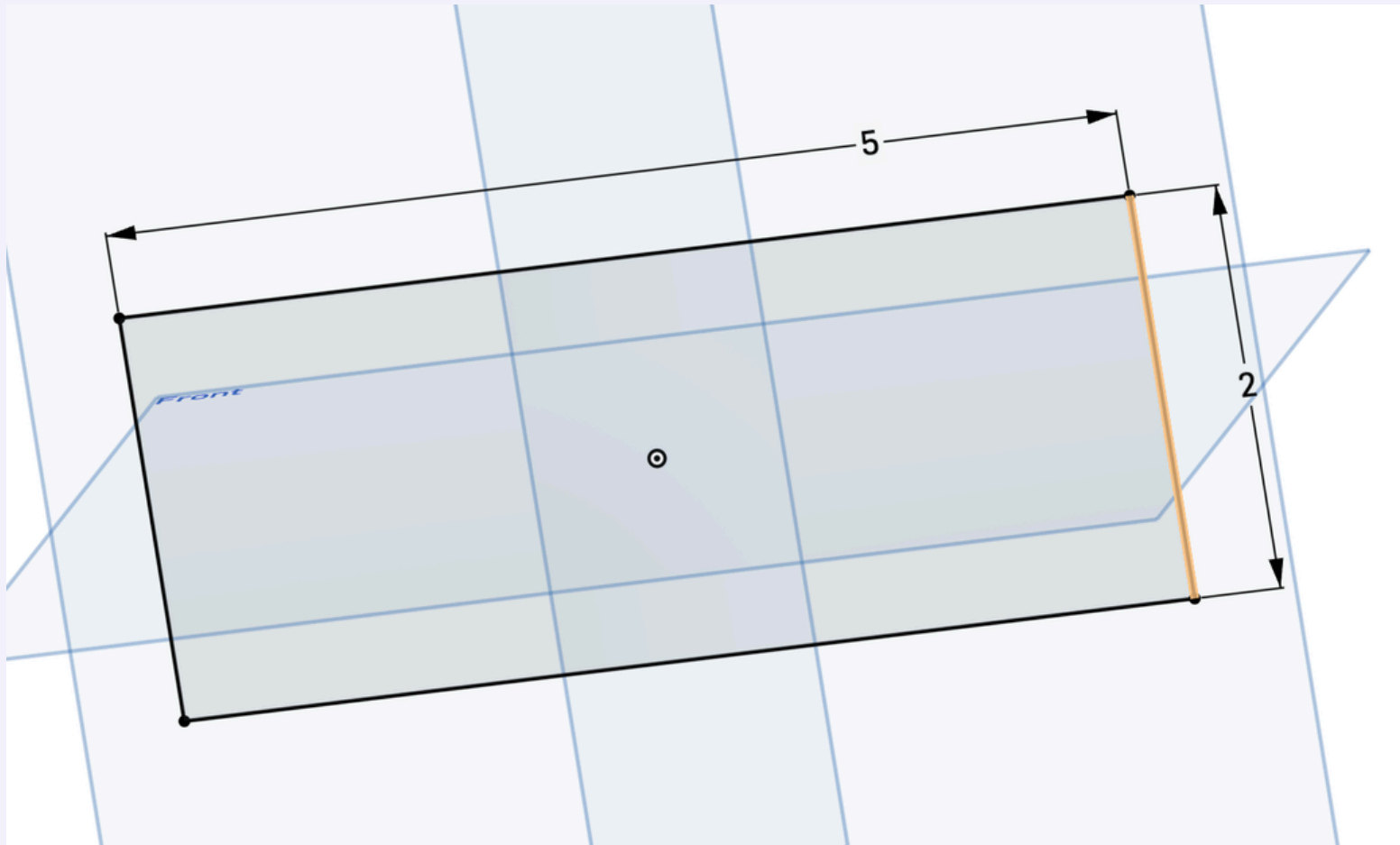
Construction Lines



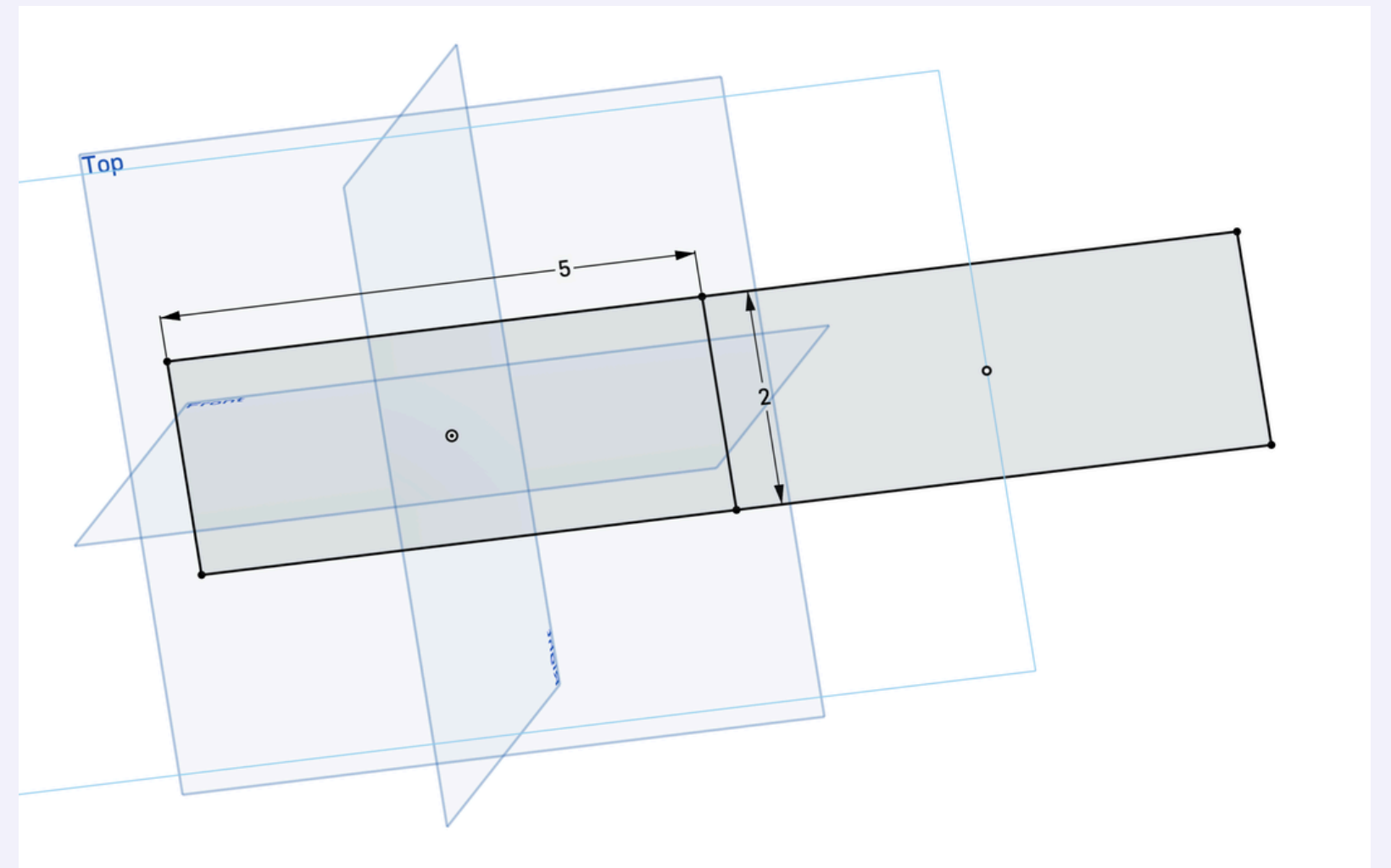
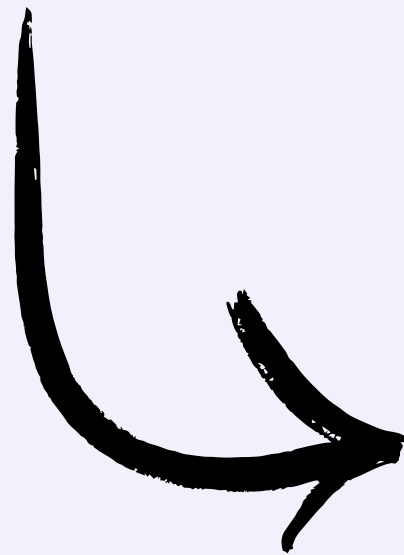
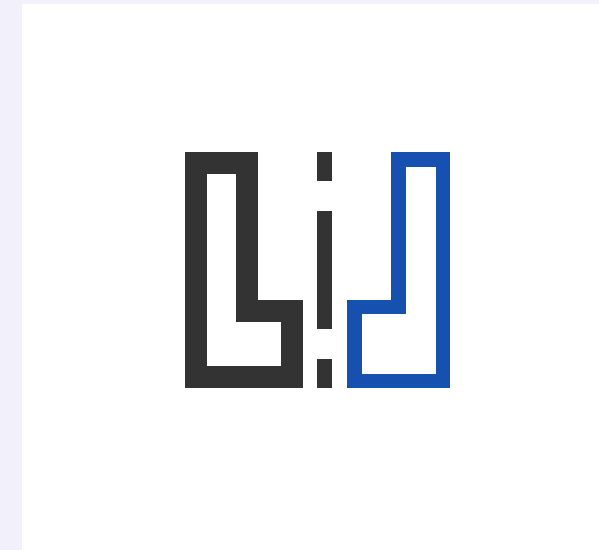
- What they are: Helper lines for sketching—they guide your design without turning into 3D shapes.
- Shortcut: press Q to toggle any line or drawing tool on or off.
- Best for setting up symmetry, center axes, and placing hole patterns cleanly.



