

How to Make a Model Boat in Onshape

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Introduction

Ahoy there, and welcome to Cyclone RobSub!

As part of the team's training regimen, you and your team will be asked to design, model, assemble, wire, and program a small model boat. This document aims to familiarize you with the fundamentals of computer aided design (CAD) in Onshape. No prior experience in any CAD software is required to follow along. By the end of the tutorial, you will have created a model boat like the one seen in [Figure 1](#).

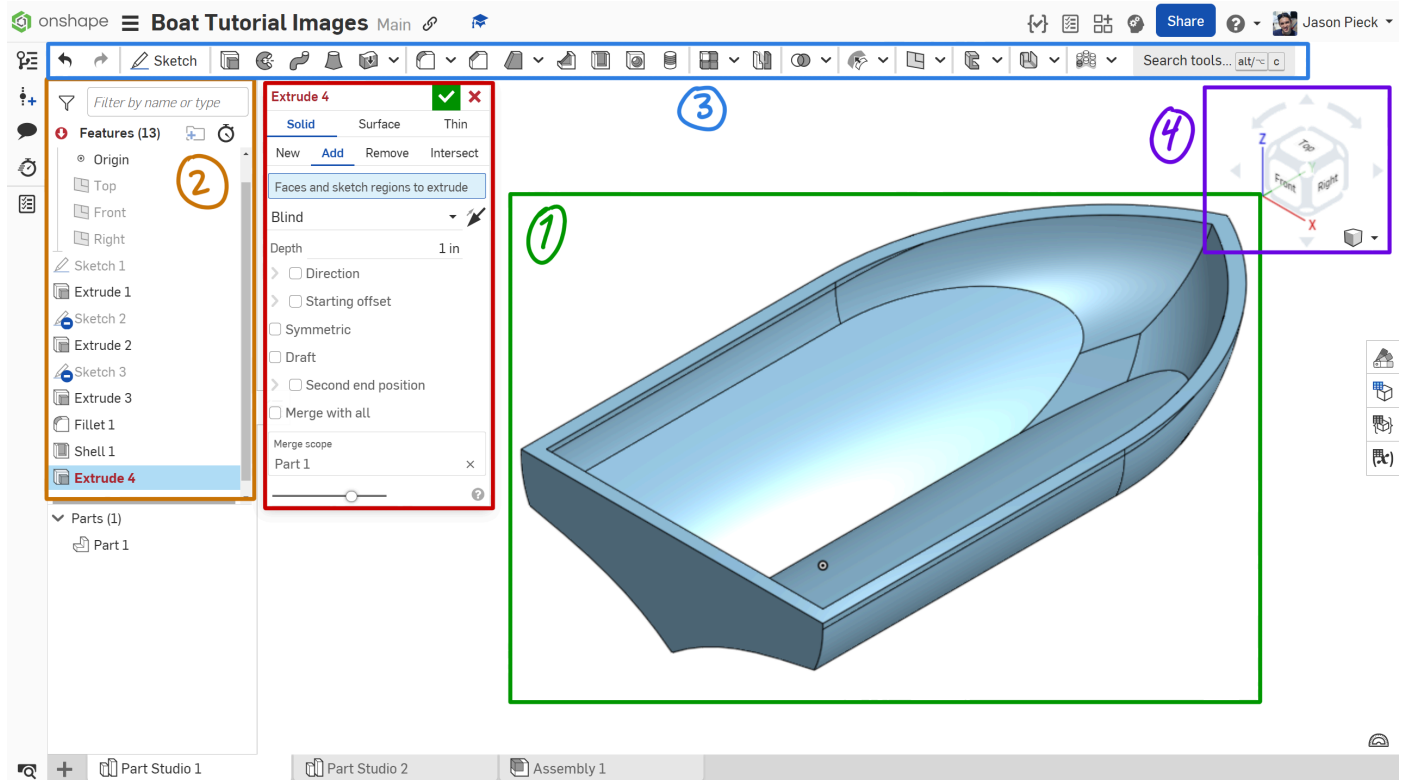


Figure 1

Overview of Onshape's User Interface

Looking closer at [Figure 1](#), Box 1 contains the the "Graphics Area" in which you view and edit your model. Select a part of the model to see a corresponding "Feature" highlighted in the "Feature List" in Box 2. The "Feature List" keeps a history of every operation performed, allowing you to go back and make changes to earlier states of the model. To add to features to the "Feature List," select actions on the "Toolbar" in Box 3. Selecting a tool will open a "Dialog Window" as seen in Box 4 and houses the parameters to modify the current operation. To see how tools affect your design, click on the "View Finder" in Box 5 to change your view angle.

