Making Your Fist Boat in Onshape

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Introduction

Ahoy there, and welcome to Cyclone RobSub! We are thrilled to have you with us!

As part of our training regimen, you and your team will be asked to design, model, assemble, wire, and program a small model boat. This document covers the modeling portion of your training. No prior experience in any CAD software is required to follow along.

In this document, you will be guided through the process of creating a sample boat that you are encouraged to build upon for your teams design. The primary goal of this document is to familiarize yourself with the fundamentals of CAD design in OnShape.

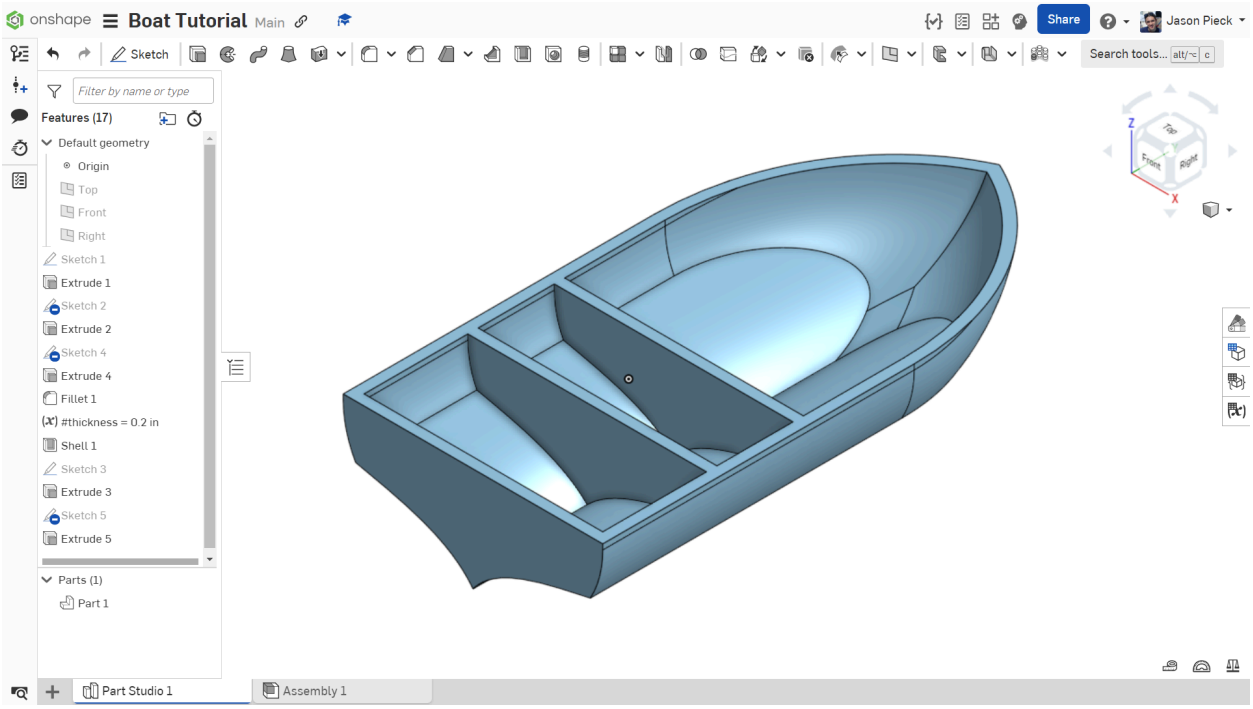

Figure 1: caption of the static figure

Roadmap

Warnings

Tools & Materials

Steps

Step 1 - Starting Your first Sketch

The first thing that we need to do whenever making a model in CAD is making a sketch. A sketch is a 2D stucture that exists on a plane. To make your first sketch, click the `Sketch` button in the tool bar as seen in Figure 2.

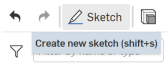

Figure 2

Onshape will then prompt you to select a plane. Select the top plane by either clicking on the plane in the view-window or on the feature tree as seen in Figure 3.