Sketching – Mirror



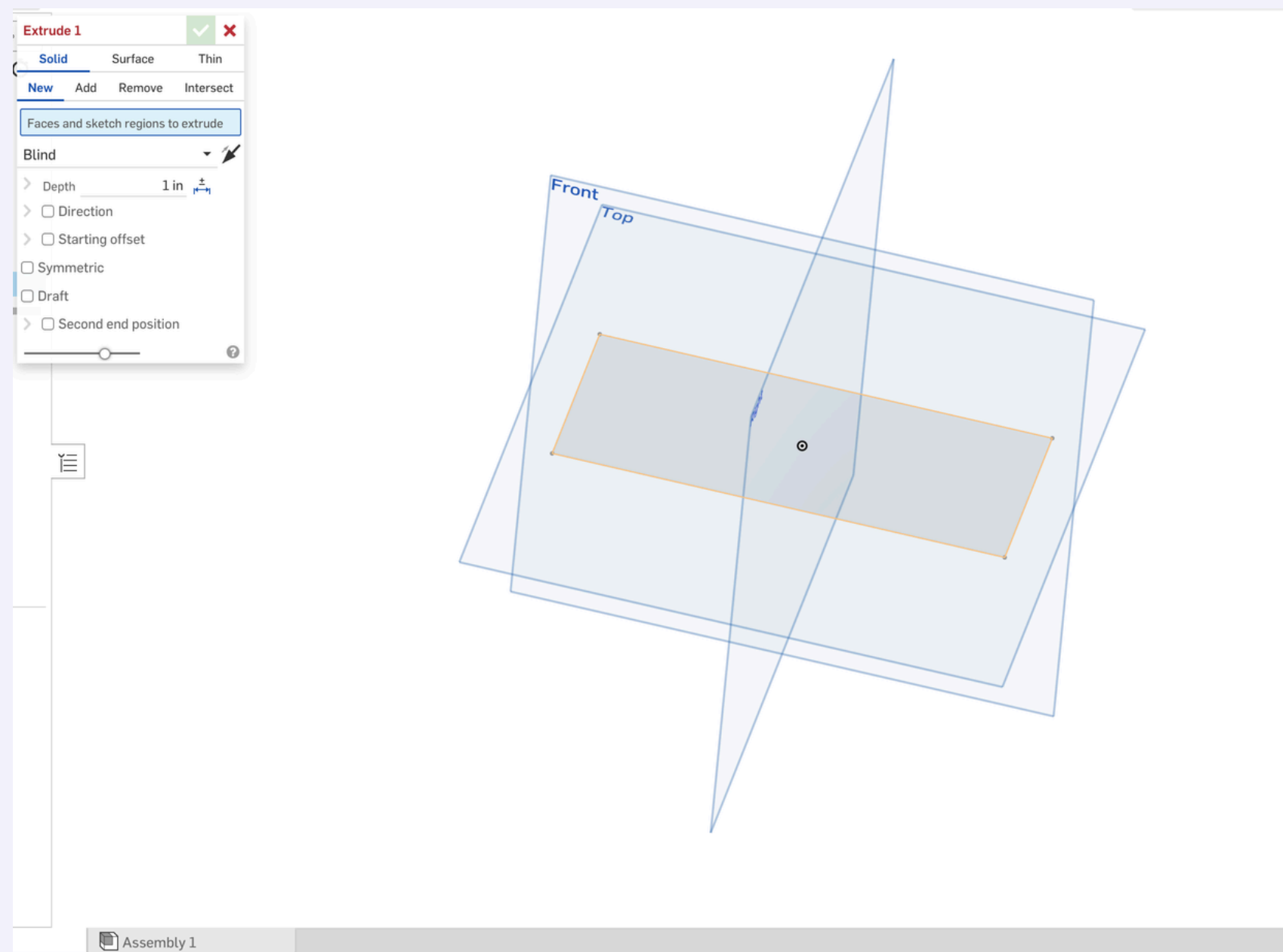
Mirror Icon



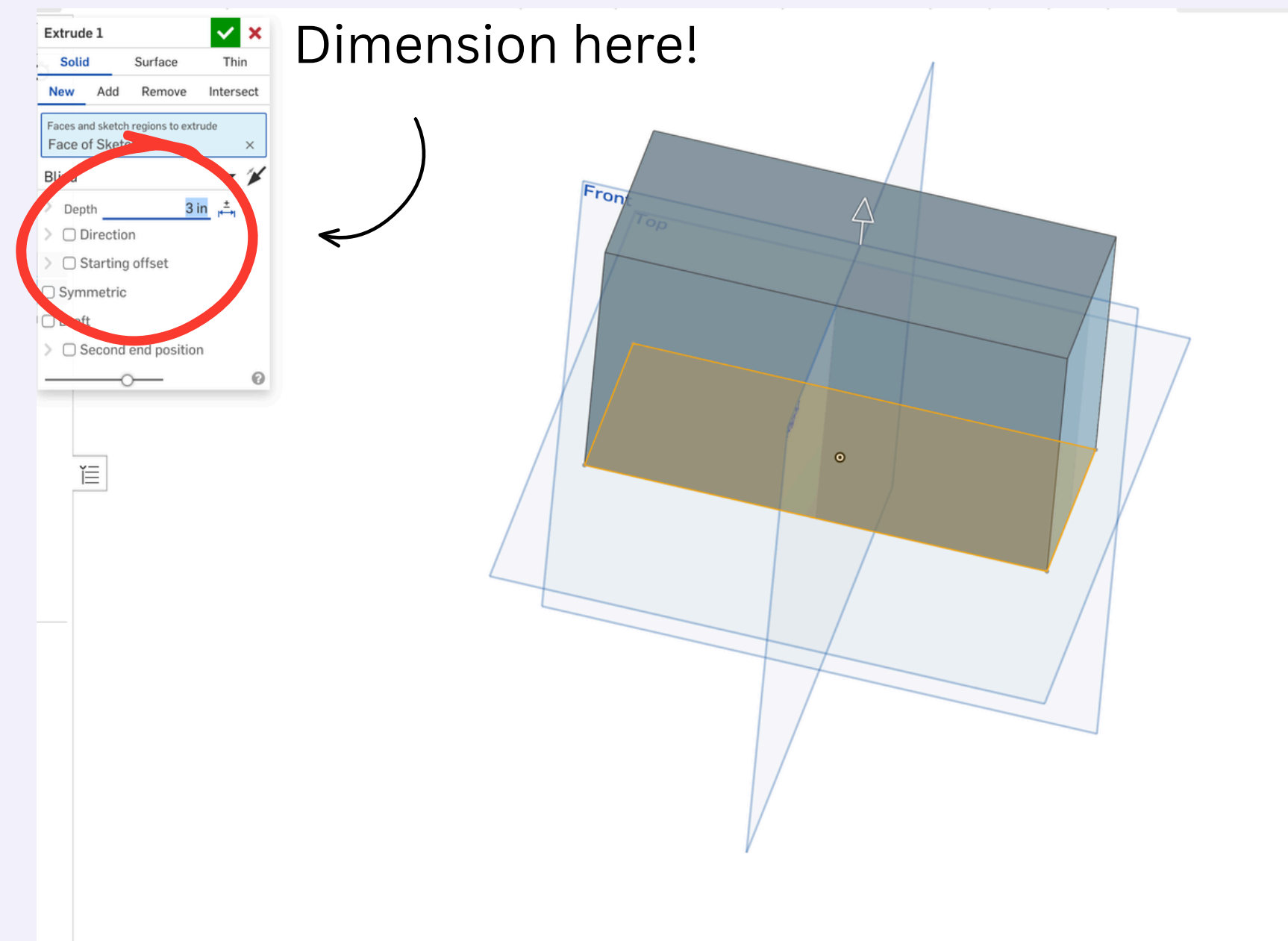
Extrusions



