FISH @ UVA

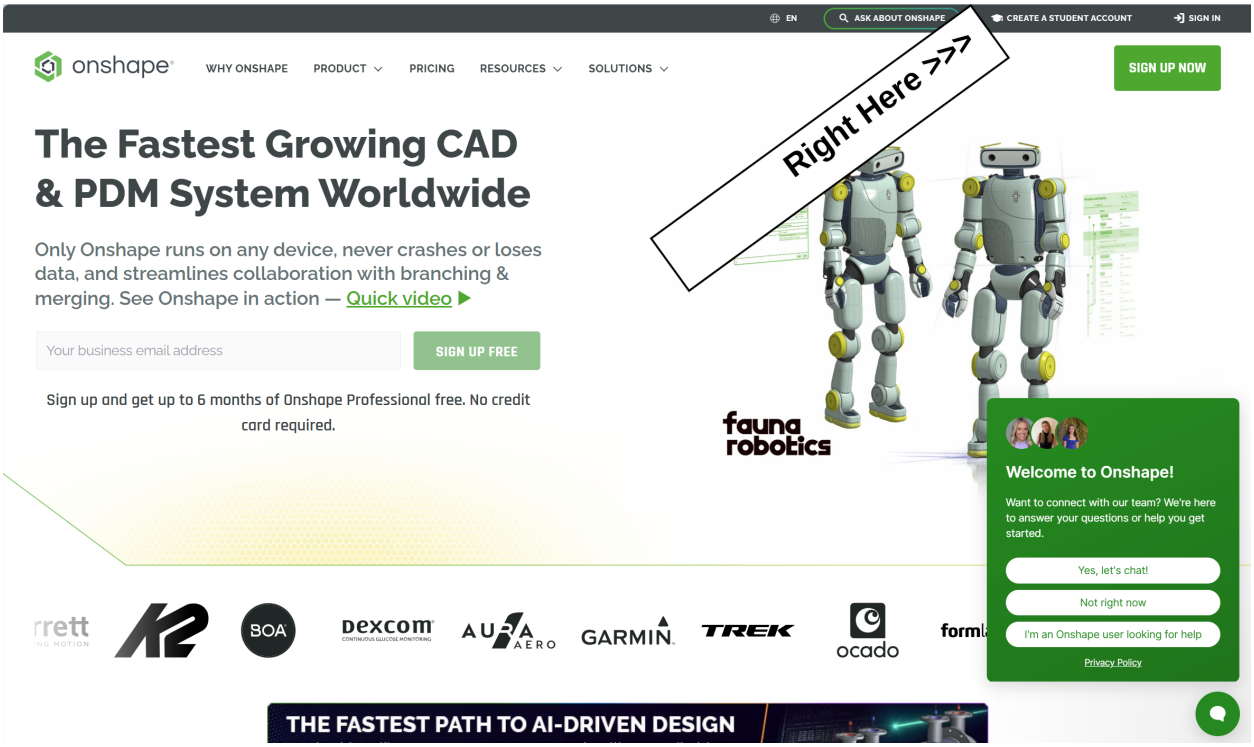
Introduction to CAD on OnShape

Medical Device Make-A-Thon | November 5–8, 2026 | University of Virginia

Setting up Onshape for the First Time

OnShape is an online Computer-Aided Design (CAD) software used mostly by students and hobbyists, although its industry usage has grown rapidly in recent years. It is a powerful and fully-functioning system, but with the added benefit of being online (no download needed) and free (no money needed!)

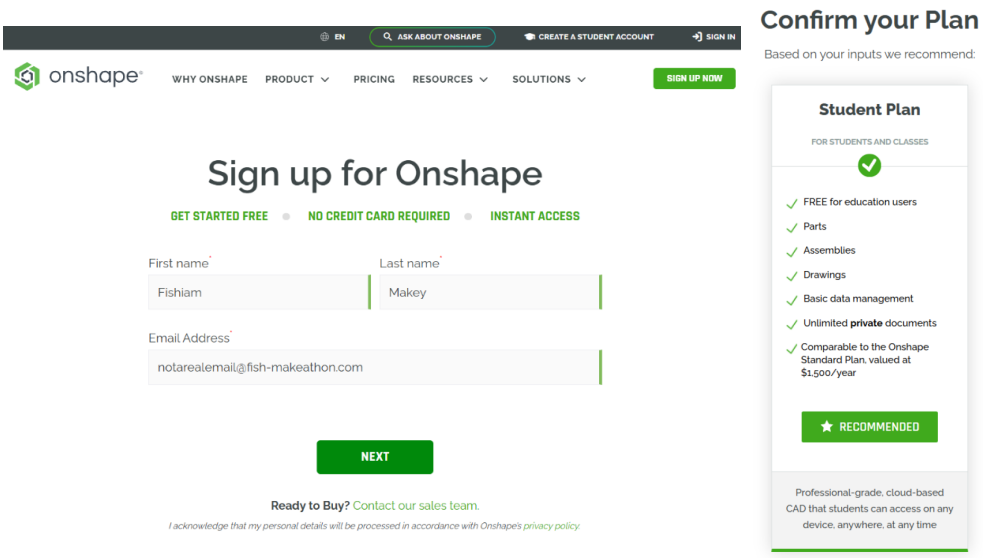
To set up an account, go to **www.onshape.com** and click the box at the top of the screen that says **Create a Student Account**



You can use either your school (@virginia.edu) email or a personal one, it shouldn't affect the process either way.

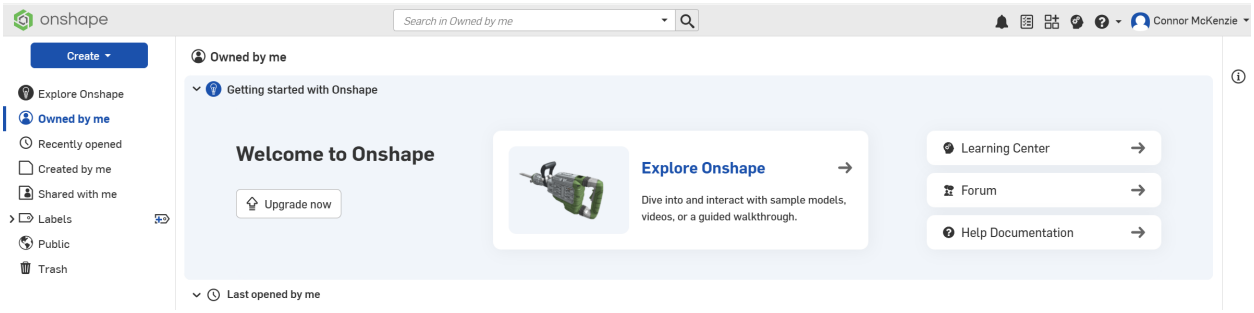
Make sure to select the free **Student Plan**! This will give you more features than the regular free plan. For school name, select **University of Virginia**, and for website enter *www.virginia.edu*.

Fill in the information it asks for to verify your eligibility as a student, and then check your email for a code or link to confirm your account. That's it!



Using Onshape

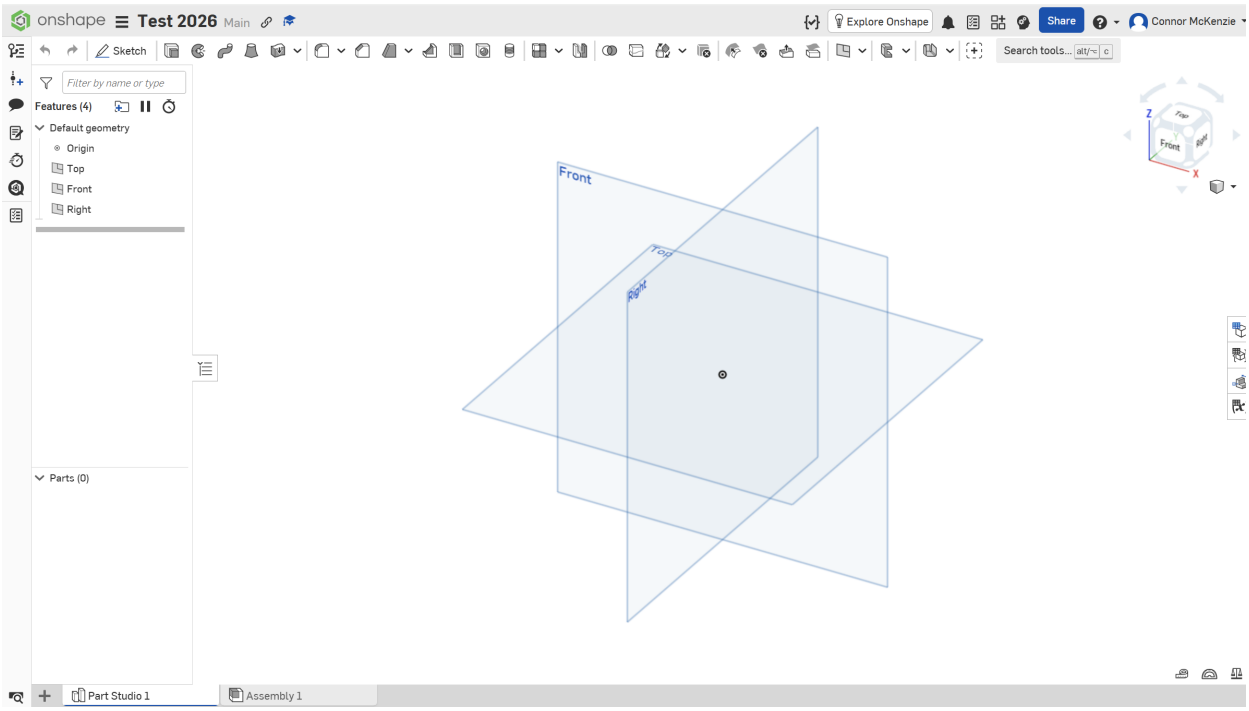
When you return to Onshape, make sure to log in with the email you signed up with originally!



