

Schedule for remaining courses

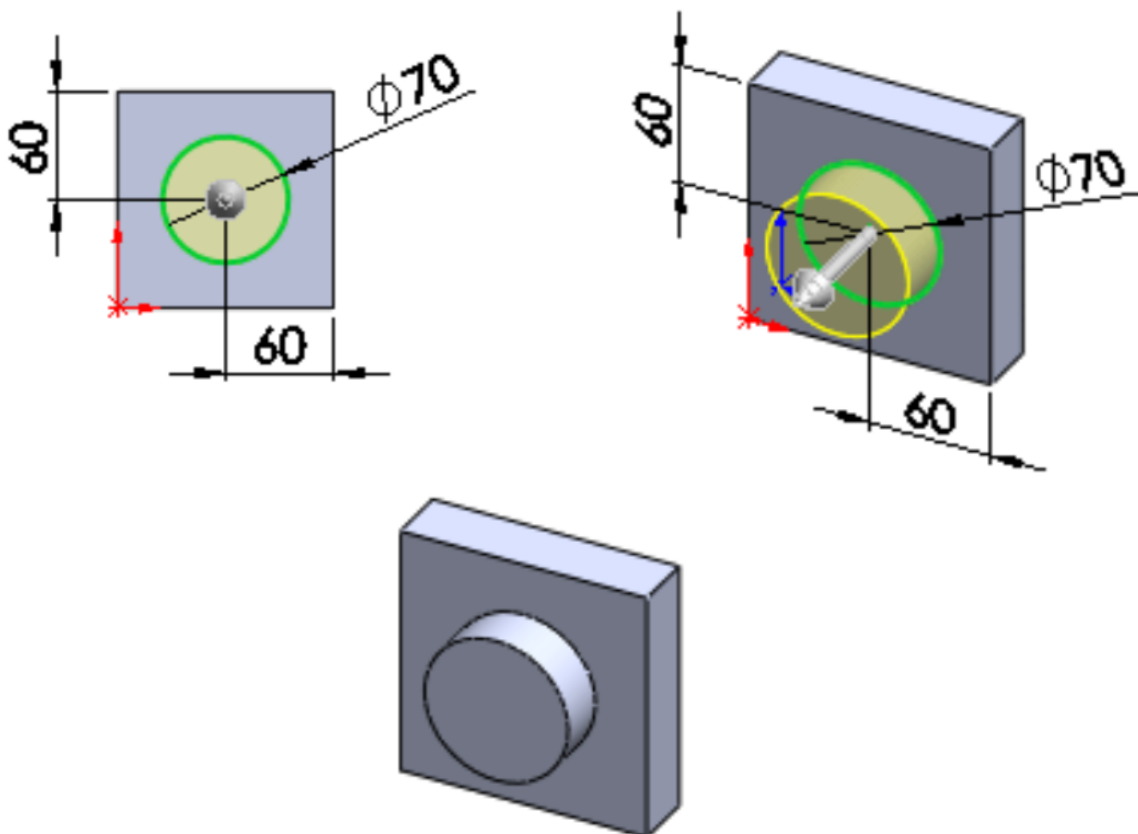
Nov 3	Intro to Solidworks 2018
Nov 5	Solidworks tutorial - solidworks file and PDF due by 11/8 midnight
Nov 10	Project
Nov 12	Project
Nov 17	Review

