The following set of instructions are an optional replacement for the “SolidWorks Extrude” slides. This demo should help prepare the students for the Out of Class HW.

Student + Instructor:

1. Open Extrude.BLK.O_Base.SLDPRT file located in SolidWorks 2 EEIC website page. Note that the sketch is **Fully Defined** using geometric (vertical, horizontal, coincident, equal, symmetric, parallel, etc.) and dimension constraints which will be covered in upcoming lectures.

2. **Select Sketch 1** to turn the existing sketch **BLUE** and click on Edit Sketch and use the **Offset Tool** to create an inner offset .75” wide by first selecting the offset tool, second clicking on reverse to force the offset inside the existing sketch, third setting the dimension to .75, fourth click on any line in the existing sketch and finally clicking on the green arrow to accept the offset. Note that using the offset tool avoids having to draw the inner pattern using the Line tool.

3. Accept the edit and select Feature and then Extruded Boss/Base using an outward depth of 1.0” as shown below to create the solid Block O. Accept the extrusion.

4. Select the front surface of the Block O (**turns BLUE**) and then Edit Sketch and apply 2 offsets of width of 0.1” as shown. For the outer offset select reverse and set the dimension to .10. The inner offset (no reverse checked) will need to be created by sequentially click on all inner line segments while depressing the Ctrl key.

5. Select Feature and then Extruded Cut setting the depth to .25”, selecting any line on the existing offset patterns and finally accepting the cut.
6. **Select the front surface** and then Edit Sketch. Select the “A” Text Box tool and type “OSU”. Uncheck the “Use Document Font” box and Select Points Points type in "96" in the generated window. Position the text box as shown by dragging the “dot” on the lower left corner. If it accidentally becomes “locked” to one of the drawing lines, delete the coincident restraint and continue to re-position the text. Accept the “OSU” sketch.

7. Select Feature and then Extruded Boss/Base and extrude the “OSU” text outward for for .125” and accept the extrusion.

8. Finally select the top surface again and then Edit Sketch select again the “A” text box and employ the default setting(Use Document Default) to create the label reading:

   Prof. SolidWorks

   Seat: Front

9. Position the label as shown and accept the sketch to produce the final part Drawing as shown. Use Ctrl 7 to zoom to the ISO view. Note that 2 positive and one negative extrusion were used to create the part. Remember that a completed part should have **NO "Sketch" in the Display Tree without a parent function above it except for the label containing name and seat number.**