You should arrive at this screen after logging in, which is where you will see any files you have worked on in the past and the folders you can create to organize your designs. There is also a large blue **Create** button in the top left corner, which you will use to start a design.

The **Create** button will show a drop-down menu with *Document* or *Folder* underneath. Folders show up on the home page and allow you to organize your documents, while documents are the actual design files you will be creating. Select **Document.**

Name the document whatever you want, and then you'll see this screen:



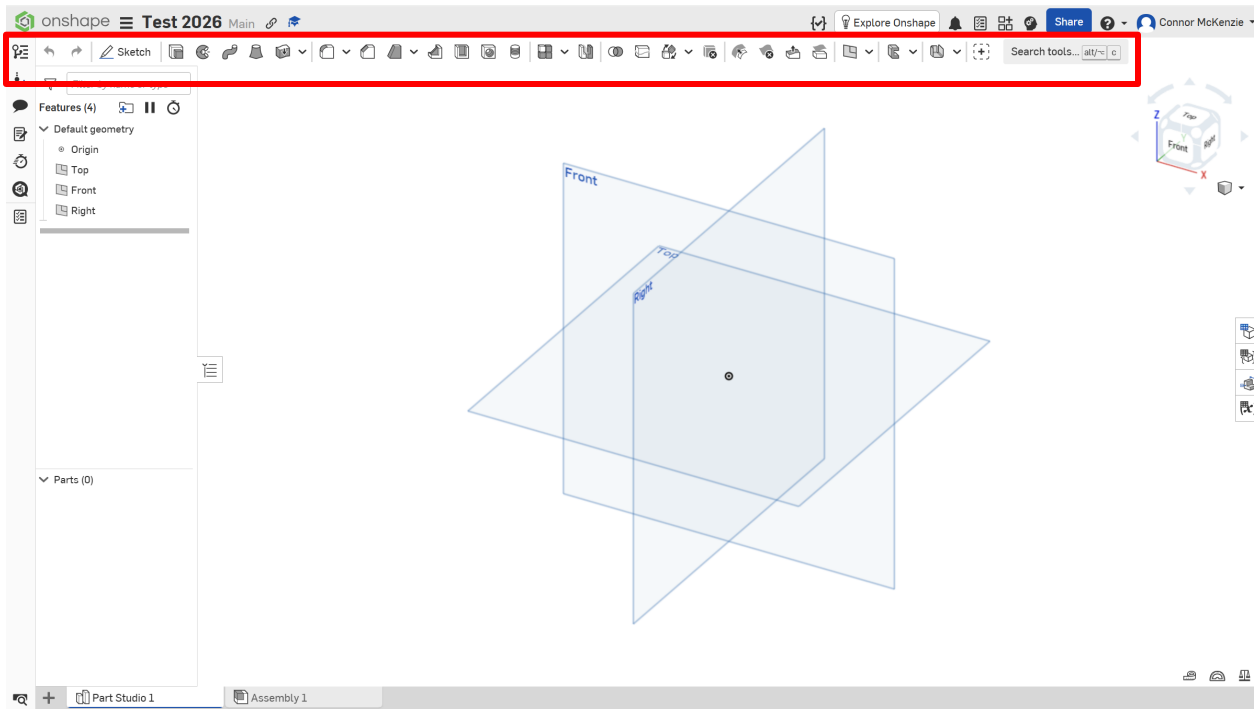
This is your main workspace, and will allow you to perform all of the functions needed to design a part.

The **Top**, **Front**, and **Right** planes are visible in the workspace at the moment, and are three orthogonal planes you can use to help start a design.

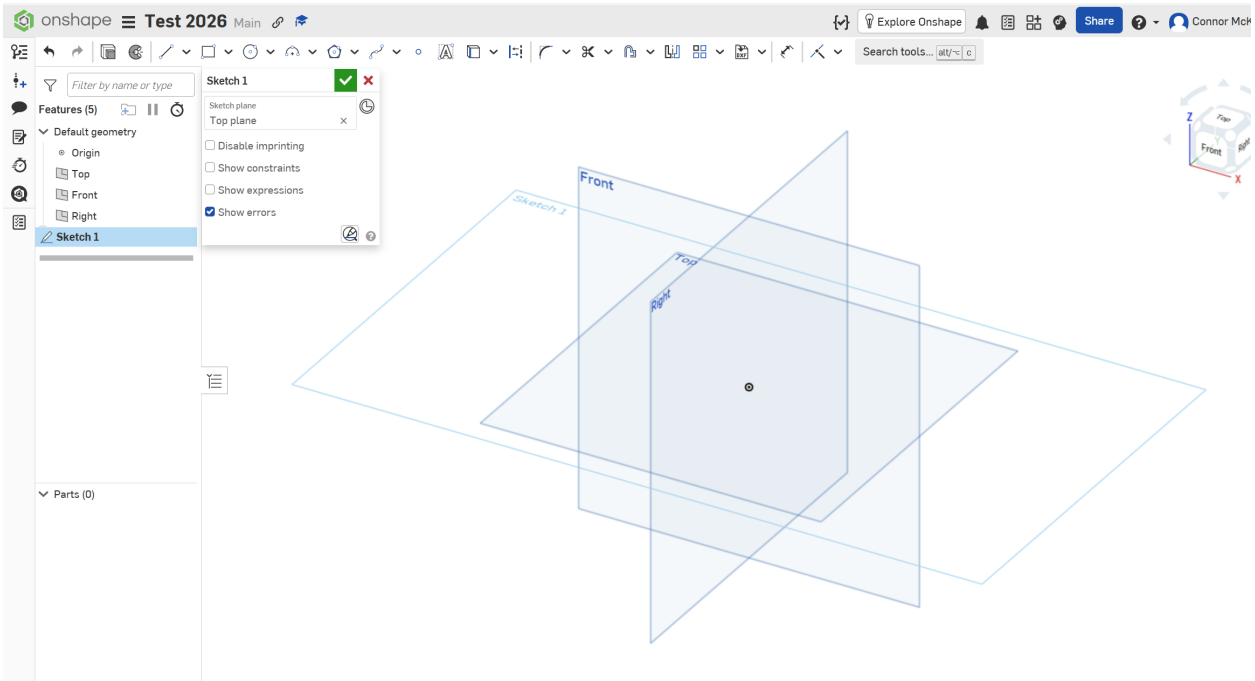
Basic CAD workflow

Generally, when using a CAD software for engineering purposes (some art-based softwares are different, but don't worry about them), most workflows revolve around the idea of **2D Sketch -> Make it 3D -> New 2D Sketch on Part of the 3D object -> Repeat**

Onshape has a variety of tools to accomplish this (and make it fancier later on), all of which can be found here:

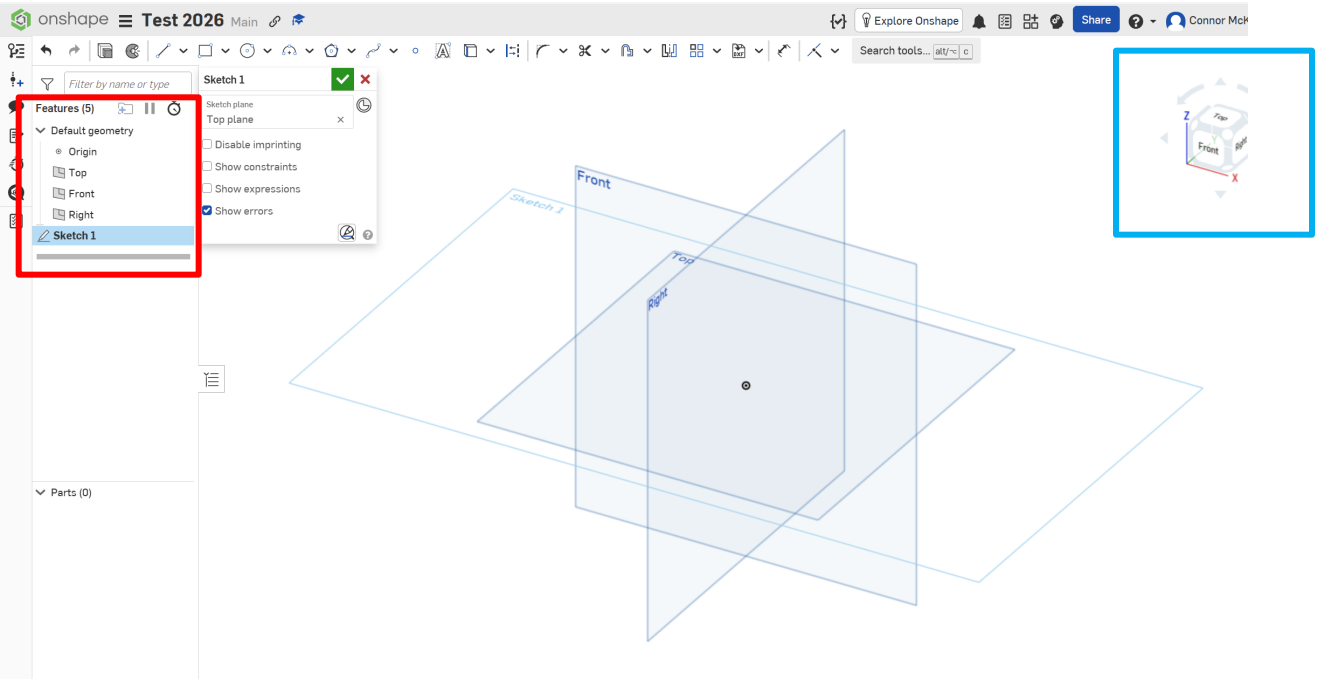


Of these tools, the first one you'll use in almost any situation is **Sketch**. This is how CAD softwares create 2D images, which you'll later make 3D. Click **Sketch, then pick one of the three planes** in the middle of the screen.



This is what it looks like once you have created a new sketch. You essentially have made a flat, 2D plane that you can draw shapes on with the new **Sketch Tool Bar** that replaced the one above.

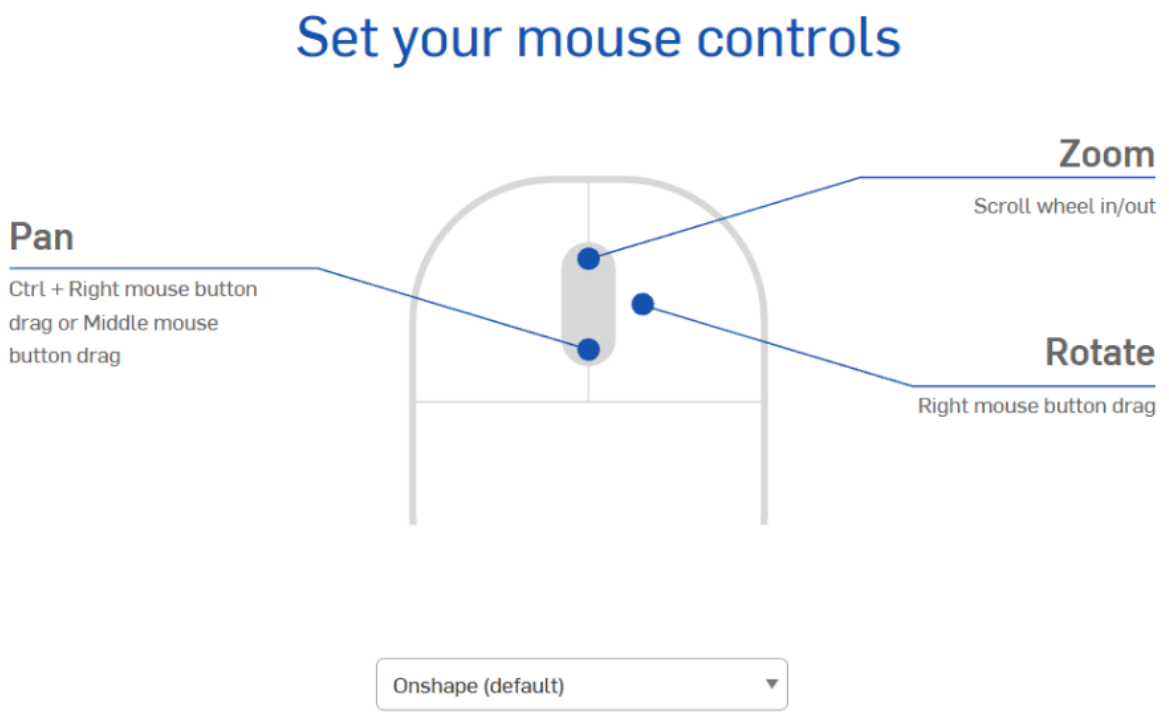
Next, some more features of the screen that will be useful.

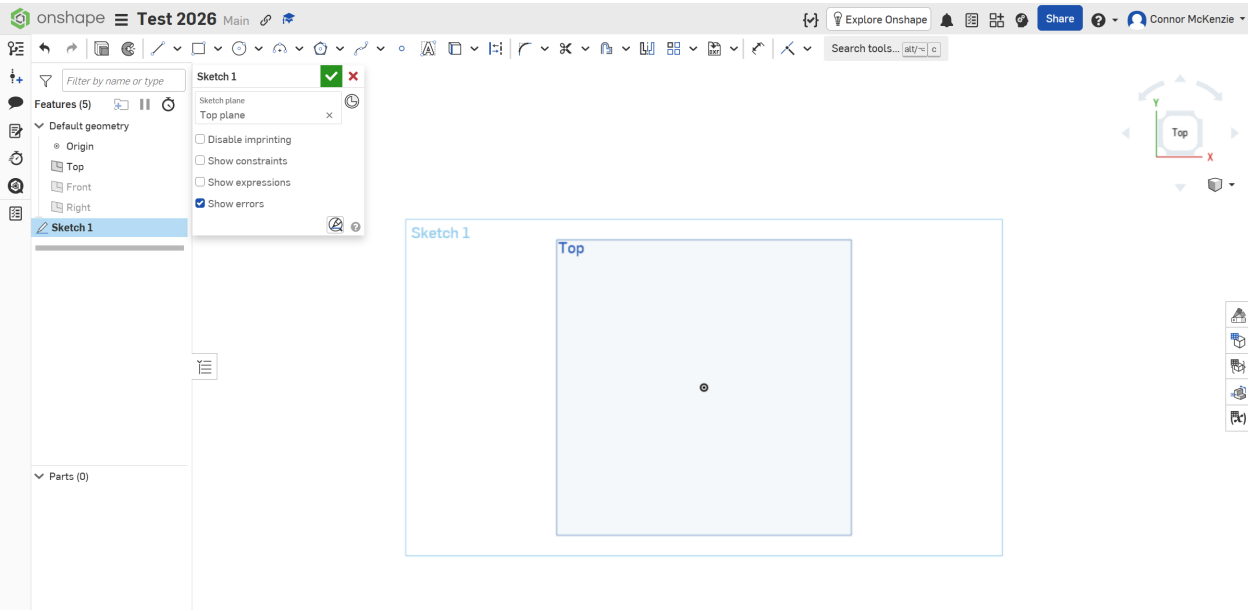


The RED section of the screen is called the **Feature Tree**, and shows all the stuff you've made so far. Right now, there should just be the three planes (which were there to start with) and the sketch you're working in. **Hover over the other two planes, and deselect the eyeball to hide them.**

The BLUE section of the screen shows the direction you're currently viewing the model from. You can use it to rotate the model if you don't have a mouse, and clicking on one of the faces will take you to a top-down (perpendicular) view of that plane. **Click on the face your sketch is currently on (Top, in my case).**

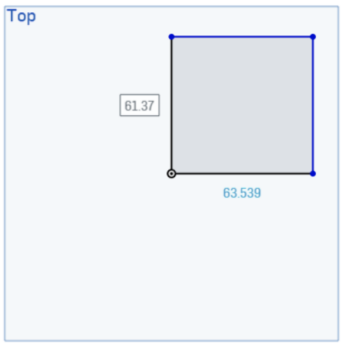
If you do have a mouse, you can control the screen with the following:





Now that you have a nice clean view looking straight down at your surface, you get to start drawing!