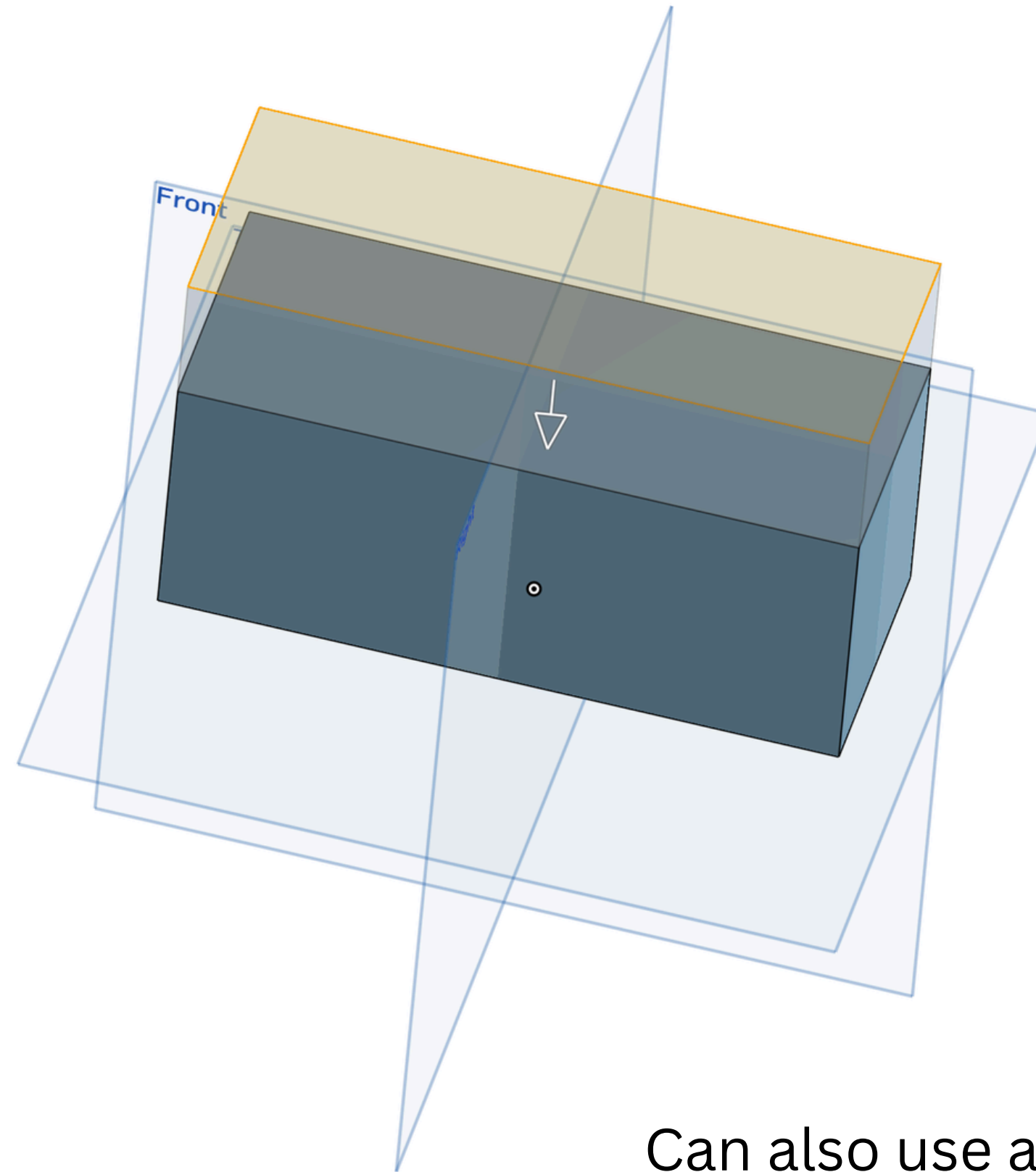
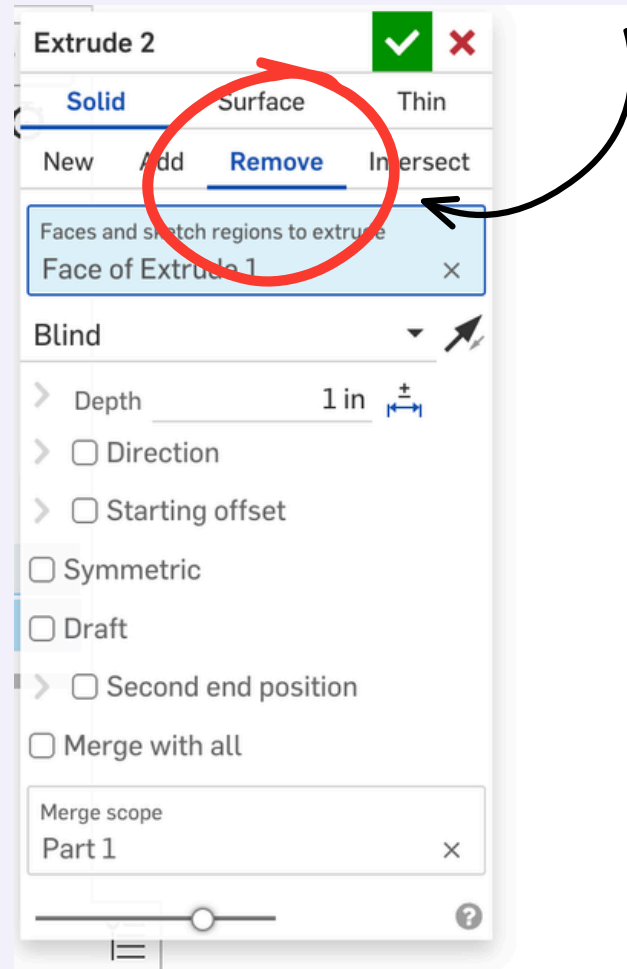
Click on the surface you want to extrude