Tools & Materials

As mentioned before, this tutorial utilizes the CAD software "Onshape". Before beginning this tutorial, create an account at onshape.com/en/education. Using this link gives you access to the free education version of Onshape.

Note

You can follow this tutorial in a different CAD Software. While the user interfaces differ, this tutorial utilizes conventional operations that are common to all CAD software.

Roadmap


We will start by making the basic shape of a boat, and then slowly carve away at it to refine its geometry that so that it can be propelled cleanly through the water.

Warning

CAD's power manifests in the wide verity of ways that modeling challenges can be approached. Consequently, this tutorial cannot be fully comprehensive and will only focus on one method to achieve the desired shape. If you are interested in alternatives, I highly recommend you check out the wider verity of resources available online.

Procedure

1 - Starting Your first Sketch

To make your first sketch, click the  Sketch button on the left side of the tool bar. 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 2](#). To view the plane head on, select Top on the "View-Finder" also seen in [Figure 2](#).

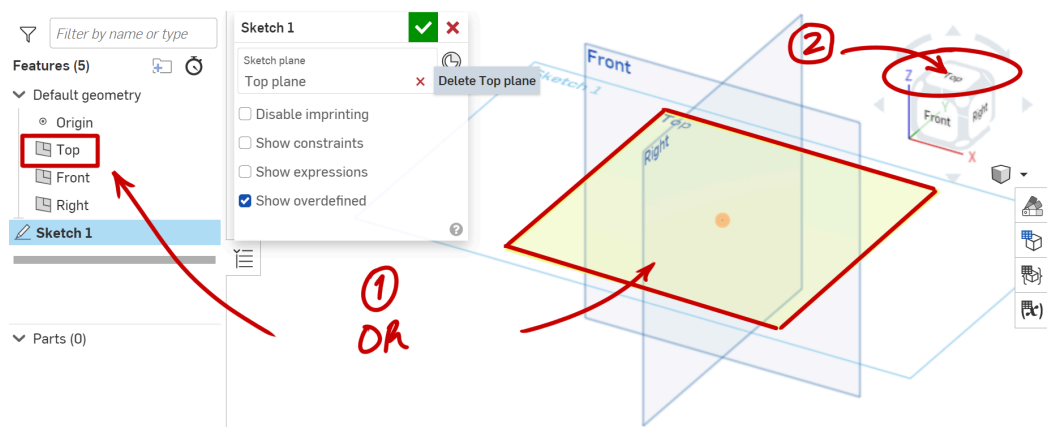



Figure 2

2 - Sketching base of the boat contour

Select the  Center point rectangle tool from the rectangle tools dropdown as seen in [Figure 3](#). After selecting, your tool should change in a cross-hair shaped like a plus sign.

Click on the origin point and then move your mouse outwards as seen in [Figure 4](#). Right now, it does not matter how far out you go, nor that the numbers that you see in [Figure 4](#) match your own.

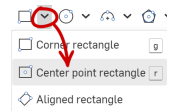


Figure 3

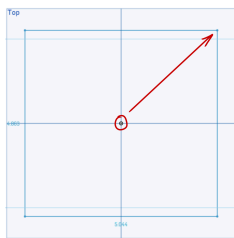



Figure 4

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.

Now that we have shape, we can add dimensions. Select the  Dimension tool from the right side of the toolbar. 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 5](#).

Instead of adding another dimension, create a relation called **= Equal** between two perpendicular sides of the rectangle. Select the two sides and then select **= Equal** as show in [Figure 6](#). Now if we change the dimension set earlier, all sides of the

square will change size automatically.

