How to Make a Basic 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 covers the modeling portion of your training and 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.

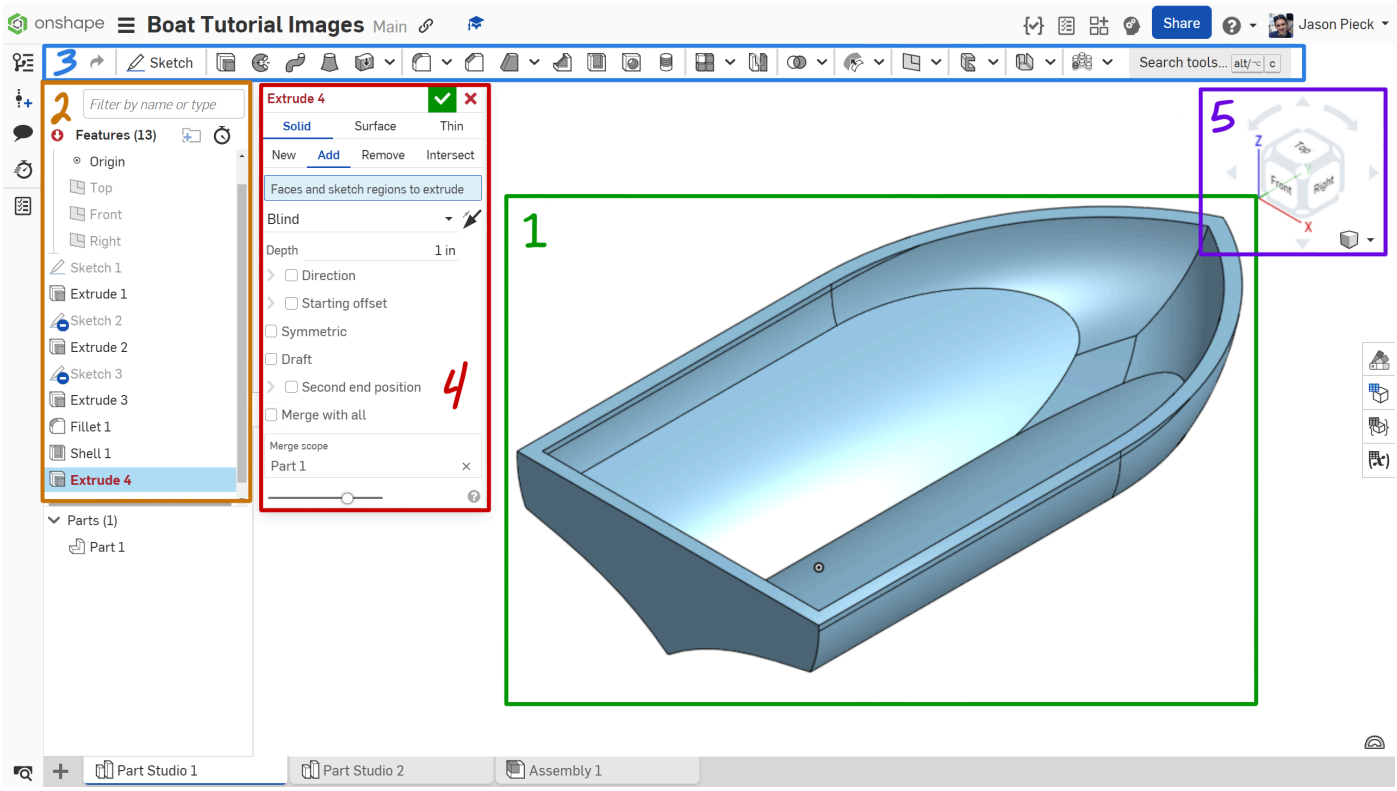

Figure 1

Overview of Onshape's User Interface

Looking closer at Figure 1, Box 1 (green) 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 (orange). 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 like `Sketch` and `Extrude` on the "Toolbar" in Box 3 (blue). Selecting a tool will open a "Dialog Window" as seen in Box 4 (red) and houses the parameters to modify the current operation. To see how tools affect your design, click on the "View Finder" in Box 5 (purple) 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.

Roadmap

We will start by sketching the basic shape of a boat before "extruding" (projecting) that outwards. Then through a series of additional sketches and extrudes, we will slowly carve away at it to refine its geometry. Lastly, we round out some the front edges for a classic boat form and hollow out the interior to make room for storage.

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 outcome.