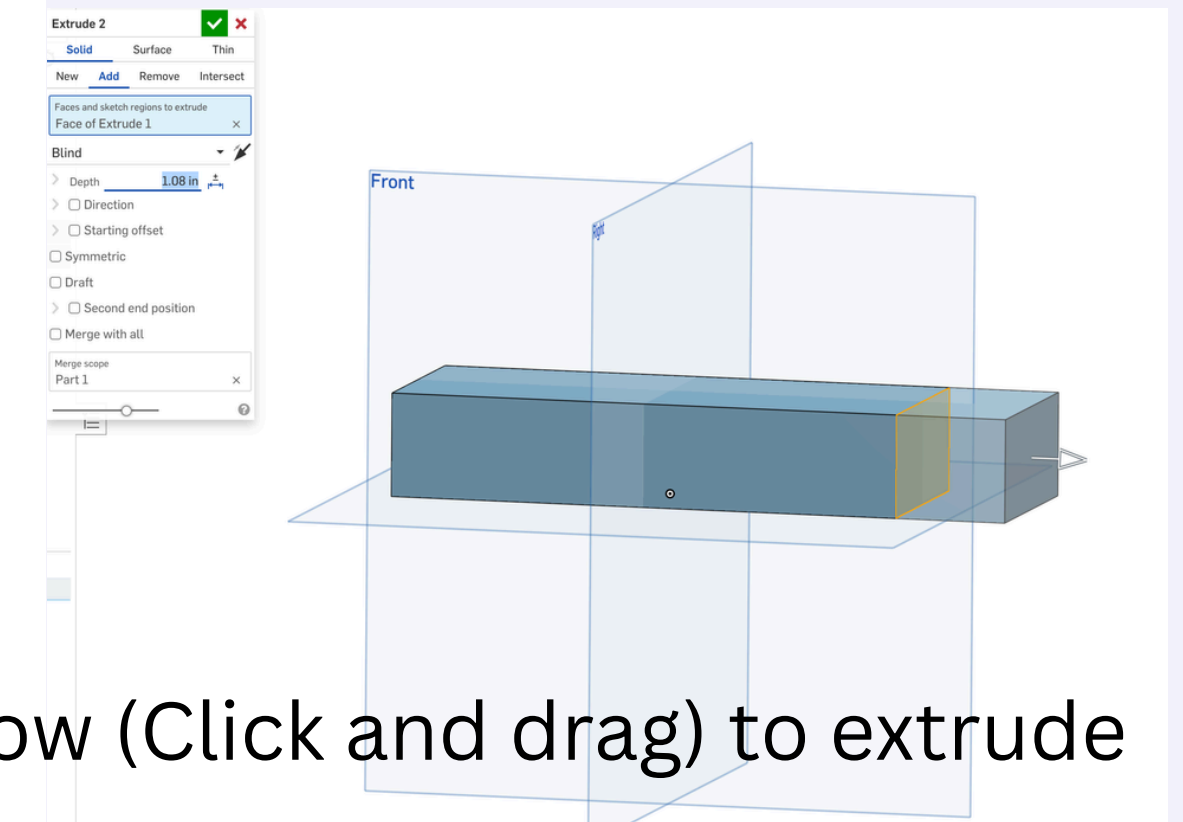
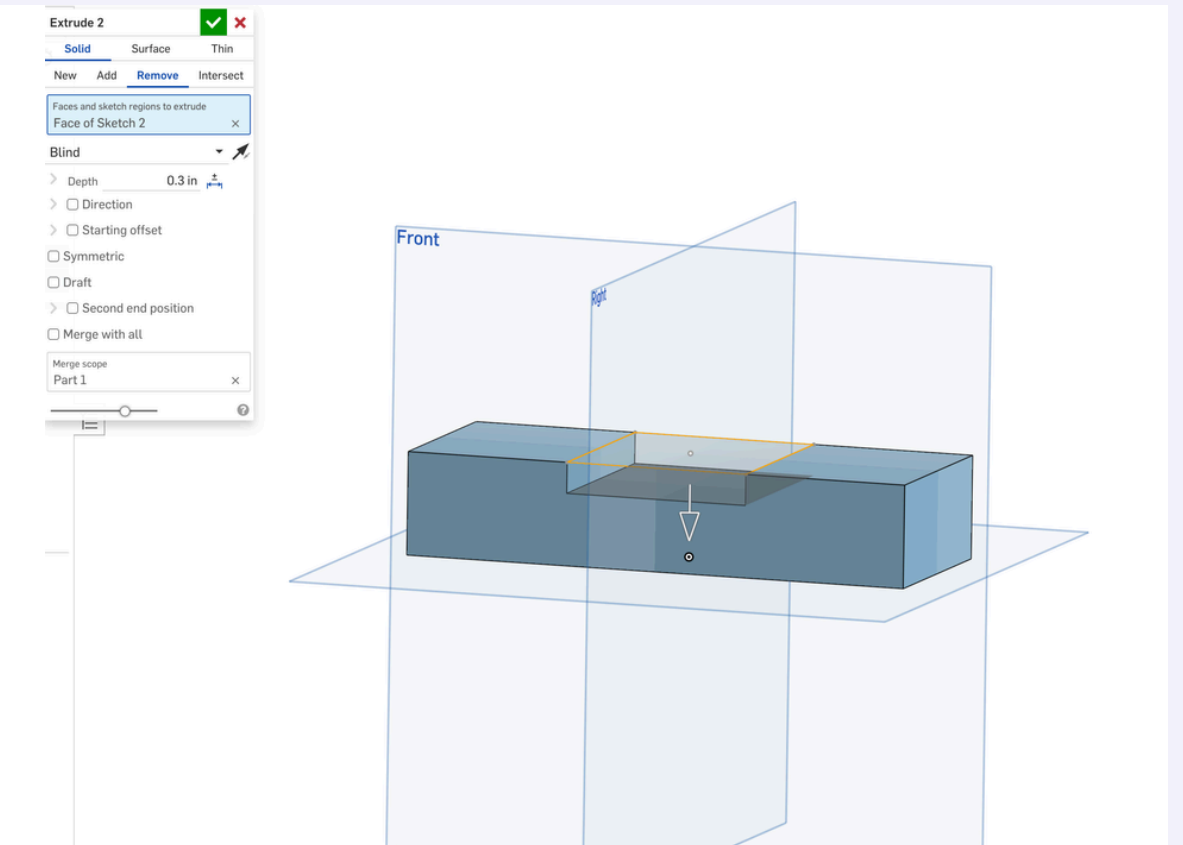
Make sure to leave Sketch mode for this!