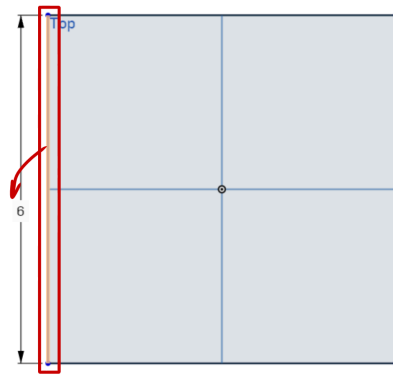


Figure 5

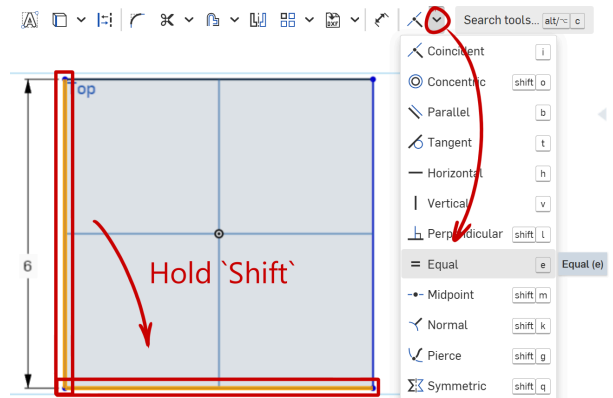


Figure 6

3 - Sketching the front of the boat contour

Using the Circle tool, add two circles to your the top corners of your sketch as seen in [Figure 7](#). 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.

Select the Trim tool, and click and drag your mouse of the sections of the circle as signified in [Figure 8](#). Once you are finished, your sketch should look like [Figure 9](#). Select the green check mark, and your sketch is finished!

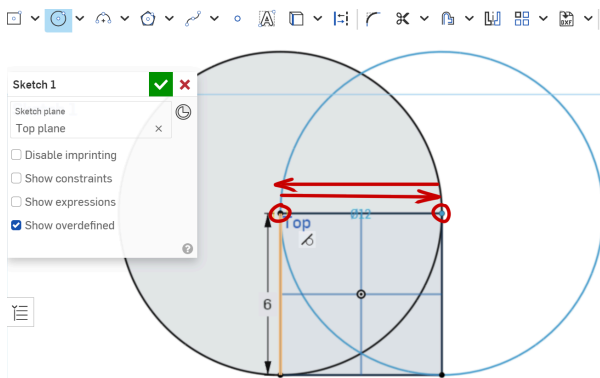


Figure 7

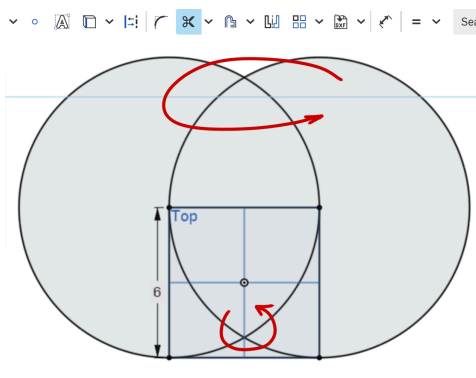


Figure 8

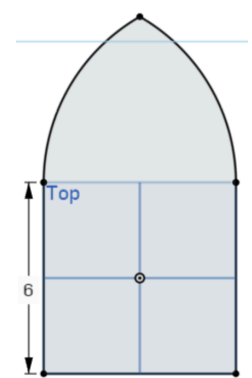


Figure 9

4 - Extruding the Sketch

Now that we have a sketch, we can “extrude” it to make it three-dimensional. Click the Extrude button in toolbar [Figure 10](#). and then select the face of sketch 1 as shown in . As in Step 1, you can either click on the face in the graphics area or on the feature list. Set the **Depth** of the extrude to **2 in** and click the check mark. If you reorient your part, it should now look like [Figure 11](#)

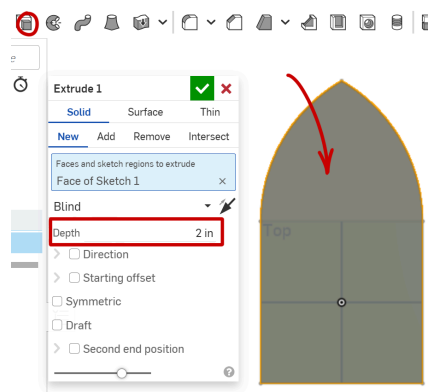


Figure 10

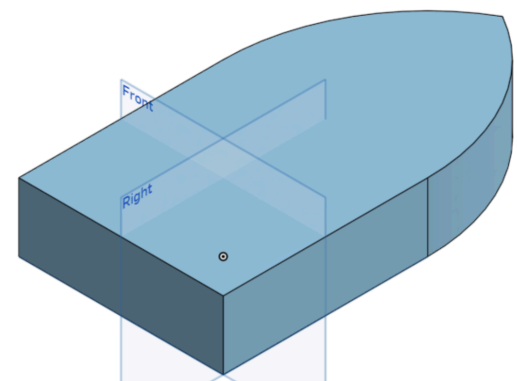



Figure 11

5 - Adding a sketch to non-planar geometry

Create another  **Sketch**, this time clicking on the flat surface side surface of the boat as seen in [Figure 12](#).