I. Solidworks

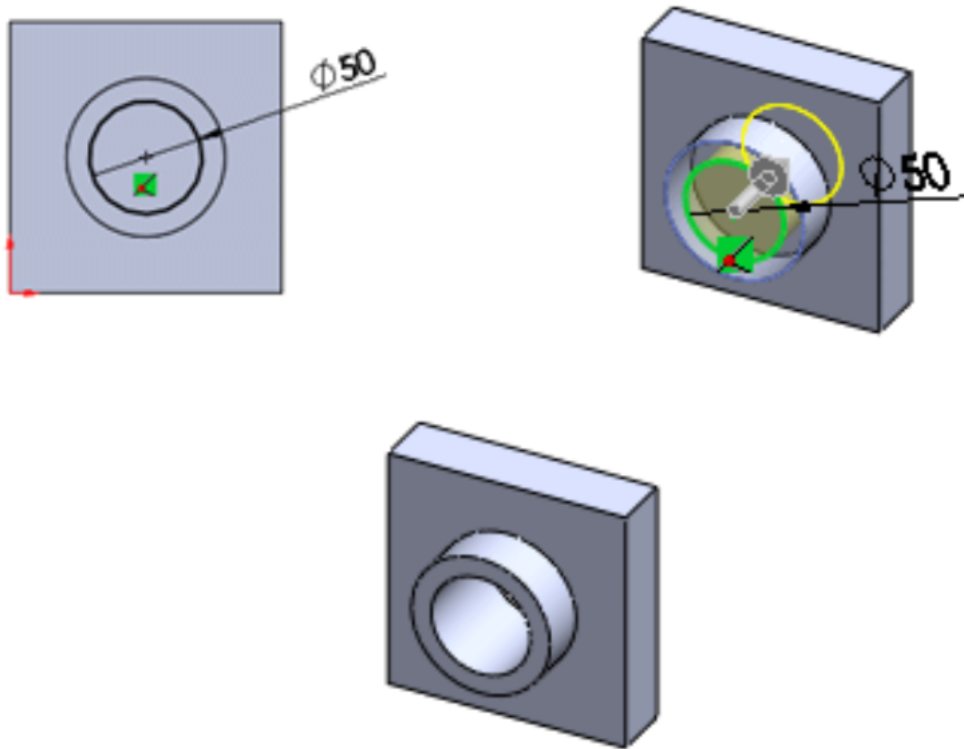
A. General idea

1. CAD = computer aided design
2. Create a 3D object by using primitive shapes
3. Some basic concepts:

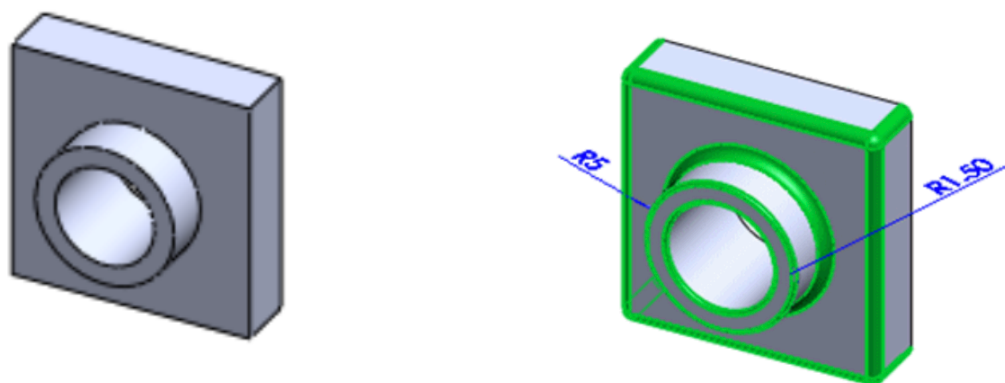
A circle is **extruded** to become a cylinder



An **extruded cut** is used to make holes. For example, a concentric circle is used to identify where to make a cut through a cylinder