Remove Material

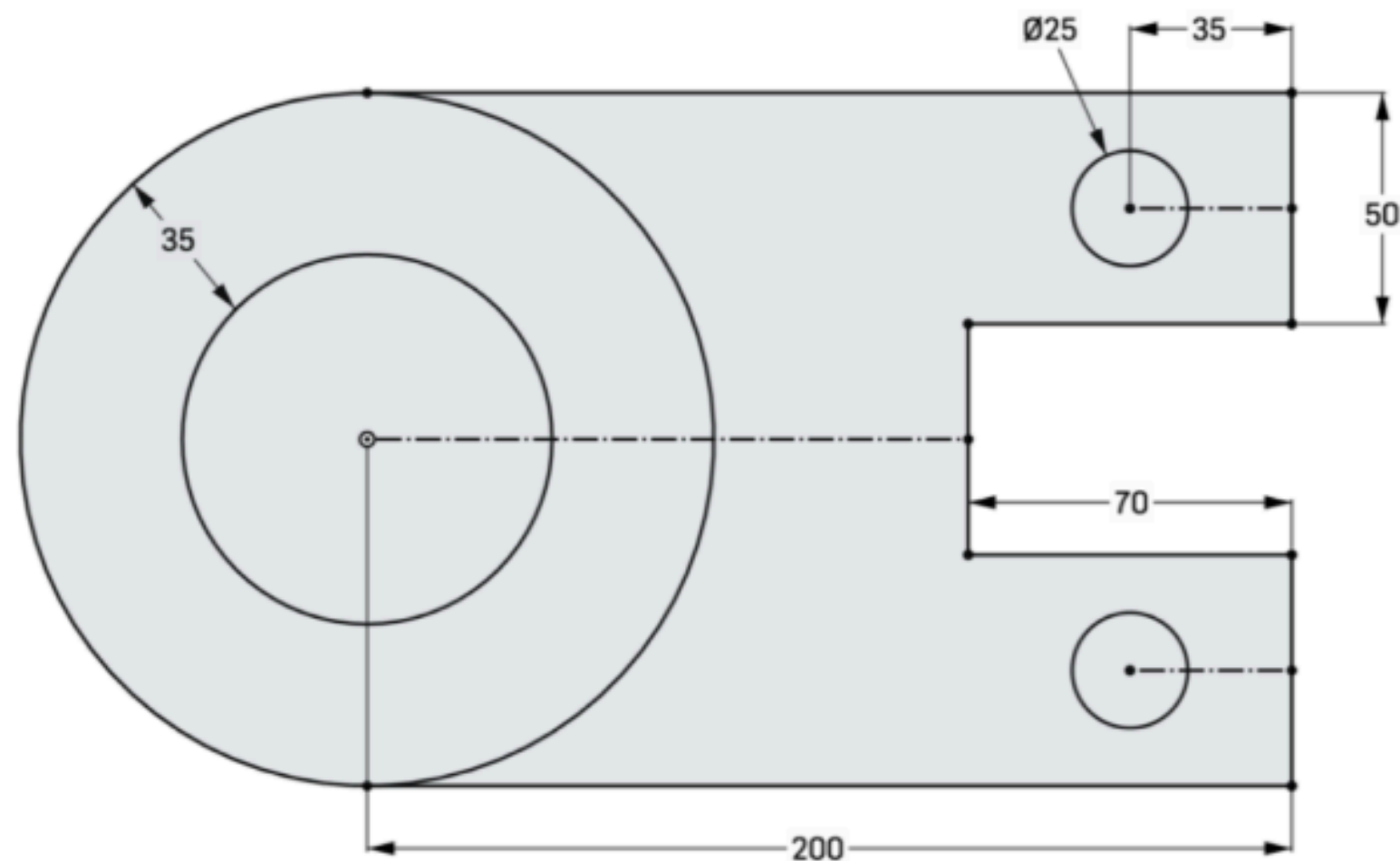


Extrude on any surface (Based upon sketches)



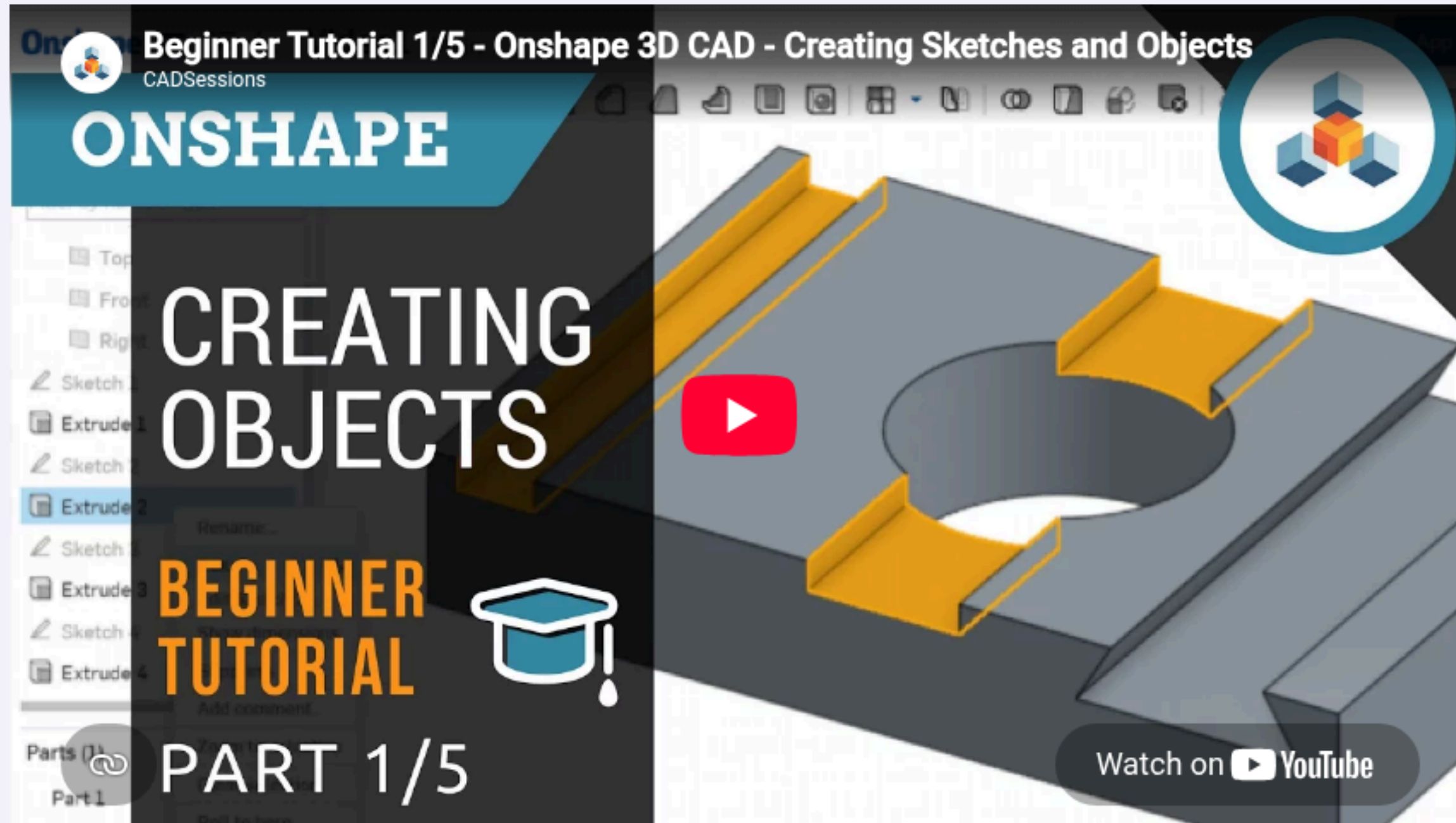
Can also use arrow (Click and drag) to extrude