All of the tools in the tool bar serve different purposes, so feel free to play around with the ones visible to get a sense of what they do. Afterwards, make a **rectangle** of any size and shape.



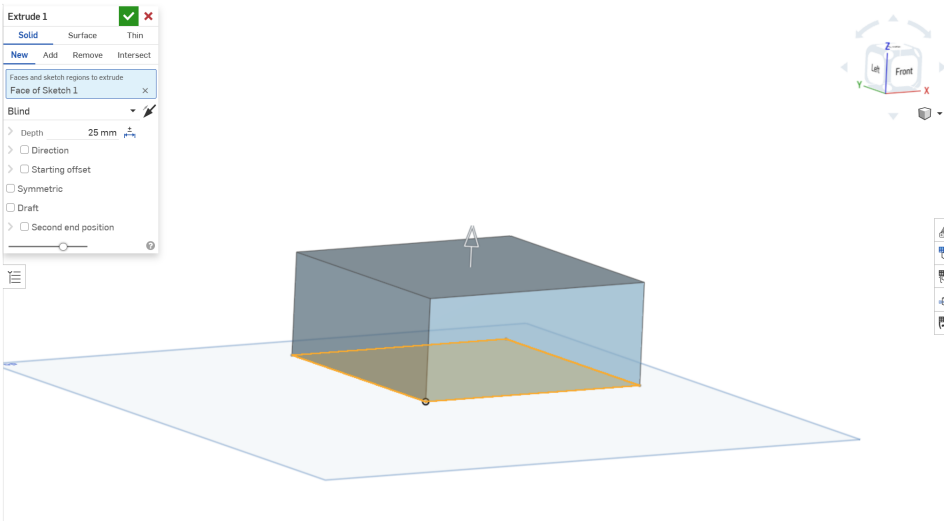
When you first create a shape of any sort, it will be *undefined*. This just means you haven't set numerical dimensions for it yet, so the computer thinks it could be any size. Hit the **Dimension** button in the toolbar, which looks like this: |<-->|. You can also hit the **d** key as a shortcut. This will allow you to set a fixed value for the sides by clicking on them and entering a value.

Moving around in the view: CTRL+Right-Click or Middle Mouse Button allow you to pan (move side to side). Right Click by itself rotates the object, and Scrolling will zoom in or out.

Once you're happy with your shape, click the Green Check Mark next to Sketch 1, which will confirm the sketch and **exit** back to the 3D view.

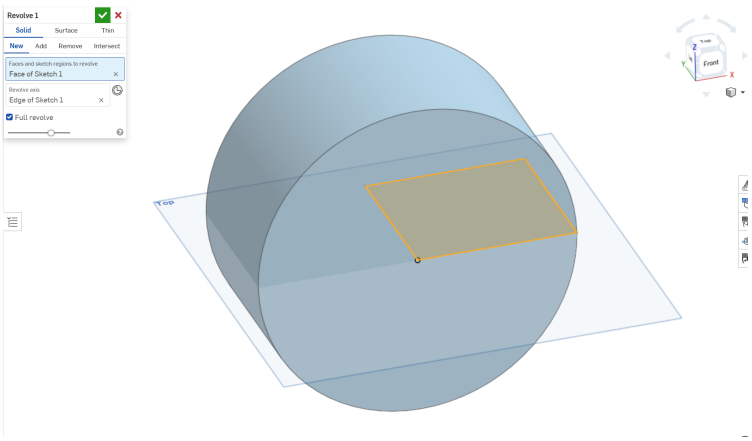
Now, we can use those 3D tools that were initially in the view. The two most basic ways to make a shape 3D are **Extruding** and **Revolving**. These are the leftmost tools, right next to the **Sketch** button at the top.

Extrude:



Takes the 2D shape and extends it straight up or down by a certain amount. Select **Extrude**, then click on the rectangle you made earlier.

Revolve:

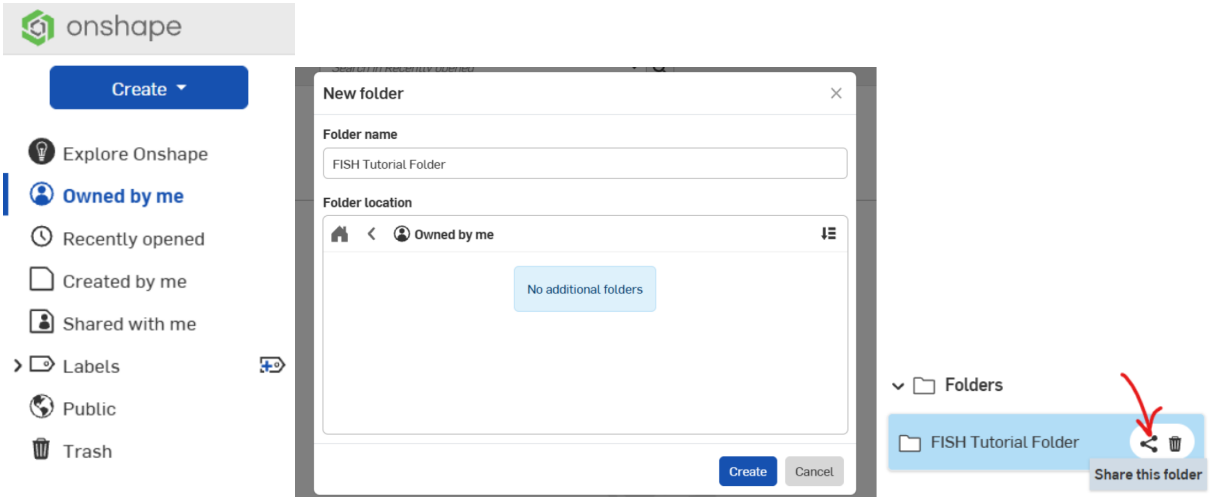


Rotates the object around a line of your choosing to make a 3D object. In this case, I set **axis of revolution** to be the left edge of the rectangle by selecting the box in the left Revolve Menu, and then selecting the line. I then selected **faces and sketch regions to revolve,** and selected the whole rectangular face.

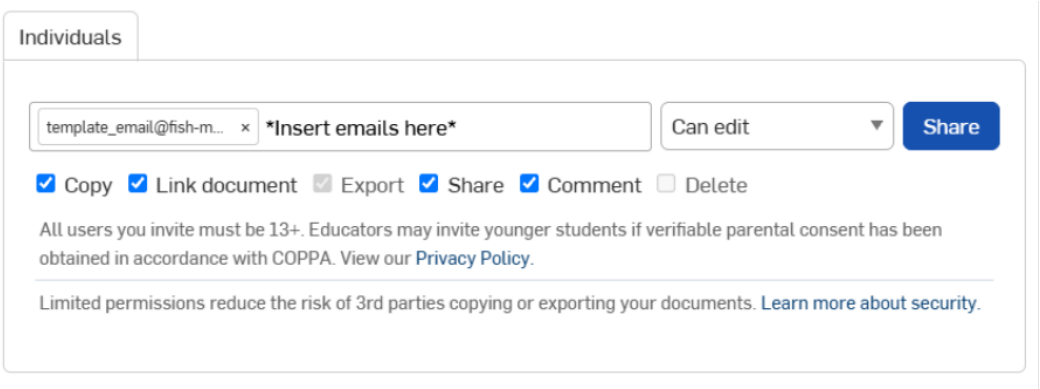
Sharing with others

Onshape also has the benefit of being able to share folders with others (like your Make-a-thon Teammates!)