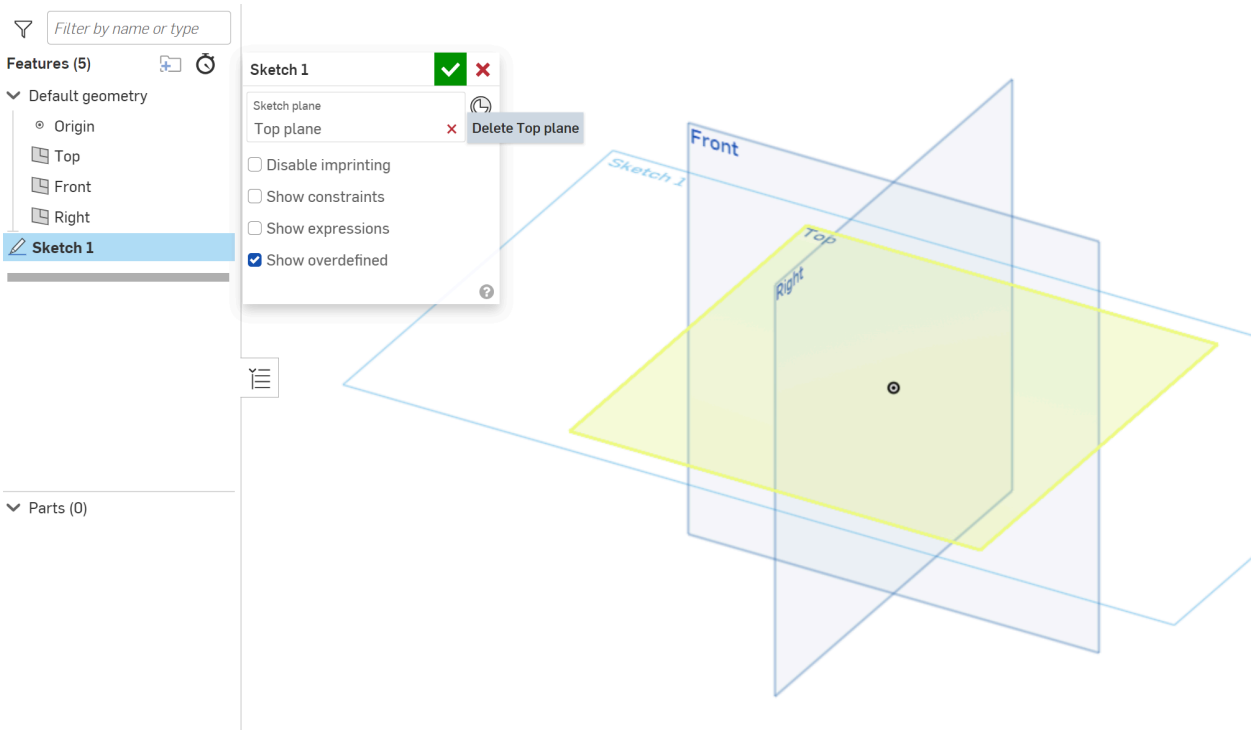

Figure 3

It is best practice to be looking directly at the plane you are working on, so navigate to the view-finder as seen in Figure 4 and click on the `Top` surface. This will reorient your view-window such that you now face the top plane directly head on.

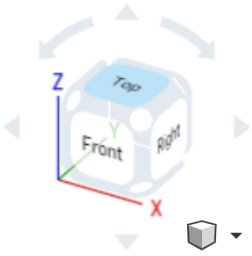

Figure 4

Step 2 - Sketching base of the boat contour

Select the `Center point rectangle` tool from the rectangle tools drop-down as seen in Figure 5. After selecting, your tool should change in a cross-hair shaped like a plus sign.

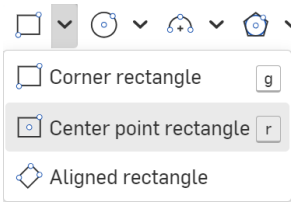

Figure 5

Click on the origin point and then move your mouse outwards as seen in Figure 6. At this point in time, it does not matter how far out you go, nor that the numbers that you see in Figure 6 match your own.

Warning

Failing to select the origin point when starting the sketch will result an under-defined sketch that does not align with the default geometry of the model. This can cause issues later down the line in more complex models, and should be avoided whenever possible.

Now that we have shape, we can add dimensions. Select the `Dimension` tool as seen in Figure 8. Your mouse should once again change to a cross-hair. Click on the left side of the rectangle, and then click off to the side as seen in Figure 7.

