# Introduction to OnShape

This guide provides an introduction to the Computer Assisted Design (CAD) program OnShape. OnShape can be used without charge by educational users and is the CAD program adopted by the International Technology Education and Engineering Association for its use in Learning by Design curriculum.

In this introduction, you will learn to create the endcaps for the armature of a linear motor. The armature controls how far the magnet in the center of the solenoid tube moves. The end cap protects the armature when it strikes the end post that limits the movement of the armature. The end cap is shown in red in the diagram below.
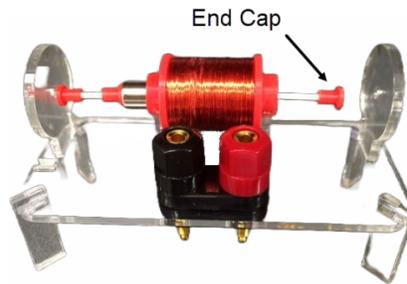
The end cap consists of a hollow tube with a disk on the end. The hollow tube fits on the armature. The disk on the end of the end cap strikes the end post.

This object was selected as an initial project because it is relatively straightforward, consisting of only two elements. The process will involve three steps: (1) the disk will be created, (2) then the hollow tube will be created, and finally, (3) the two elements will be joined to create a single object before printing.
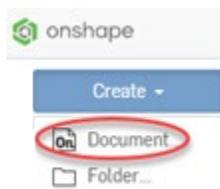
Setup

Go to https://www.onshape.com/en/ and select *Create a Student Account* in the top right corner of the screen, above the *Request a Trial* button.

Enter the required information and select *Create EDU Account*.

Starting a Document

1. Sign into OnShape
2. Select the *Create* button in the top left corner of the screen and select *Document* from the dropdown menu.

3. Name the document "End Cap" and Select *Create*.
4. If prompted, select *Millimeters* as the unit of measurement.

Creating a Sketch

Three dimensional designs can be viewed as a group of two-dimensional drawings that interact with one another. The default view in Onshape shows all three sketch planes.
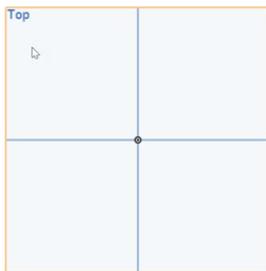
1. Select the top plane by clicking *Top* on the navigation cube in the upper right corner of the screen.
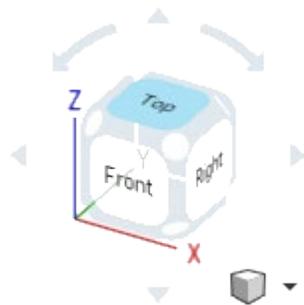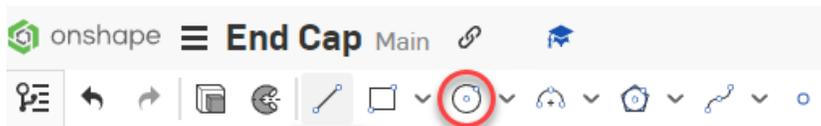

2. In the upper left corner of the screen, click *Sketch*.

3. Click the work plane to select the top plane as the area you want to use for the sketch.


4. Select the *Circle* tool from the menu bar at the top of the screen.


5. To draw a circle, click anywhere in the workspace and then drag the mouse. The number that appears will tell you the diameter of the circle. To manually set the circle size, release the mouse, type the desired diameter, and then press *Enter* on the keyboard.
6. Draw three separate circles with the following diameters: 2.25mm; 5mm; and 8mm.
7. Press *Esc* on the keyboard to cancel the circle tool.

8. Align the circles with one another by selecting the dot in the center of one and dragging it to the dot in the center of another.
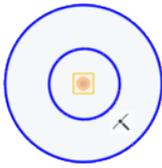
9. Release the mouse and the two points will snap together. Repeat this process so that all circles share the same center point.
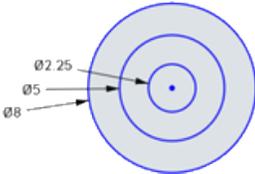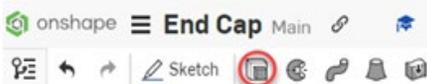
10. Select the green check mark in the *Sketch* window to complete the sketch.
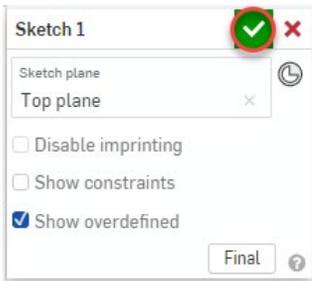
Extruding a Shape

1. Select the *Extrude* item from the menu bar.

2. From the *Extrude* menu, select *Solid* and *New* to create a new solid shape.

3. Click inside of each circle to select the sketches to extrude. Once selected, the sketches will turn dark grey.
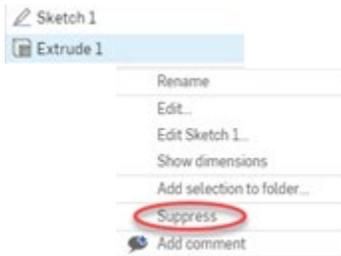
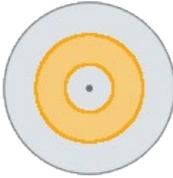4. Set the extrusion depth to be 2mm, and then click the green check mark.