Guided Tutorial – Follow Along!

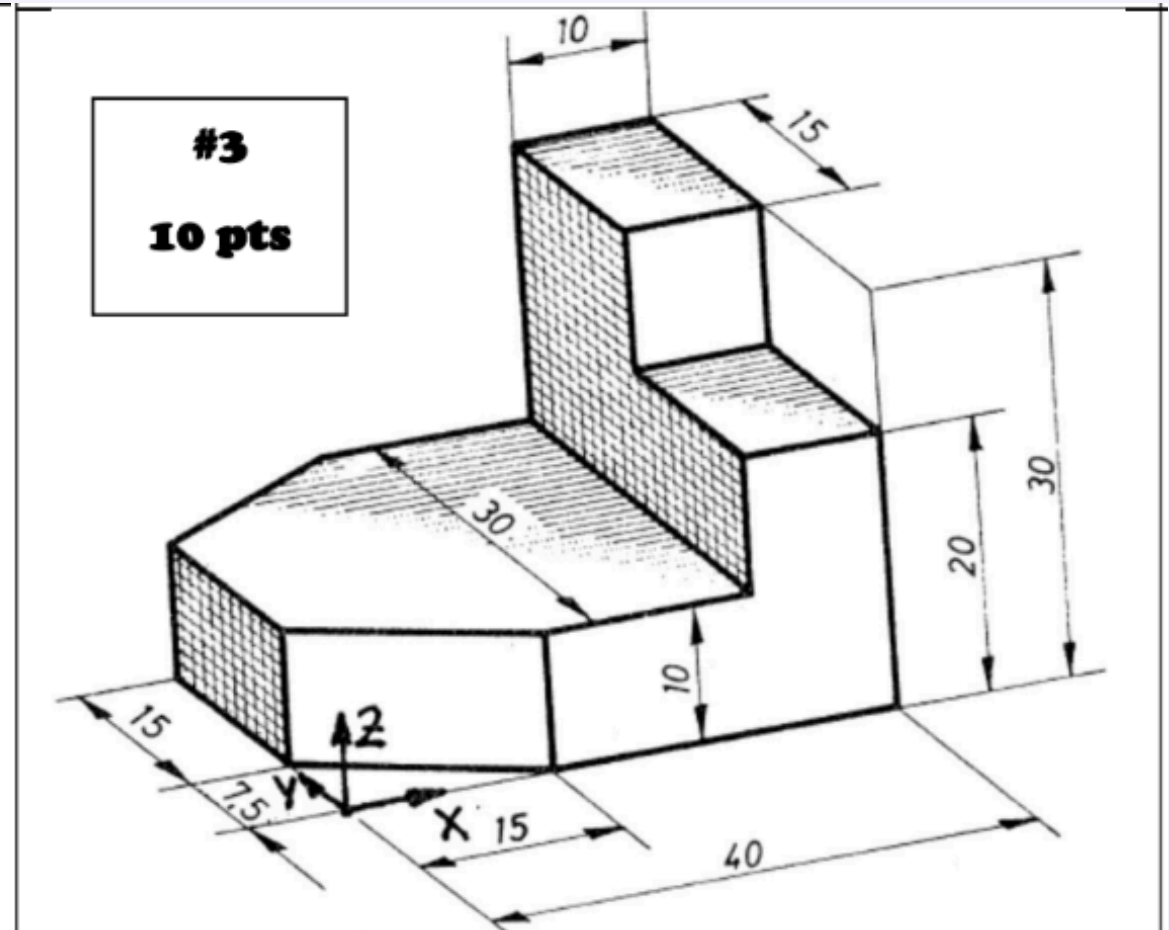
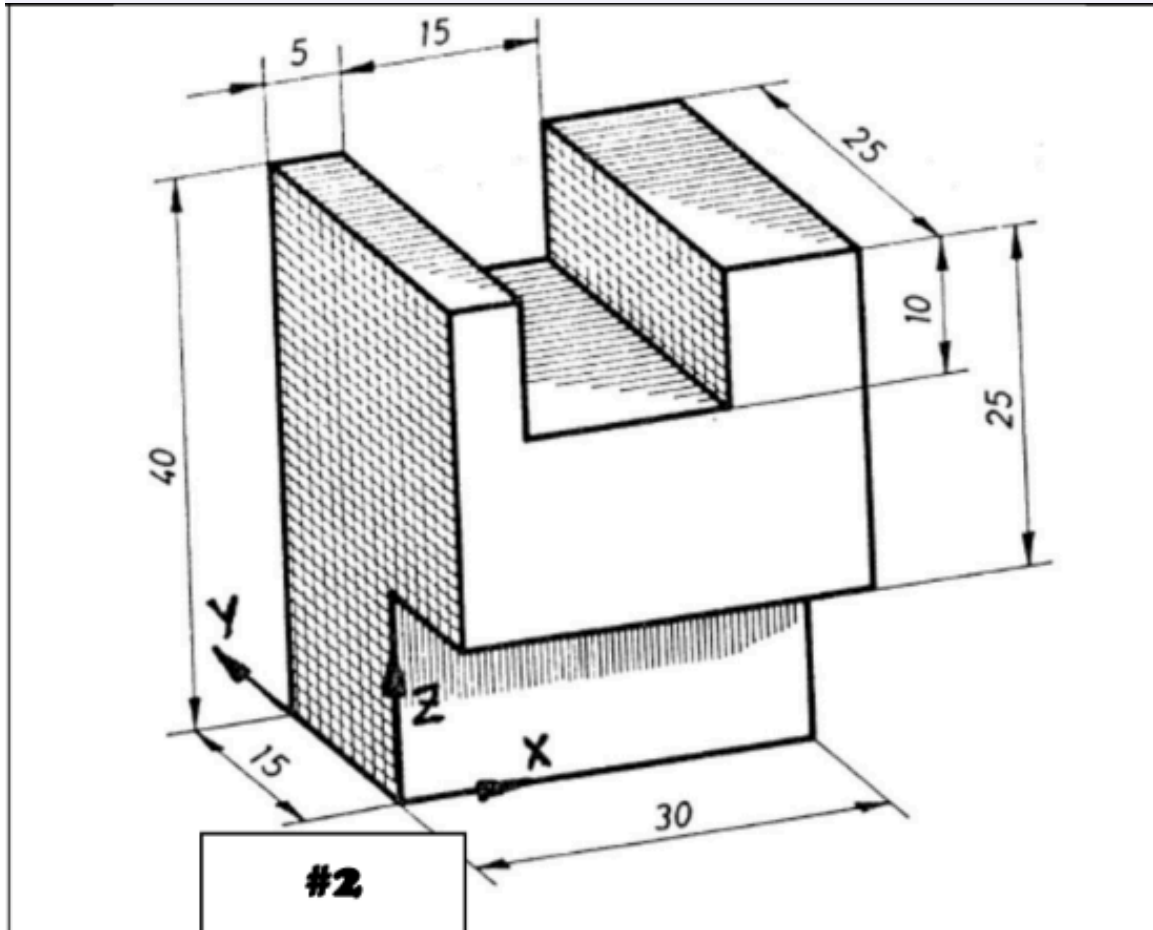