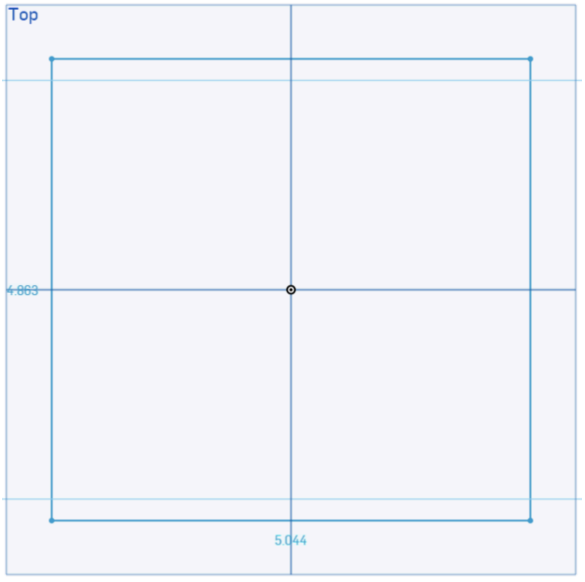

Figure 6

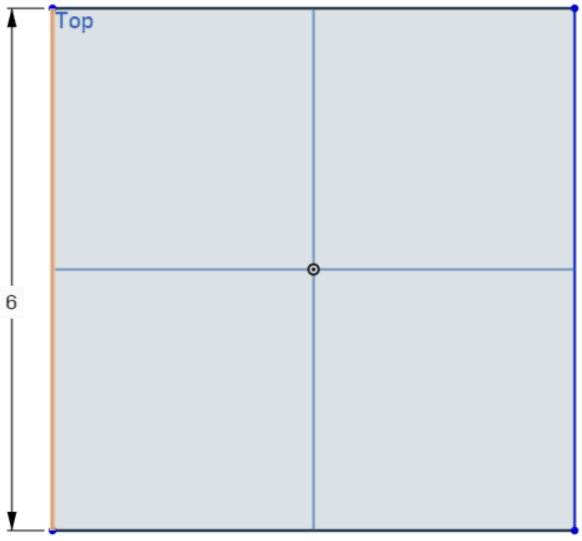

Figure 7

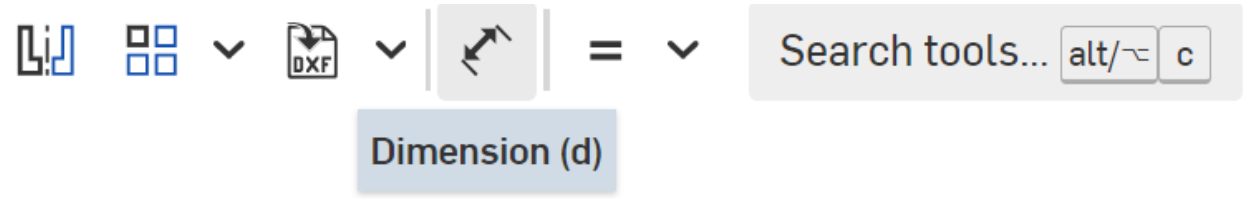

Figure 8

We want this rectangle to be a square, and while we could add the same dimension to the bottom or top side of the rectangle, we can add a "constraint" that forces those sides to be equal. We can do this by first holding `Shift` on your keyboard, and then selecting the two sides that you want to be equal. Then go to the constraints menu, and select `Equal` as show in Figure 9. Now if we change the dimension set earlier, all sides of the square will change size automatically.