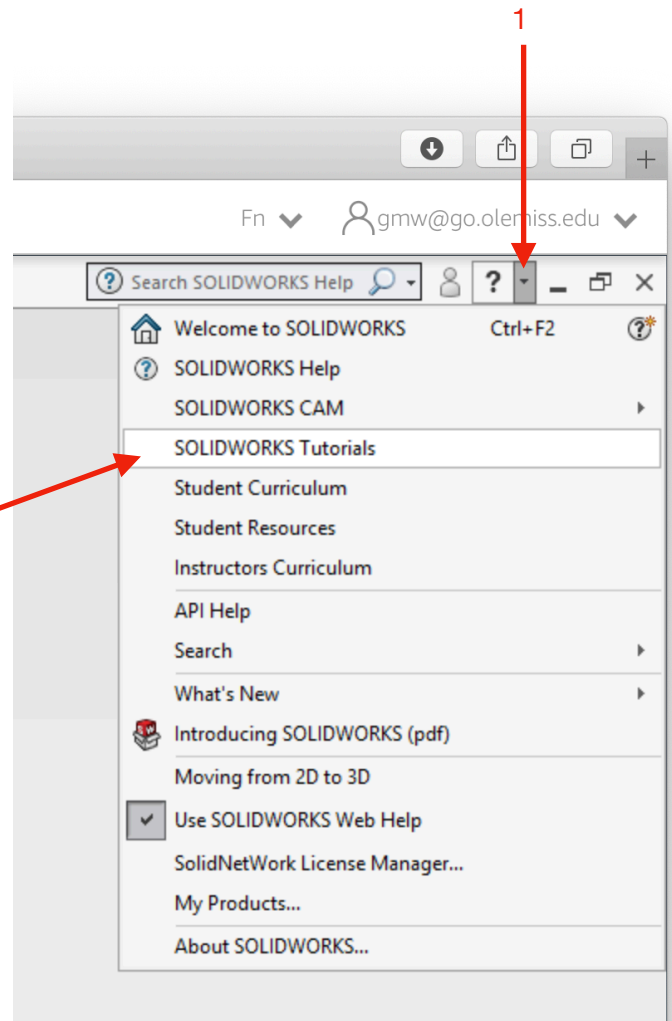
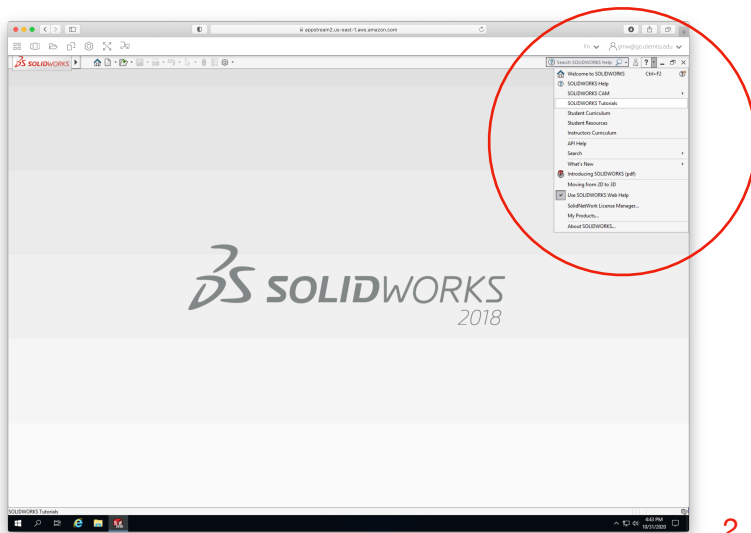