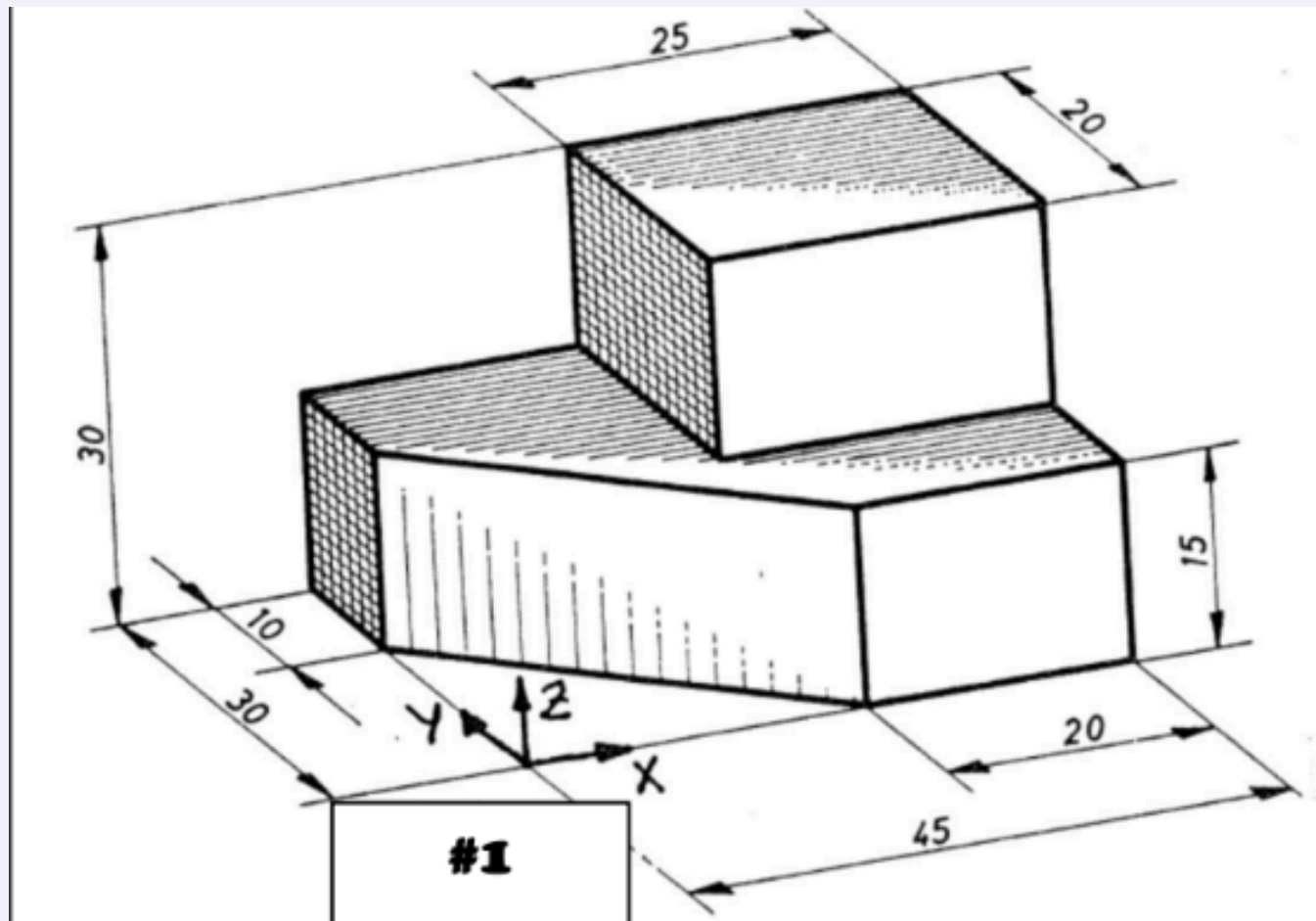


Follow along to learn how to sketch this

Youtube Tutorial



Independent Practice



CAD these models!