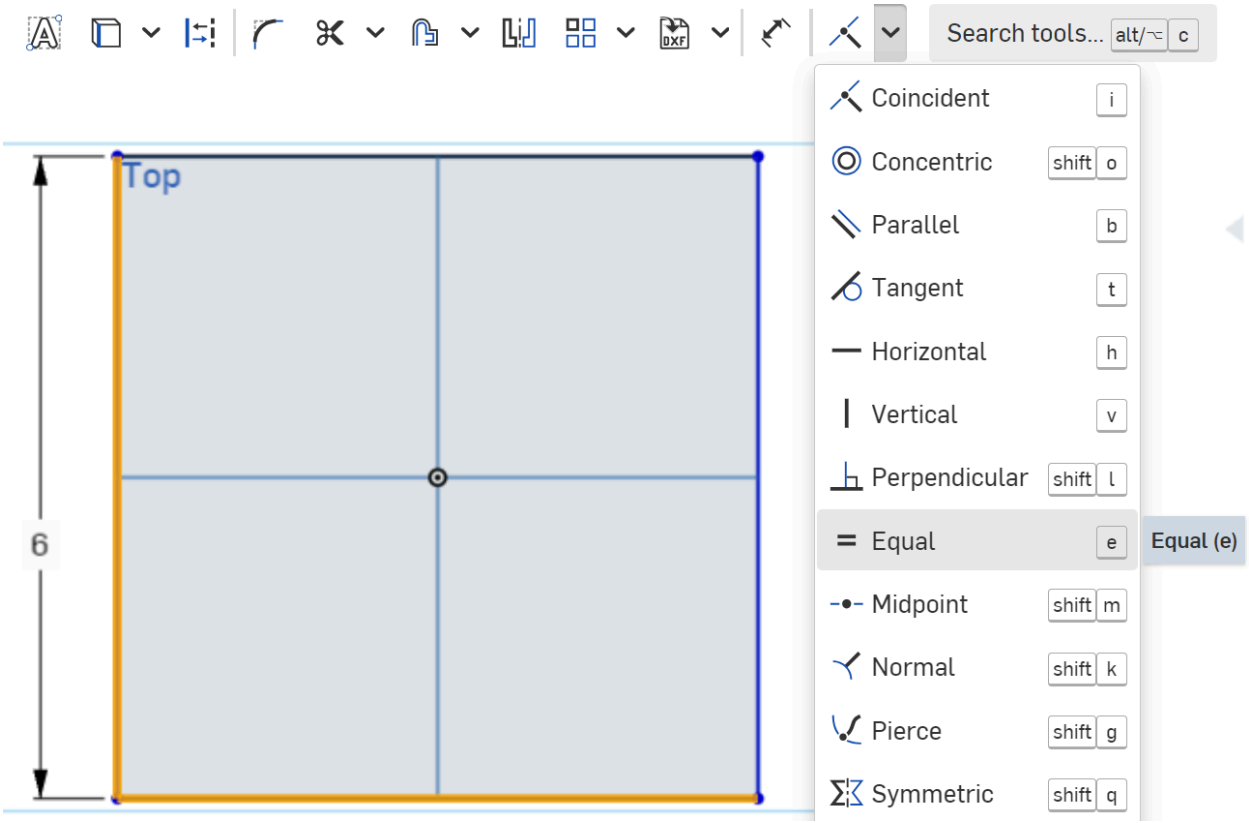

Figure 9

Step 3 - Sketching the front of the boat contour

Using the `Circle` tool, add two circle to your the top corners of your sketch as seen in Figure 10. When selecting the size of the circle, line up your cursor with the opposite opposite sided corner. As you get close, the line will change colors to orange and the icon for `Tangent` will appear. The icon signifies that Onshape is automatically adding a relation to the sketch. If done correctly, the circle will appear black in color.

