



TOPIC

Curriculum Guide

Week 2

07/10 - 07/16



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Announcements

In Week 1, you learned how to navigate the Onshape user interface and basic sketching functions.

This week, you will be introduced to basic tools for creating CAD parts. We will explore:

- Basic Part Tools
 - Extrusion and Cuts
 - End conditions
 - Revolve
 - Fillet
 - Sketches on parts
 - Mirror feature
- Part Appearances
 - Materials
 - Face colors

Reminders:

- Please complete all tutorials by July 16, 11:59pm
- Mentor office hours can be found in the “Schedule” tab of the Discord server.



- Note that the 20-min grace period applies to all mentor's office hours – if no one shows up to the call 20 minutes after the indicated start time, the mentor can disconnect from the call for the week.

Weekly Learning Goals

- Using construction lines to define and aid with sketching
- Mirroring and dimensioning functions
- *Cut / Extrude* with further options to specify the operation (cut to face, blind)
- Applying *feature linear pattern* to efficiently reproduce the same features
- Using *offset* to efficiently create borders

Tutorial Breakdown

This week will be OnShape intensive! We will be making all the individual parts of a cabinet / treasure box to practice our CAD skills. Next week, we will assemble them into one final product. The tutorial videos are listed as below. Below is some information supplemental to the video guides.

Additionally, here are some tips to trouble-shooting in CAD:

1. Take breaks! CAD softwares can be frustrating. If you have been spending > 1 hour on a part, rest for a bit before diving back
2. Ensure you understand the “**why**” behind each step in making a part. For further clarifications, ask your mentor!

Part 1: Modeling a cabinet

[Fundamentals of 2D Sketch and 3D Parts I | Video 2 Week 2](#)

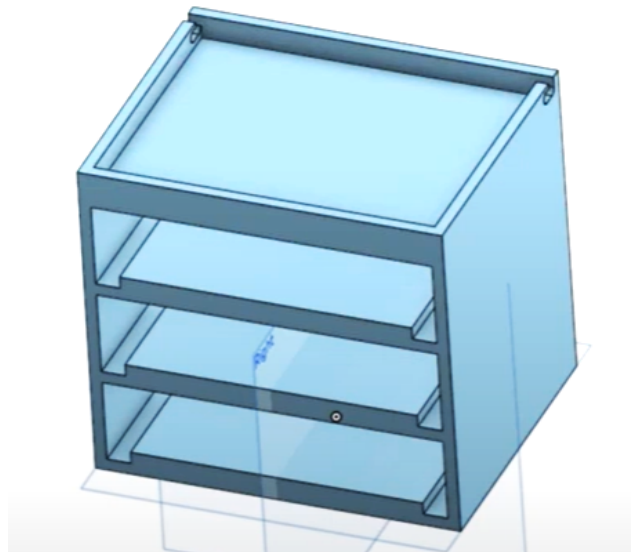


Figure 1. Finished model of the cabinet

Video 8:23 - *What does it mean to be fully defined?*

In a high level, fully-defining a sketch / model means that any future changes to the dimension will not affect the rest of the sketch / model in unexpected ways. Defining the dimensions of a system is directly related to restraining how many degrees of freedom (DOF) it has.

In statistics and mathematics, **degrees of freedom** is the maximum of independent values that have the freedom to vary. In the context of CAD, having an under-defined sketch means that if you change one dimension (e.g., height), this change can also affect other dimensions (e.g., width). Conversely, having an over-defined sketch means that you cannot change the dimensions of a shape at all.

Part 2: Modeling a lid

[Fundamentals of 2D Sketch and 3D Parts II | Video 3 Week 2](#)

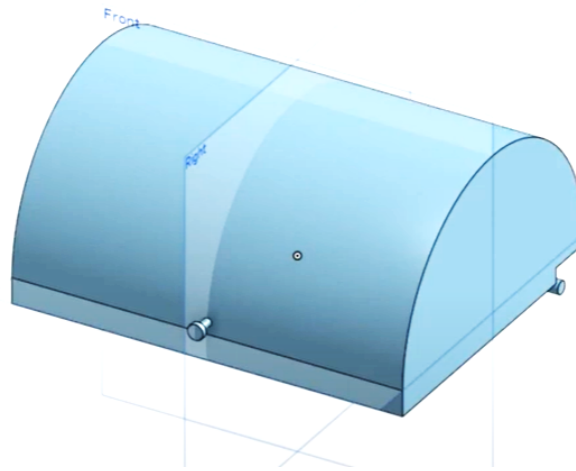


Figure 2. Finished model of the cover

Video 12:22 - *reminder* that the pegs on the sides of the cover does not have to be in the same dimension as the one illustrated in the video! Be creative and create your own dimensions.

Part 3: Modeling a drawer

[Fundamentals of 2D Sketch and 3D Parts III | Video 4 Week 2](#)

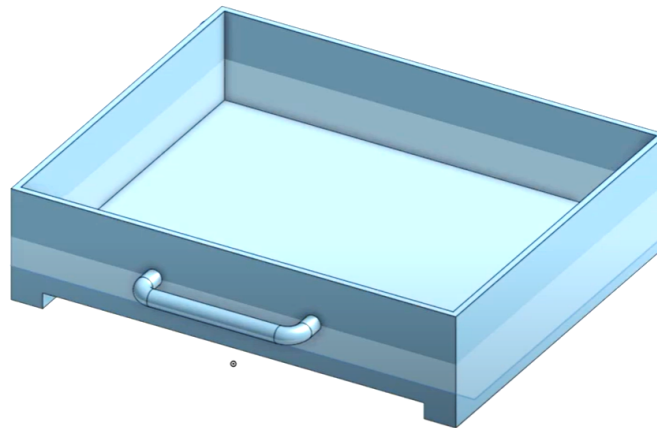


Figure 3. Finished model of the drawer

Note: if you've used different dimensions for the cabinet in video 1 (Part 1: modeling a cabinet), ensure that the dimensions of the drawer matches those of the cabinet. In the next video, we will be assembling the drawers to the cabinet, and you definitely want to make them fit!



Summary

This week, you learned how to make 3 different parts: cabinet, lid, and drawer. Specifically:

- Design and converting elaborate 2D sketches into 3D parts
- Using correct tools to fully-define and achieve accurate dimensioning of the parts
- Create multiple individual CAD components intended to be part of an assembly
- Changing the appearance of a model