A **fillet** is a rounded corner

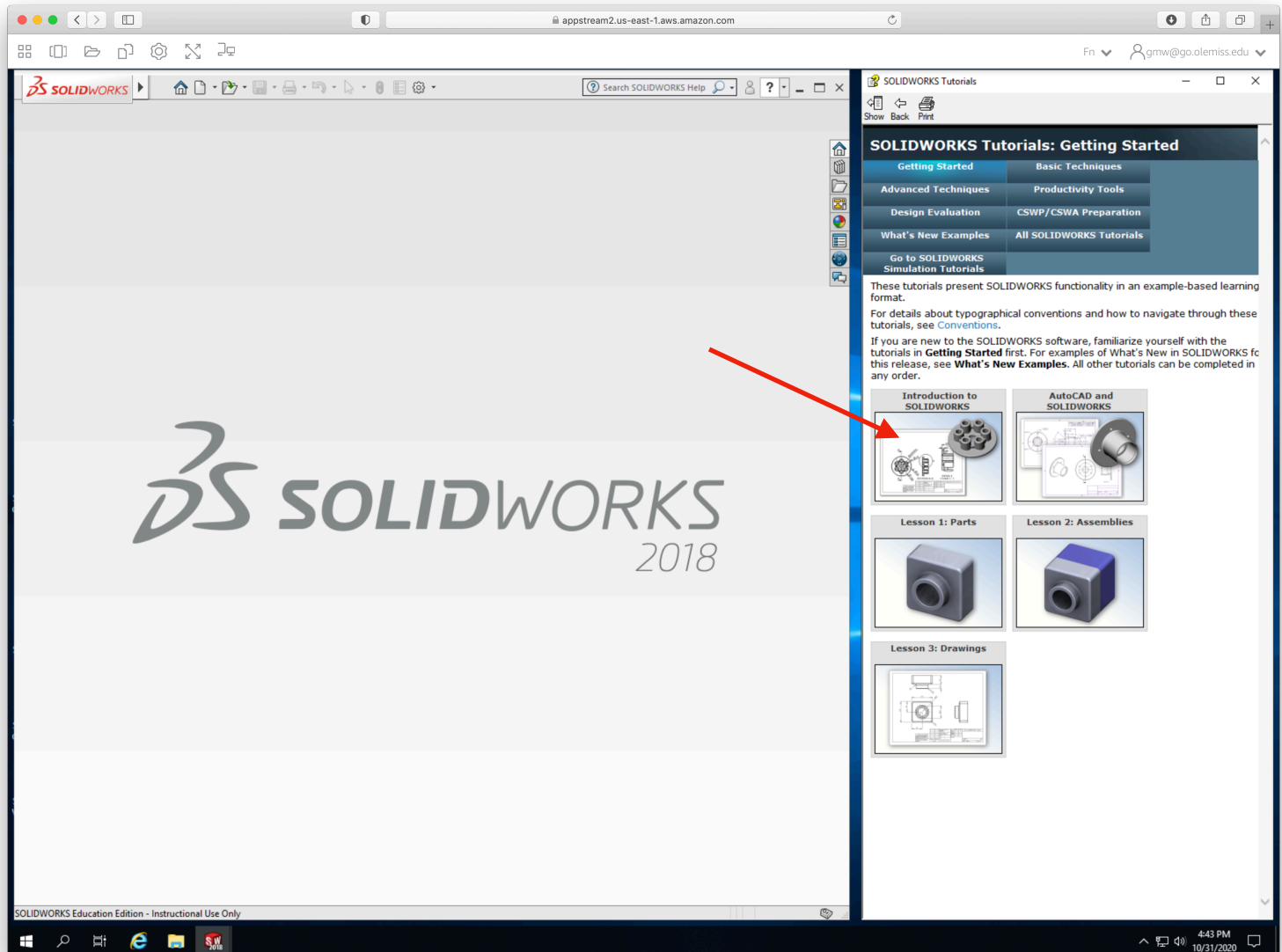


- B. Getting started (see Getting Started PDF)
- C. Start work on tutorial
 - 1. Your own copy of the SLDPRT file and a PDF of it **due by midnight 11/8**
 - 2. Once you log in and Solidworks loads, you can close this window if it appears

3. Then, open up tutorials



4. Choose this tutorial




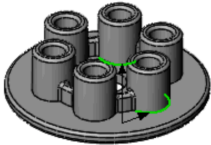
5. Stop after you complete this step



SOLIDWORKS Tutorials

Adding the Last Fillet

The last feature is a fillet that runs around the inside and outside edges of the patterned items.

1. Click **View** > **Hide / Show** > **Temporary Axes** to turn off the system axes.
2. Click **Fillet**  on the Features toolbar.
3. Select two edges as shown. You need to select one edge on the inside of the ring and one edge on the outside of the ring.



4. Click  to add a 2mm fillet.
5. Click **Save**  on the Standard toolbar.

The part is complete.