5. Double click on the sketch in the navigation menu to return to the sketch.
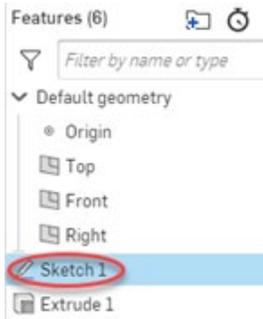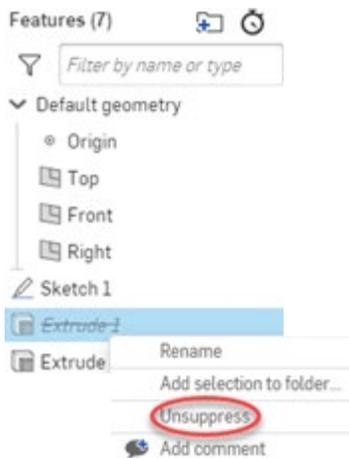

6. Right click on the extrusion and click *Suppress* from the drop-down menu. This will hide the shape you just created.

7. Repeat the extrusion process, selecting only the middle ring. This time, set the depth to 7mm.
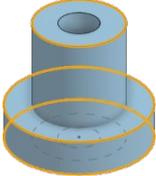

8. Right-click the first extrusion and select *Unsuppress* from the dropdown menu.
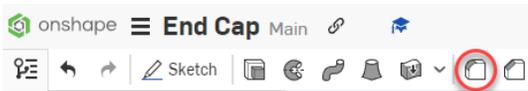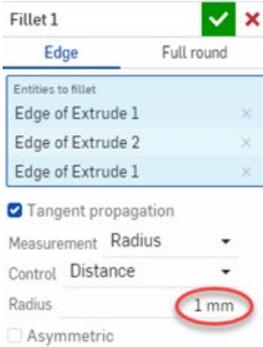
The object should now look like this.

9. To round the edges of the endcaps, select the edges to round.

10. From the menu, select the *Fillet* tool.

11. Set the fillet radius to "1" and click the green check mark. (Note: A fillet radius can never be larger than its smallest adjoining side.)

Joining Parts

So far, two separate parts have been created. These two parts will need to be joined before printing.

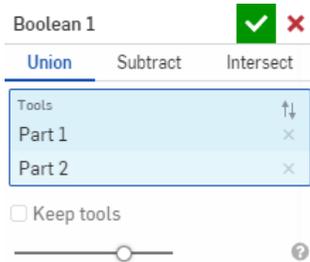1. Select both parts from the part menu. This menu is in the lower left side of the screen.
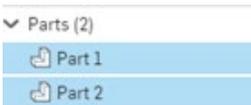
Once selected, they will turn yellow in the workspace.

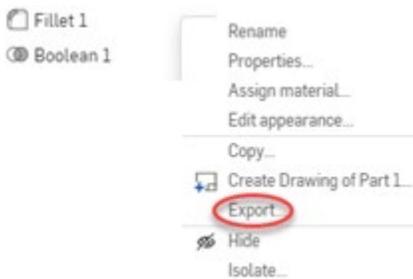2. From the menu, select the *Boolean* option to join the two shapes.

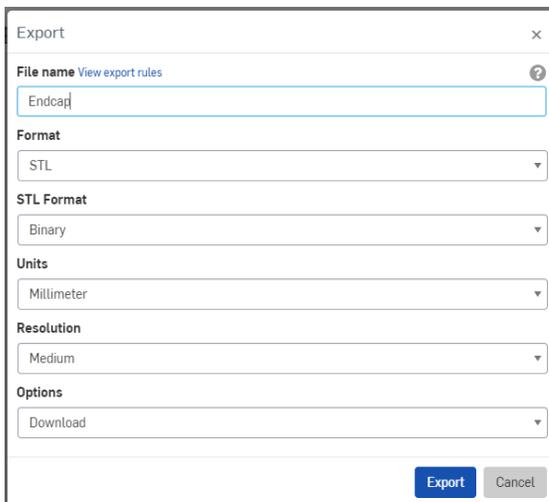3. Select *Union* and click the green check box.

Exporting a Part

1. Right-click the part you want to export (in this case, Part 1) in the menu and select *Export*.

2. In the Export window, name the file, set the format to STL, and select the units of measurement that match the units (mm) in the design. Then click *Export* to save the file.

Onshape automatically saves progress as you work, so there's no need to save the document before closing the browser window.