To start with, make a folder on your home screen:



First, select the **Create** button again, but this time make a folder and name it something descriptive (like your Make-a-thon team name and number). Next, select the **Share this folder** button next to the name:



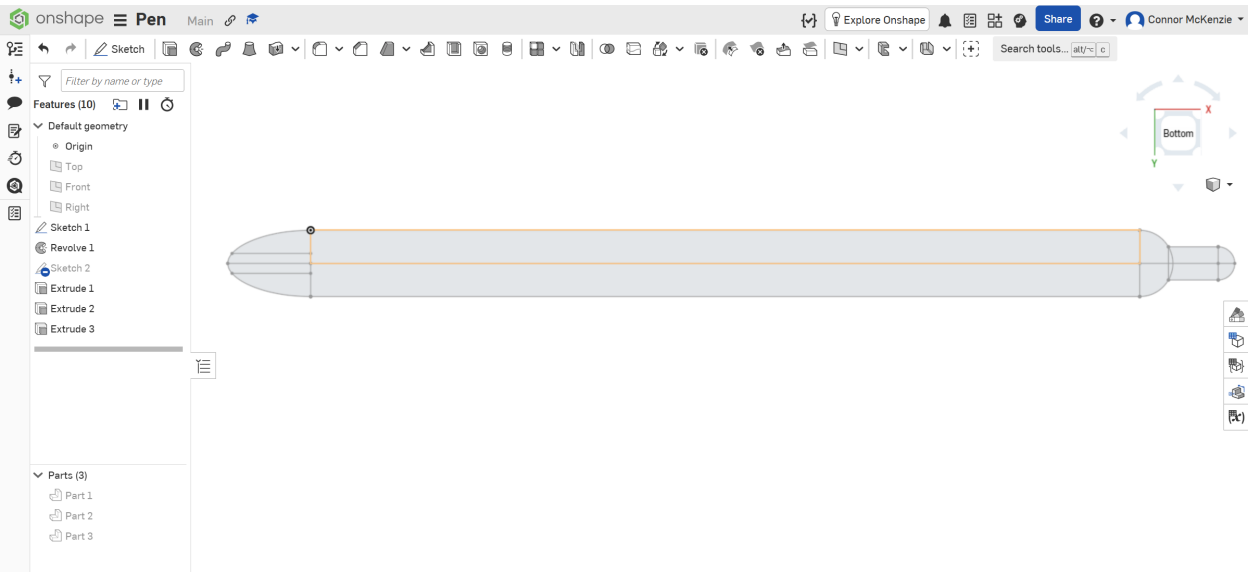
It will default to making a **Copy** as seen above, but you will want to **DESELECT** this option so that you are all editing the same group. Make sure everyone has *Can edit* selected, then share the link. That's it!

Put it all together

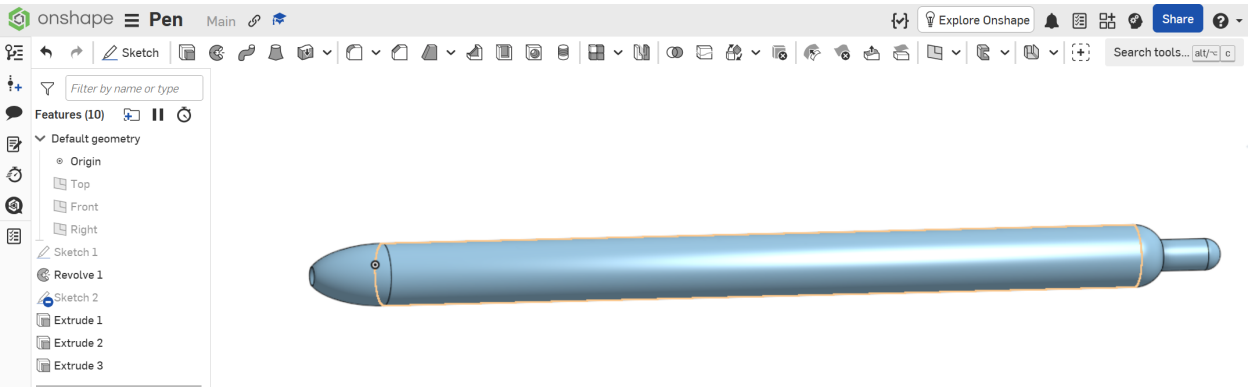
Now that you've learned the basics, try to make an object around you! I suggest starting with something simple, either blocky and rectangular (good for extruding) or round and symmetric (good

for revolving). Feel free to play around with all of the other 3D features too, including fillets (rounds over sharp edges), Holes, and Shells.

For inspiration, here's a simple pen I made with revolves and extrudes:

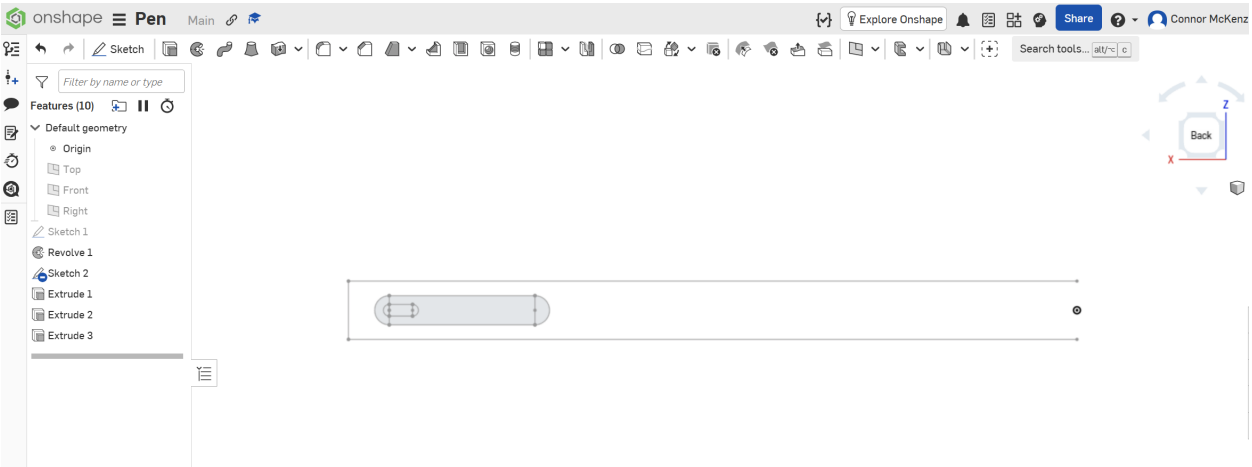


First, I drew out the shape of a pen using some rectangles, 3-point arcs, and a spline (curvy line that can be more annoying to use but lets you make shapes that aren't perfect circles). Make sure you include a centerline (the horizontal line through the whole shape above) if you want to revolve!



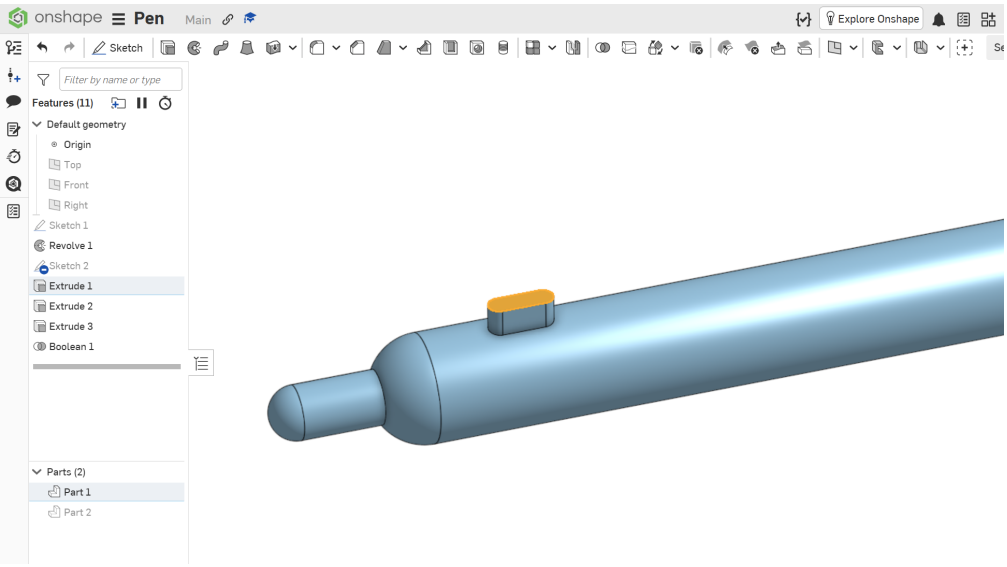
This is how it will look right after revolving. Almost there already!