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 List" as seen in Figure 2. To view the plane head on, select `Top` on the "View-Finder" also seen in Figure 2.

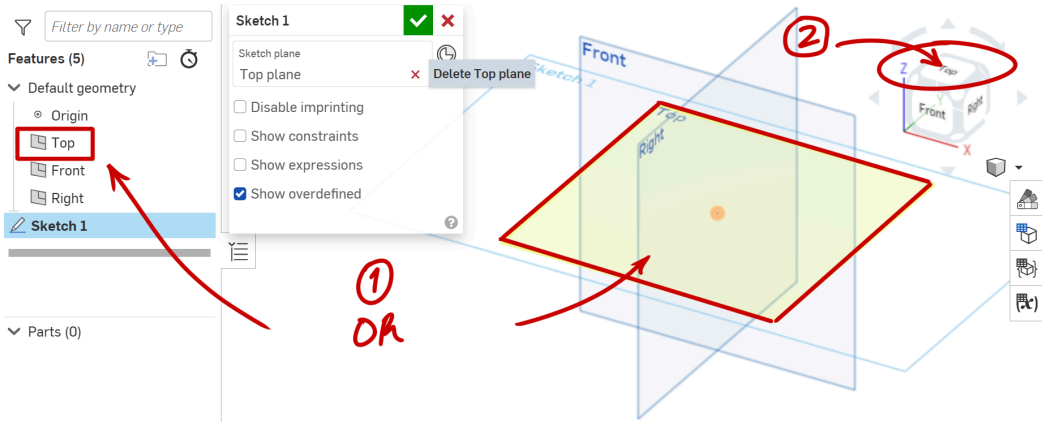

Figure 2

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.

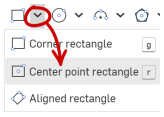

Figure 3

Click on the origin point and then move your mouse outwards in the direction indicated in Figure 4. Right now, the size and dimensions of the rectangle not matter, and does not need to mach Figure 4.

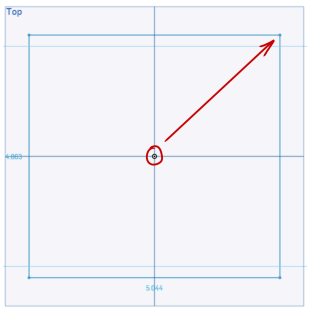

Figure 4

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. To check, zoom into the center of the rectangle. The origin and center point should be overlapping.

Now that we have rectangle, 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. When Onshape will prompts you to type a length, enter `6in` and press `Enter` on your keyboard.

To set the length of the bottom side of the rectangle, we are going to use a relation instead of the `Dimension` tool. Select two perpendicular sides and then select the `=` `Equal` relation from the dropdown as show in Figure 6. This relation forces the two sides to be equal such that changing the earlier set dimension will change both sides.

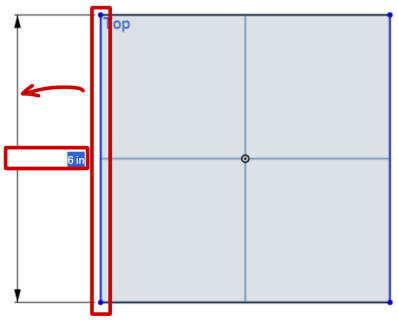

Figure 5

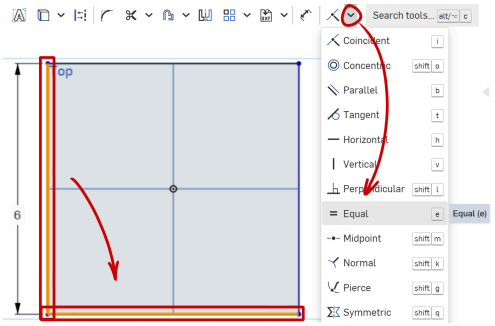

Figure 6

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 sided corner. As you approach the corner, 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 due to being fully defined.

Next, select the `Trim` tool and click and drag your mouse along the paths of the arrows drawn in Figure 8. This process will remove parts of the circle from the sketch. Once you are finished, your sketch should look like Figure 9. Select the green check mark as seen in Figure 9, and your sketch is finished!