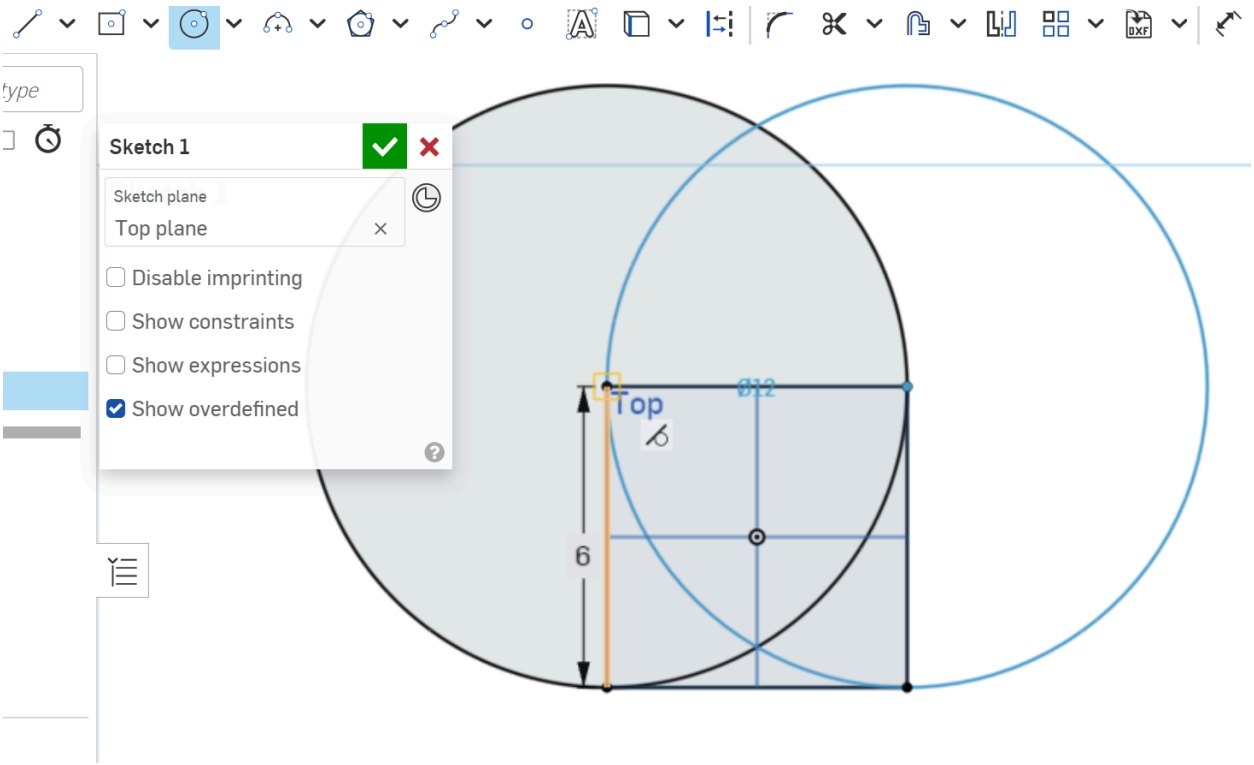

Figure 10

Select the cutting tool, and click and drag your mouse of the sections of the circle as signified in Figure 11. Once you are finished, your sketch should look like Figure 12. Select the green check mark, and your sketch is finished!

Note
The trim tool is a great way to remove unwanted geometry from a sketch.

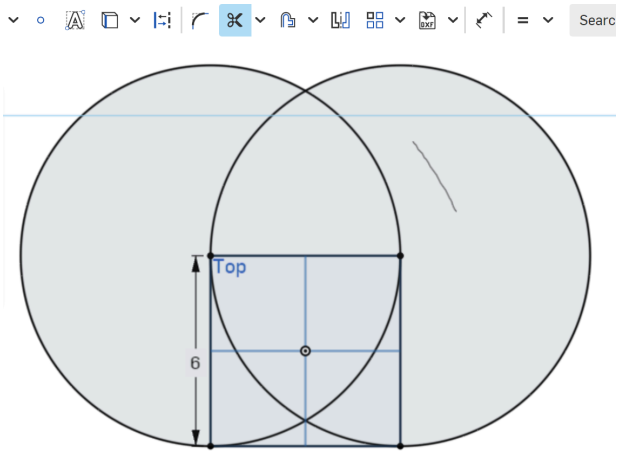

Figure 11

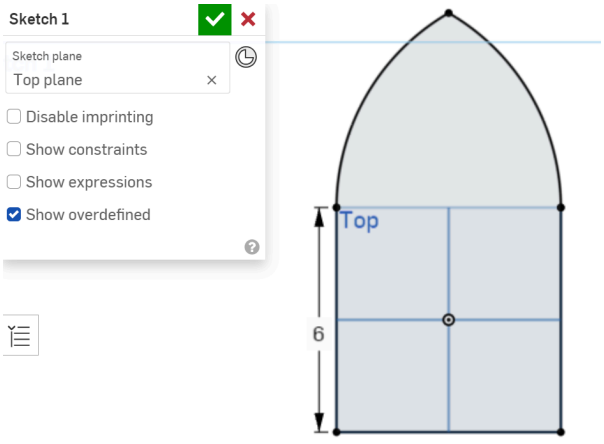

Figure 12

Conclusion