Next, for the clip, I made a new sketch on one of the planes, and drew out the shape of the clip:

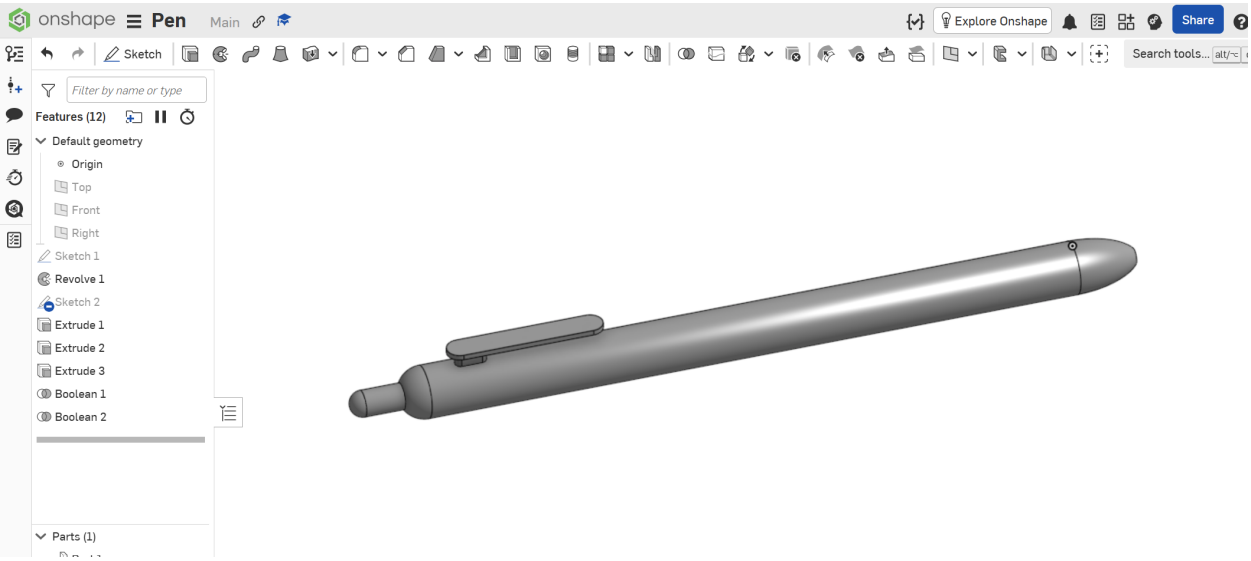


Hint: if you click the **Use** button within a sketch, you can select lines and shapes from parts you already have (in this case, the body of the pen) and it will automatically add those shapes to your sketch for you to reference. This is where the big rectangle around the clip came from in the sketch above!

Next, I selected **Extrude,** and selected just the small connector piece of the clip from the sketch above to extrude. I set the distance to 2mm, and clicked symmetrical so it would extend 2mm in both directions (this is just so there isn't an awkward gap between the body of the pen and the connector for my model, but you might not need to do this depending on how you created your object).



Finally, I hovered over the sketch to *show* it again, since it will automatically hide a sketch once you have used it once. I then selected **extrude** one more time, and this time under **starting offset** I clicked Entity and chose the top face of the connector I had just made. This face is colored orange in the image above after I clicked on it. Starting offset just means that instead of starting the extrusion from where your sketch is, it will start from the face you chose. I can then enter 1mm and accept the extrusion!



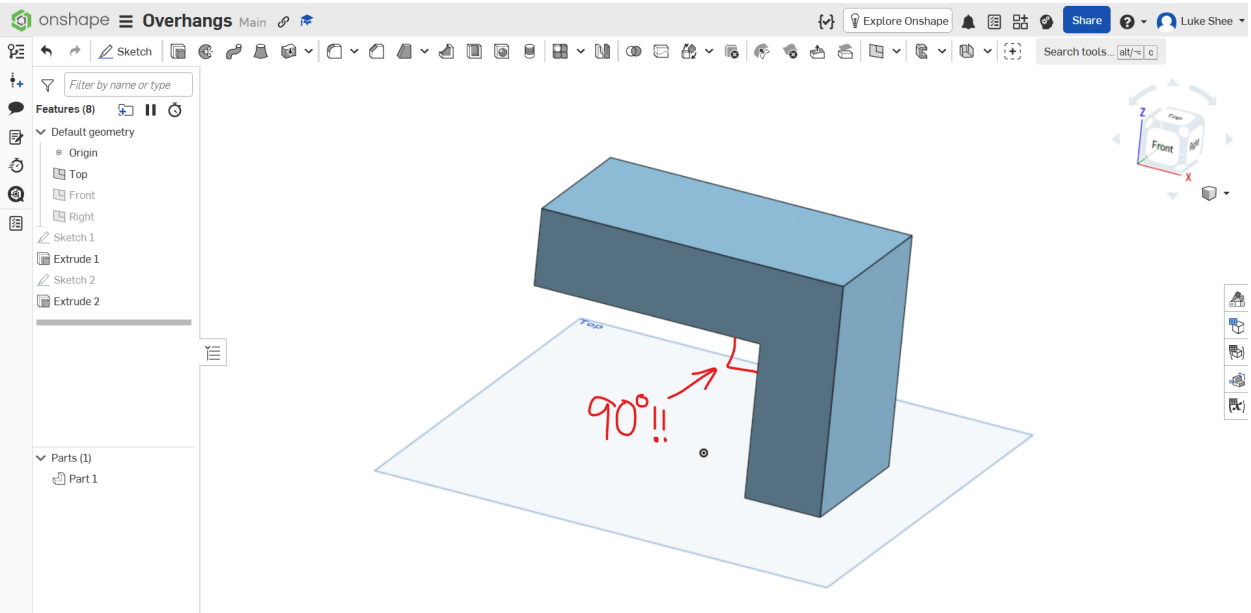
You'll notice that by doing the offset extrusion above, you get the gap between the clip and the pen where you would be able to attach it to a paper or shirt in real life. You should now have a finished pen design!

Best CAD-ing Practices

Supports and Limiting Overhangs

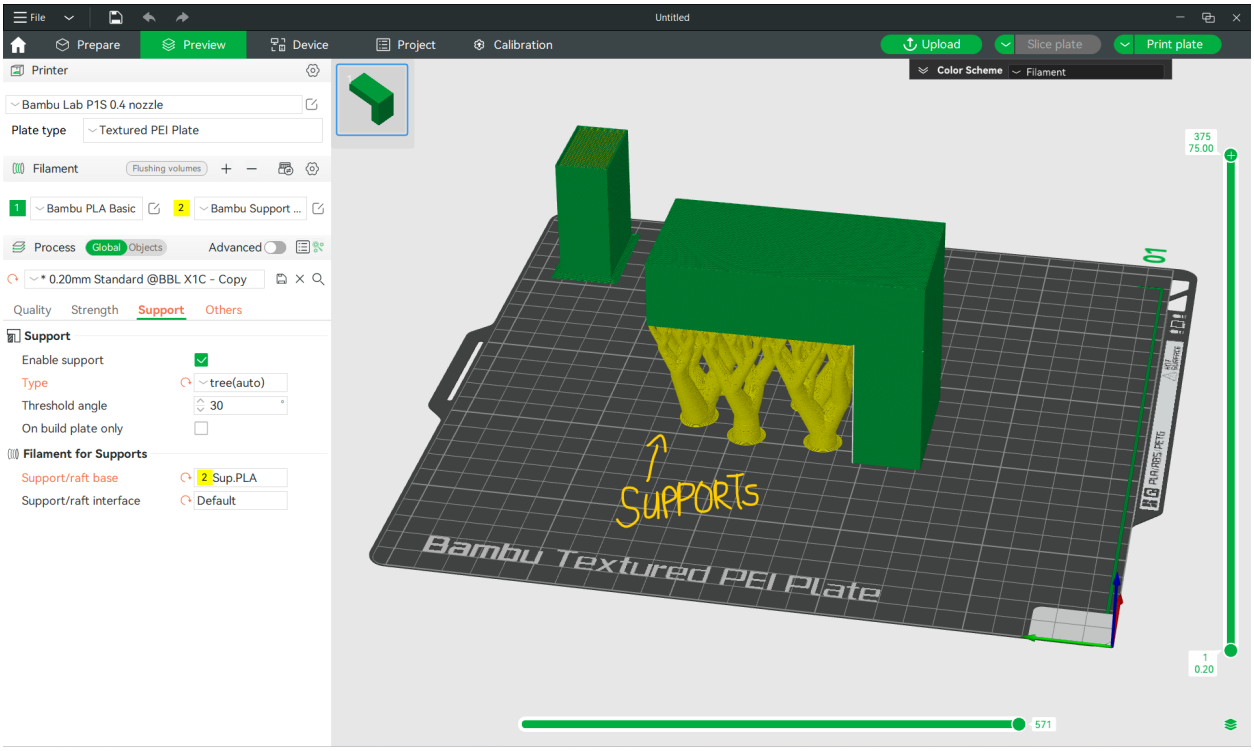
Most standard 3D printers build models from bottom to top. When the printer maps out and creates the very bottom layer of a 3D model, it continues to build upwards layer by layer from that initial foundation (similar to how someone would build a house or tower).

But what if your 3D model has an ***overhang***? Take for example the picture below, where there's an L-shaped block that's currently oriented to have a 90° overhang. While it's quite easy to make this shape in CAD software, printing it out in layers top-to-bottom (in this orientation) wouldn't be feasible since there's no supporting material below the left side of the overhang that would keep it in the air.



To solve this problem, 3D printing program like Bambu Studio provides the option to automatically add ***supports*** to an imported CAD file that are printed out with your model to give any overhang... well, *support*!