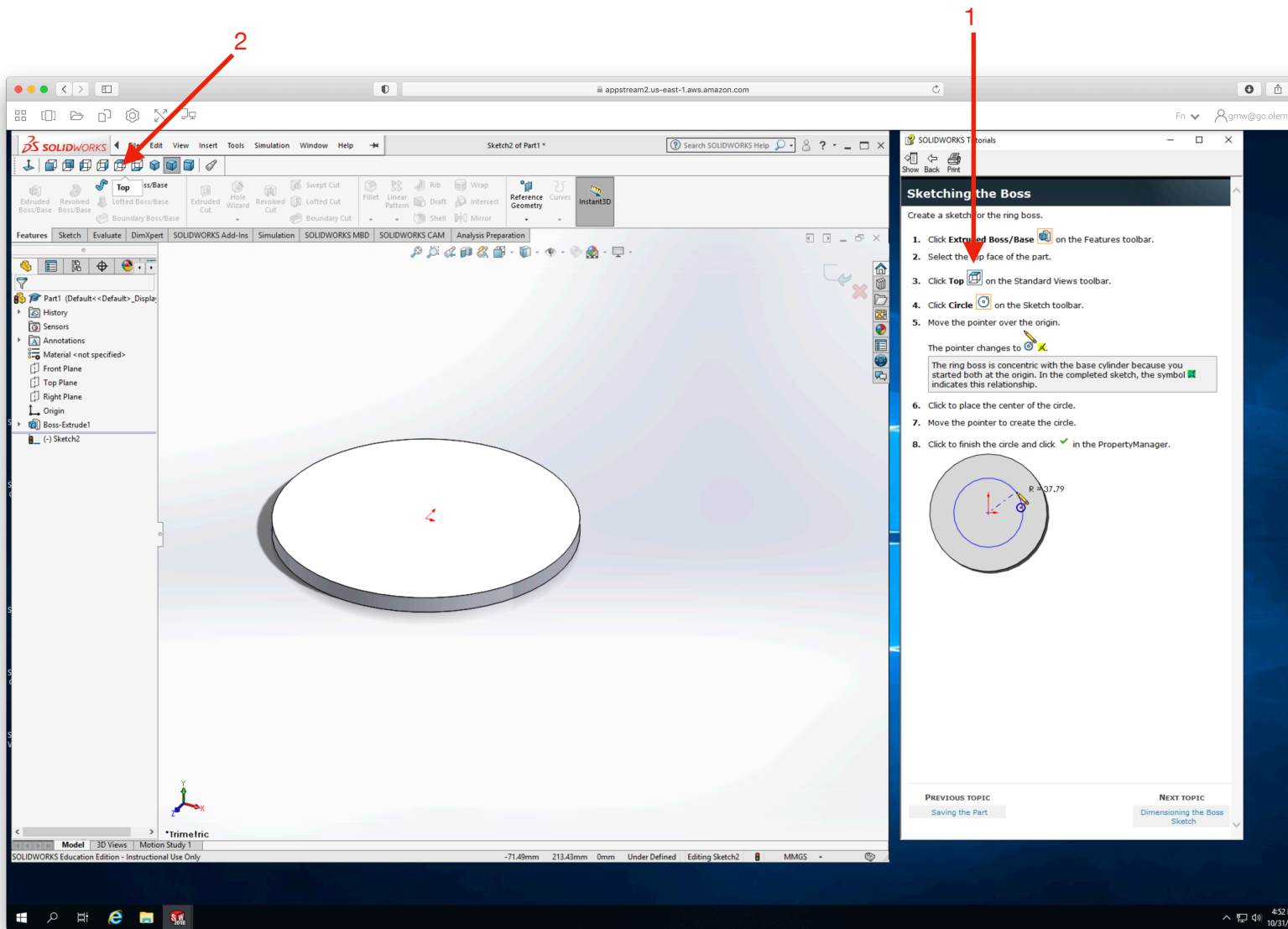
PREVIOUS TOPIC
[Creating a Circular Pattern](#)

NEXT TOPIC
[My First Drawing](#)

5:37 PM
10/31/2020

D. Misc tips for the tutorial

1. Click on buttons in the tutorial to make Solidworks show where the button is located on the menu.



2. When you get to the step that talks about a **centerline**, find the option here

The screenshot shows the SolidWorks interface with a 3D model of a cylinder. A red arrow points to the 'Centerline' option in the 'Line' flyout menu. The tutorial window on the right, titled 'Sketching the Tall Cylinder Extrusion', provides the following steps:

1. Click **Extruded Boss/Base** on the Features toolbar.
2. Select the top face of the base cylinder.
3. Click **Top** on the Standard Views toolbar.
4. Expand the **Line** flyout menu on the Sketch CommandManager and click **Centerline**.
The centerline acts as a construction line for the next circle. It keeps the center of the circle vertical with respect to the origin.
5. Move the pointer over the origin until the pointer changes to and click to start the centerline.
6. Move the mouse above the start of the centerline.
The pointer changes to to indicate the centerline is vertical.
7. When the line is about 45 mm long, click again to end it, and then press **Esc**.

At the bottom of the tutorial window, there are links for **PREVIOUS TOPIC** (Adding Fillets) and **NEXT TOPIC** (Completing the Tall Cylinder Extrusion).

3. Video from Solidworks 2016, so concepts will be the same, but button locations, etc., may not match exactly
 - a) <https://www.youtube.com/watch?v=PovLu7Mnhgc>