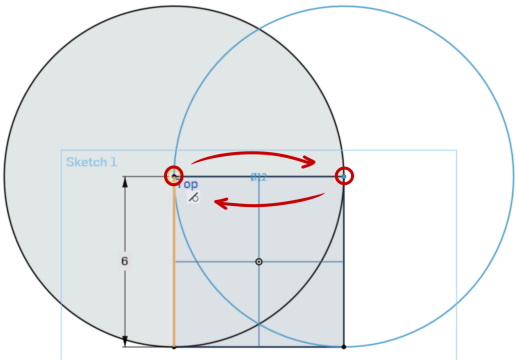

Figure 7

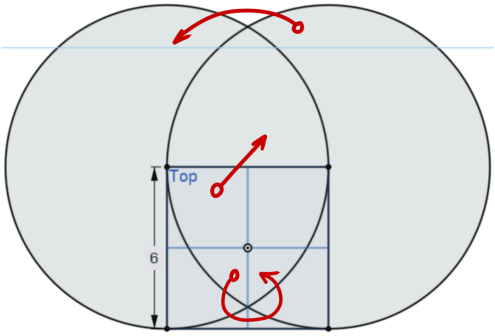

Figure 8

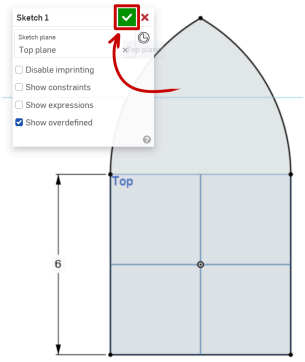

Figure 9

4 - Extruding the sketch

Click the `Extrude` button in toolbar and then select the face of sketch 1 as shown in as seen in Figure 10. Similar Step 1, you can also click the sketch from the "Feature List". Set the `Depth` of the extrude to `2in` and click the check mark. If you reorient your part, it should now look like Figure 11

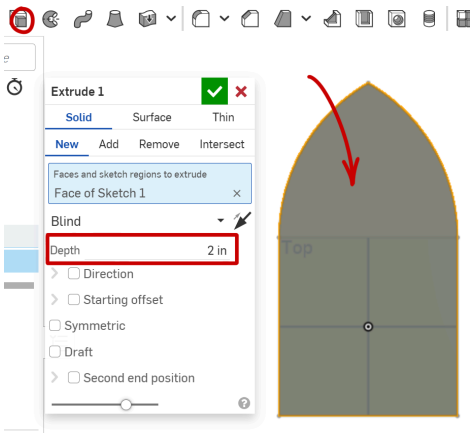

Figure 10

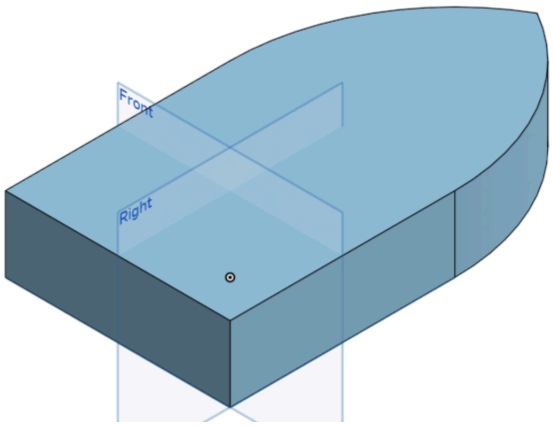

Figure 11

5 - Sketching with splines and lines

Create another `Sketch` and select the flat side surface of the boat for the `Sketch Plane` seen in Figure 12.

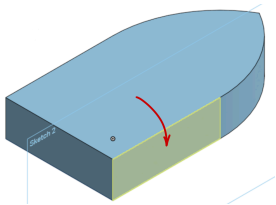

Figure 12

> **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

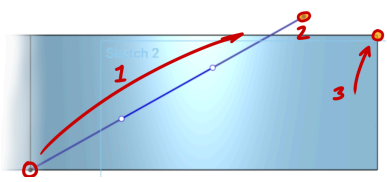

Figure 13

Looking at the tipped end of the boat, add a spline using the `Spline` tool. Start at the point where the middle face line meets the bottom edge of the boat and then click a short distance above the top of the boat as shown in Figure 13. After adding a second point, press `Esc` twice on your keyboard. Once your mouse is no longer a crosshair, 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. The position of each node along the line does not need to be exact to Figure 15, and you are encouraged to adjust them to your liking.

Complete the sketch by using the `Line` tool to draw two lines shown with arrows in Figure 16 to close the sketch. If the sketch is successfully closed, the interior of the sketch will become slightly darker in color.. Exit the sketch, and make another extrude. This time, select `Remove`, and change the end condition to `Through all`.