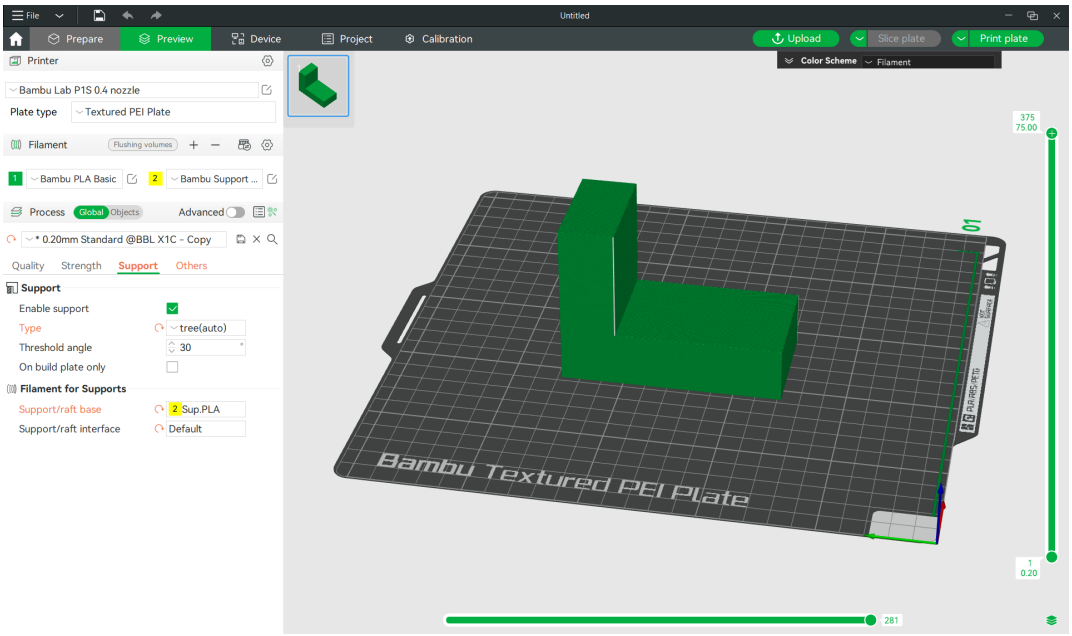
In the image below, I uploaded the STL file of the model above to Bambu studio and enabled automatic "tree" supports. The resulting preview of the printed CAD model is shown, with the supports indicated in **yellow**.



When printed, the model will have these tree supports attached to it, which can then be broken off manually to ascertain the original part design. However, breaking off these supports can be hassle if made within long crevices or tubes, and they can possibly leave detachment marks on the model that make it less smooth.

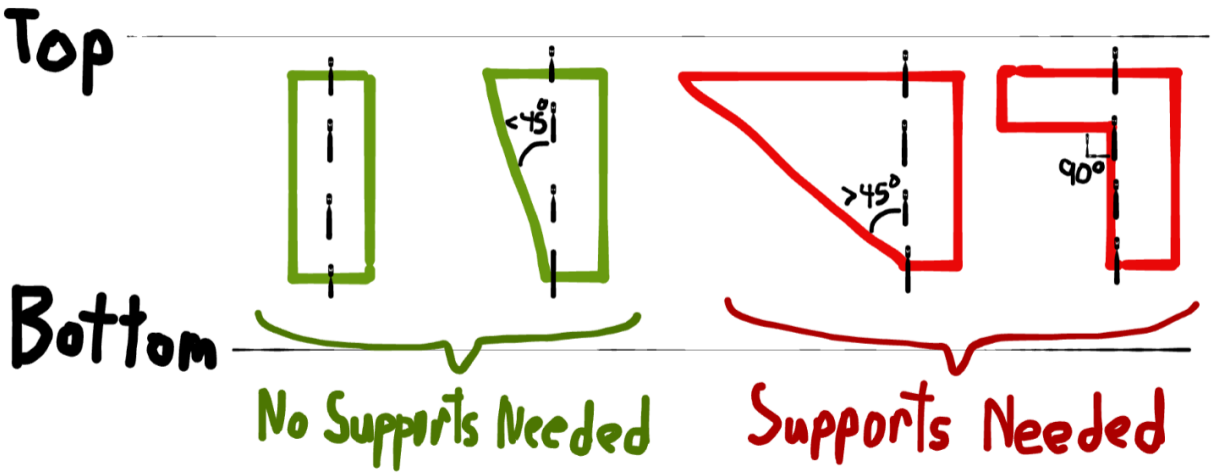
Therefore, it's good practice to design your CAD model to require the least number of supports possible. You can do this by limiting the overall amount of overhang present in your model.

Alternatively, you can also try printing your model in a *different orientation* that minimizes the most amount of overhang. Going back to L-shaped model, what if I oriented it upside-down or on its side through Bambu Studio's rotate feature? Printing in those orientations won't require any supports at all!



As you're CAD-ing a 3D part, practice asking yourself which side or orientation of your CAD model would leave the least amount of overhang and thus be the best side/orientation to print on. The bottom side of your print should ideally be flat to provide a wide-ranging foundation layer to build the rest of the model on.

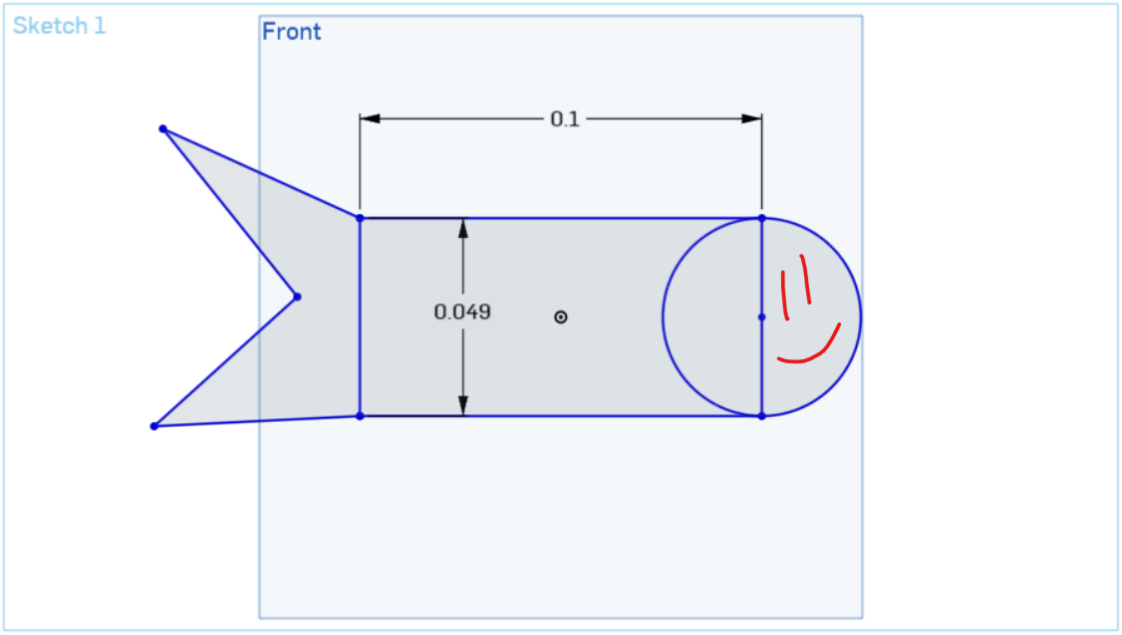
As a good rule of thumb, you should know when an overhang needs supports or a better printing orientation if the hanging angle from the part's vertical axis is greater than 45 degrees, as illustrated below.



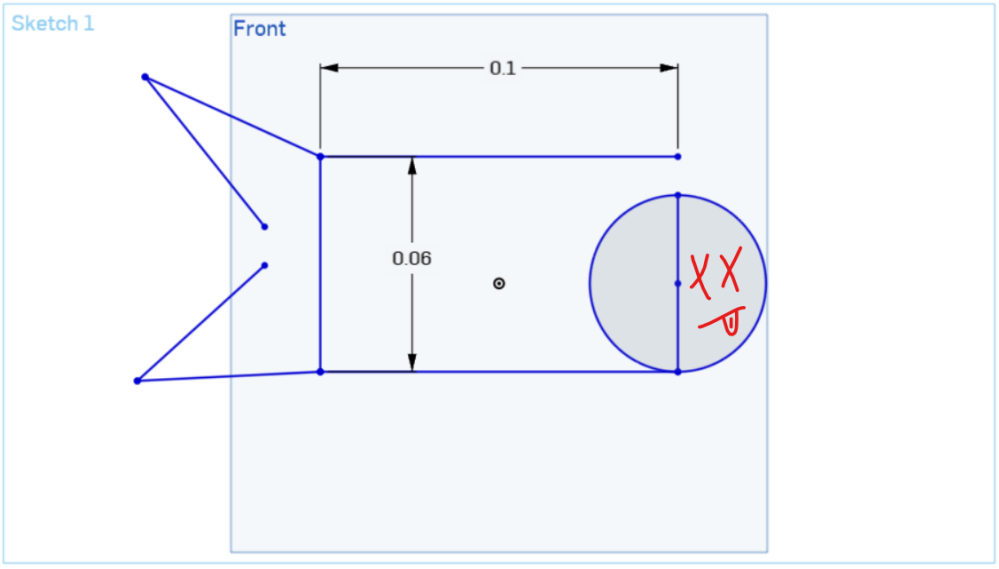
Constraints and Design Intent

Design intent refers to the rules of how a developed CAD sketch or model responds to having one of its dimensions changed, and it's implemented through **constraints**.