Note

Onshape will not allow you to built a sketch on a curved surface. As a result, we need to built a sketch that is offset from the surface

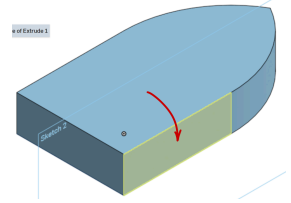





Figure 12

Looking at the tipped end of the boat, add a spline using the  **Spline** tool. Start at the bottom left of the curve-start, and then go to somewhere above the top of the boat as in [Figure 13](#). To cancel a third spline point, press **Esc** on your keyboard. Click on the end point of the spline and the tip of the boat, and then add a relation called  **Coincident**. This will force the two points to overlap.

Add another  **Coincident** relationship, this time between the spline control node and the closest edge of the boat as seen in [Figure 14](#). The nodes should be stuck to their respective lines. Drag each not to a position that gives you a shape to your liking similar to [Figure 15](#).

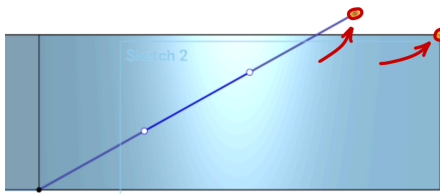


Figure 13

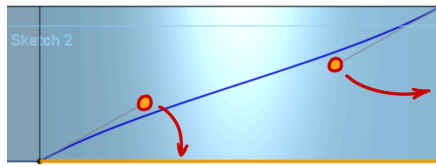


Figure 14

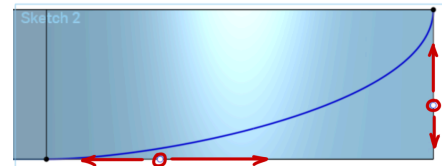



Figure 15

Complete the sketch by using the  **Line** tool to draw two lines and close the shape of the spline. If the sketch is successfully closed it will look like [Figure 16](#). Exit the sketch, and make another extrude. This time, select **Remove**, and change the end condition to **Through all**.

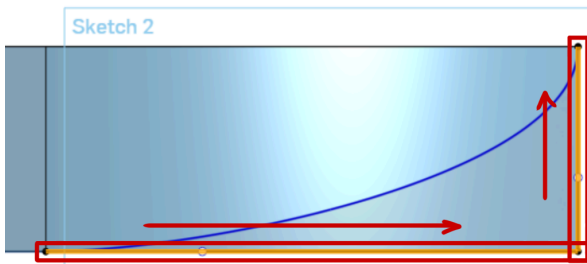


Figure 16

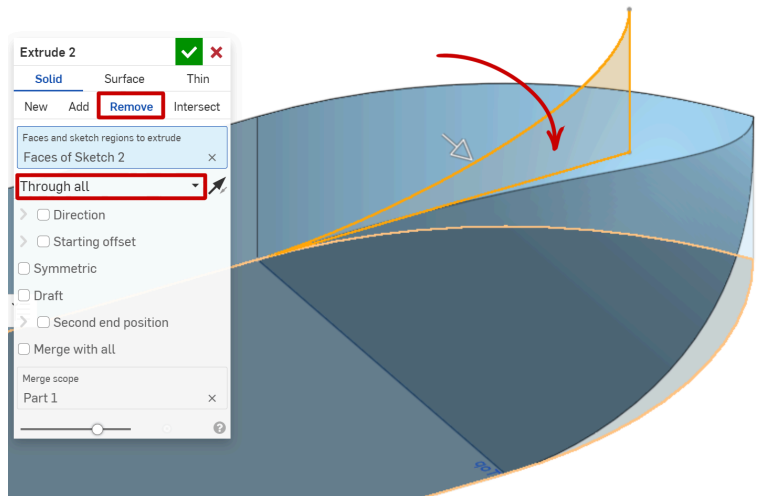






Figure 17

6 - Shaping the bottom of the hull

Create another sketch, this time on the flat back side of the boat. First create a center line using the  **Line** tool by aligning with the top and bottom midpoints of the hull. Make this line a construction line by clicking the  **Construction** tool.