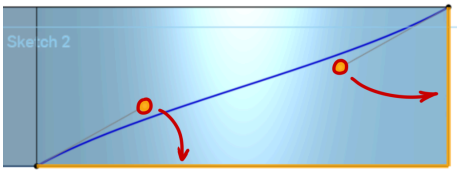

Figure 14

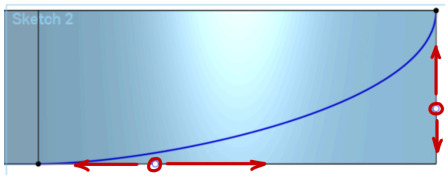

Figure 15

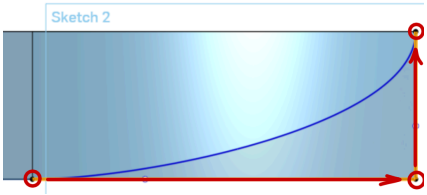

Figure 16

6 - Extrude-remove the sketch

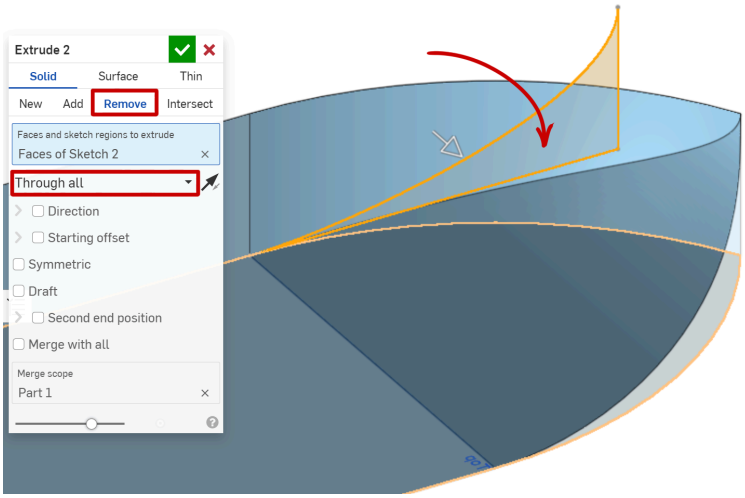

Figure 17

Exit the sketch, and make another `Extrude`. Select the previous sketch in the graphics area.

> **Warning**
> Onshape will not be able to extrude-remove a sketch if is it open and will warn you in the dialog box with `Missing Faces`. Repeat the Step 5 to correct this.

Since we are trying to remove material, select `Remove` from the second row of options in the dialog box and change the end condition to `Through all` as seen in Figure 17. This will project the sketch through the entire model and remove any interesting geometry, resulting in a curved hull nose.

7 - Shaping the bottom of the hull

Create a third `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 as seen in Figure 18. Make this line a construction line by selecting the `Construction` tool.

Next, using the `Spline` tool, create an arc same similar to Figure 18. Once more, the nodes do not need to be precisely aligned, but should closely match Figure 18.

> **Note**
> Here, we do not need close the sketch like in Step 5. This is because the flat sketch surface allows us to use "imprint" the already existing geometry of our boat to the sketch.

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.

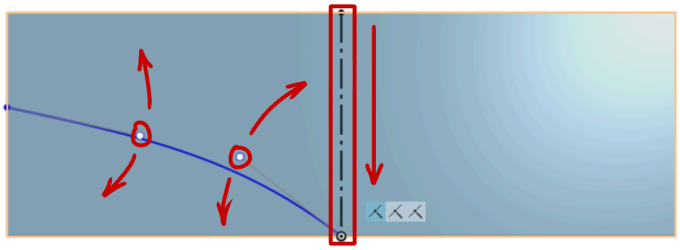

Figure 18

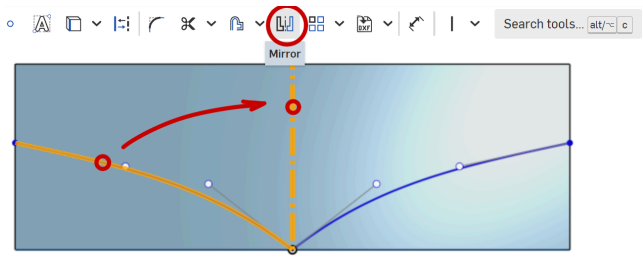

Figure 19

8 - 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 two edges we selected create a curve that propagates to the rest of the hull as seen in Figure 20. Lastly, click the check mark to close the dialog window.