To help you better understand design intent, check out this cool fish I sketched out below! (don't laugh...)



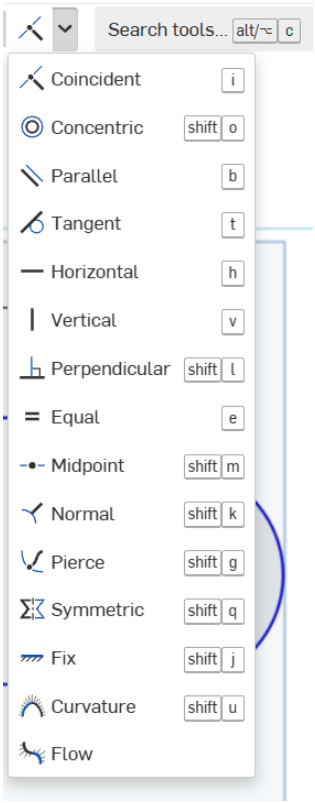
Nothing seems too out of the ordinary with this sketch at first glance, other than all the lines being blue. However, say I wanted to make the fish thicker. I would just need to adjust the height dimension of the rectangle body from 0.049 m to 0.6 m, and the fish would then perfectly thicken...



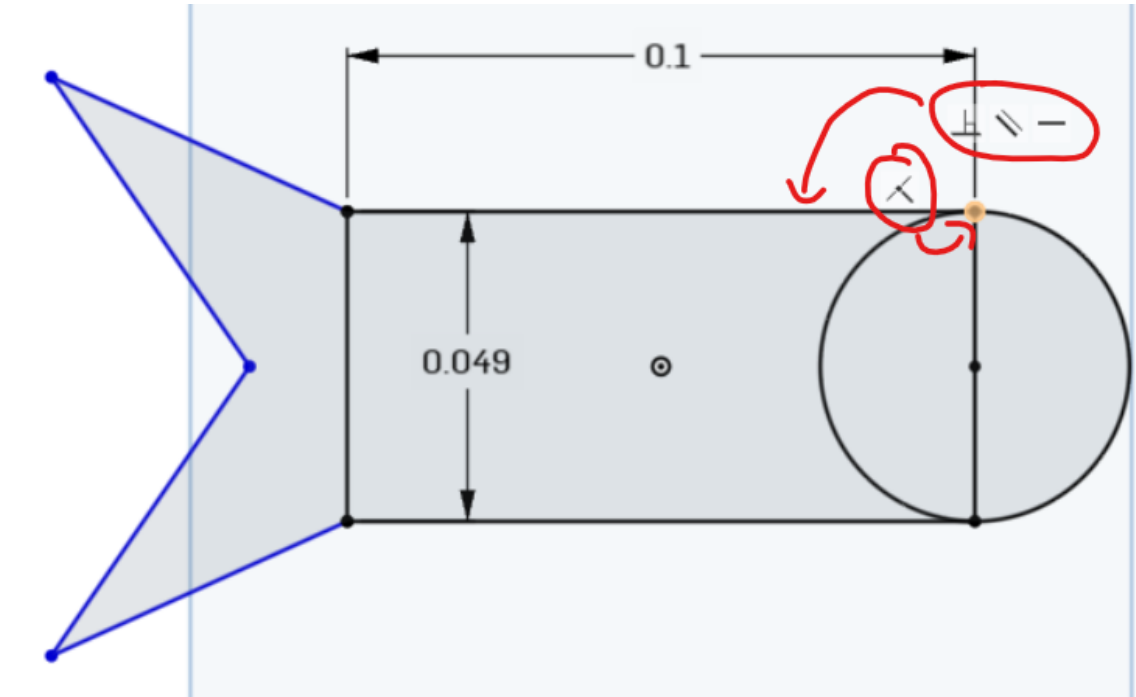
Ah...

In truth, I did one crucial thing to the sketch: I deleted all the pre-generated constraints!

Constraints are essentially the "rules" you enforce on a sketched CAD model that define specific relationships between different parts of the sketch. You can add constraints through the dropdown button in the top bar, between the Dimensions icon and Search bar.



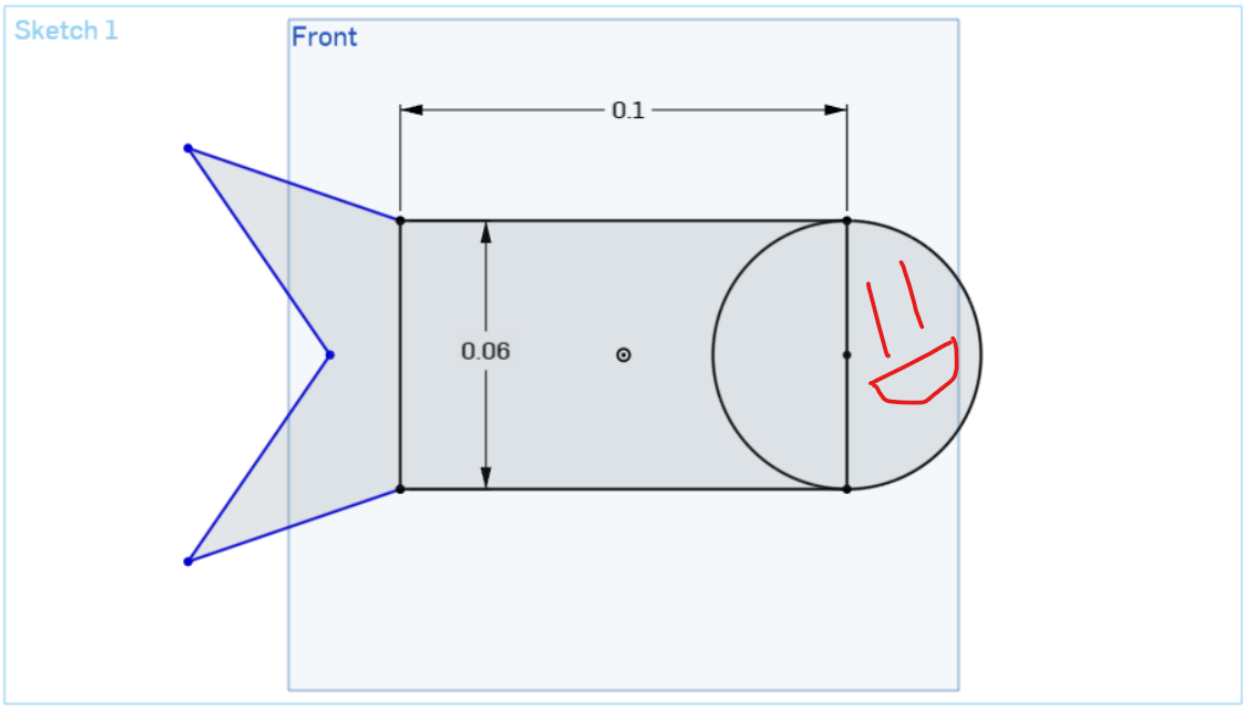
As you can see, there are plenty of constraints you can experiment with, and we encourage you to experiment with them on your own. But for now, let's revert the fish sketch back to when it had constraints:



When you hover over certain points and lines of the sketch, little icons appear that show which constraints have been applied to them. For example, the top line of the rectangle body has three constraints: a Perpendicular  constraint to ensure 90-degree angles with its sides, a Parallel

 constraint that makes the line parallel to the bottom line of the rectangle, and a Horizontal  constraint to guarantee that the two ends of line will stay at the same horizontal level with each other even if something like the circle head increases in radius.

On the intersection point between the line end and the circle head is the Coincident  constraint, making sure that point on the line and circle sticks together when dimensions change, sort of like a glue. With that constraint, we can now increase the thickness of the fish so that the head correspondingly increases in size, thus preserving the design intent for the fish head to be flush with the rectangular body in the sketch.



(Tip: Onshape automatically adds constraints to sketched shapes if you make them on specific points, like the midpoint or any other points on preexisting sketched lines. For example, the rectangle body of the sketch above was created on the midpoint of the sketch plane, so now the body has a Midpoint constraints that keeps the rectangle centered on the sketch plane even when I updated the length or height of it)