Next, using the  **Spline** tool, create an arc same similar to [Figure 18](#). Here, the nodes do not need to be precisely aligned, and you are encouraged to experiment with different shapes.

Next, click the  **Mirror** tool to mirror the spline across the centerline you created earlier. Onshape will prompt you for the order in which objects should be selected. The final result will look like [Figure 19](#), and any future adjustments will keep both sides identical.

Warning

When making the spline, make sure the endpoints don't terminate at a corner of the hull. This is to allow for the use of "imprinting" when extruding the sketch.

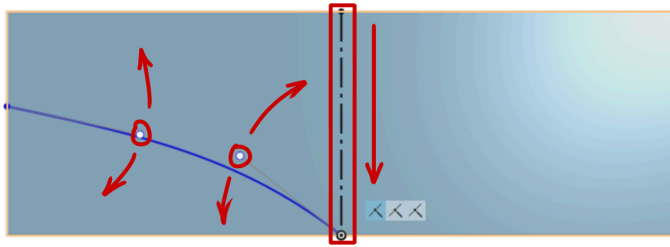


Figure 18

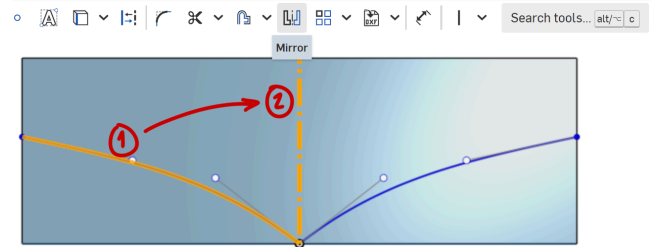


Figure 19

7 - Rounding the corners

Select the Fillet tool from the tool bar, and select the top two arcs found at the front of the ship. In the dialog box, set the Radius to 1.9 in. Notice that the curve propagates to the rest of the hull as seen in [Figure 20](#).

Warning

Setting the Radius to the height of the boat will result in the fillet failing. Therefore, we make the Radius 0.1in less than the height of the ship.

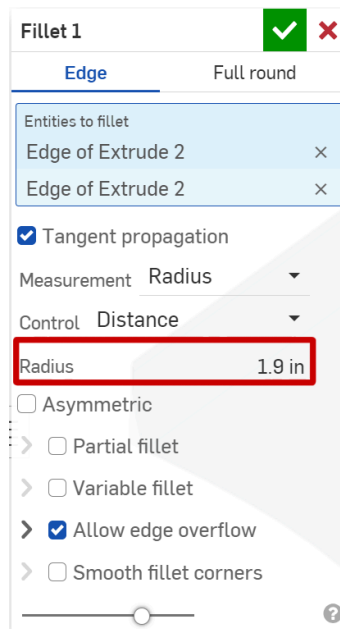


Figure 20

8 - Hollowing out the hull

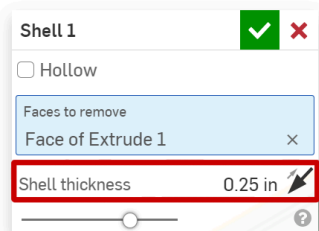


Figure 21

Now that we have shaped the exterior of the hull, we can use the Shell tool to carve out its interior. Select the Shell tool from the tool bar and click on the top surface of the boat. A preview of the feature should appear in the graphics area as seen in [Figure 21](#). Set the Shell thickness to 0.25in.

Tip

The resulting geometry of the hull interior is not ideal for placing electronics. Therefore, consider adding additional features to create a flat surface.

Conclusion

Congratulations! You have just finished making a model boat in Onshape! This model is a great base for any boat design, but needs to be expanded on to be viable for competition. Consider adding cutouts for your motors and propellers, and try to find ways to protect your electronics from getting splashed with water. Try and think of other ways that a boat can be modeled. Thanks for following along, and see you at the next meeting!

Learned Skills

- Make a sketch using lines and arcs
- Constrain sketch geometry using relations and dimensions
- Extrude add and extrude remove geometry
- Fillet and shell a model