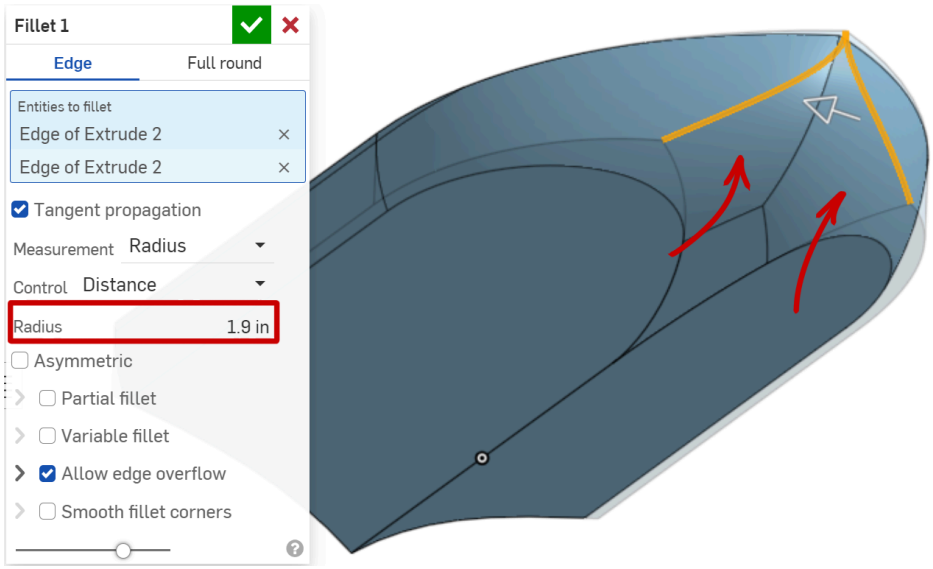

Figure 20

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.

9 - Hollowing out the hull

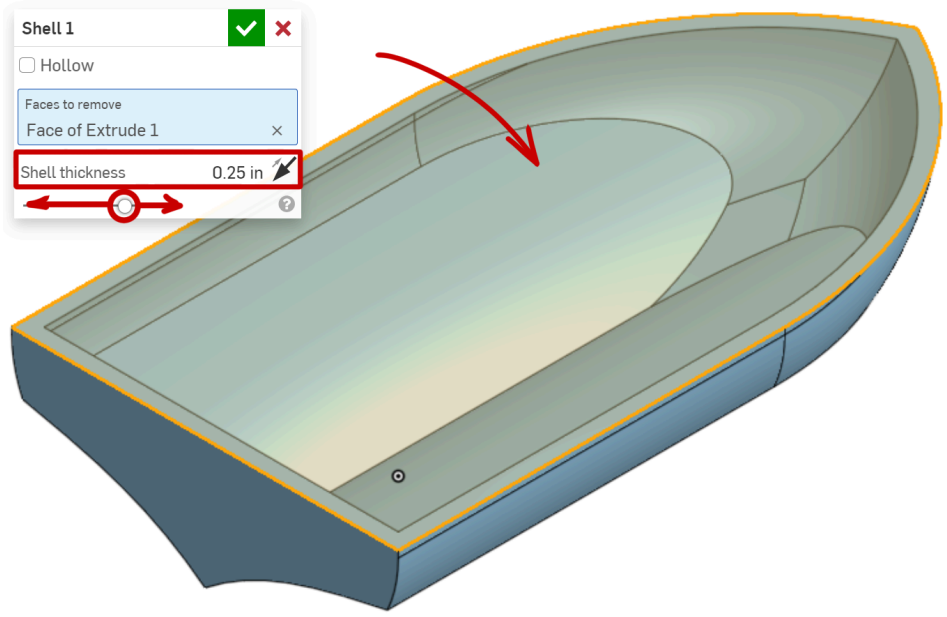

Figure 21

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. Drag the slider directly under the `Shell thickness` to fade in and out the preview of the feature.

Set the `Shell thickness` to `0.25in` and exit the dialog window by clicking the check mark as seen in Figure 21.

Tip

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

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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 an 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 rectangles, circle, lines and splines
- Constrain and modify sketch geometry using relations and dimensions
- Extrude-add and extrude-remove geometry
- Fillet and shell a model