Show "01 Solid Modeling Intro" slides quickly. “SolidWorks Layout” slides are on EEIC for reference.

The following set of instructions are an optional replacement for the “SolidWorks Layout” slides. This demo should help prepare the students for the Out of Class HW which is the View Options part below replacing "Front" with Student Name, Seat & Professor Name.

**Student + Instructor:**

Demo how to start SolidWorks using the SLDPRF shown on the EEIC website under Demo

1. Open SolidWorks and describe functions of Part, Assembly and Drawing
2. Place "View Option Part" in EEIC website in your local directory
3. Open “View Options Part”, noting this is a previously constructed part
4. Left click on SolidWorks in the upper left corner to get the ribbon expansion and then left click on the "PIN" to lock the display. This setting will be maintained in all future SolidWorks usage.

**Shows how to “clean up” the history tree eliminating unnecessary entries.**

Right click anywhere in the Model Tree area and select "Hide /Show Tree Items". Change the values as indicated below to hide the Sensor, Annotations and Material fields in the Model Tree. This setting will be maintained in all future SolidWorks usage.

5. When printing a SolidWorks home work problem, all “shadows” and “reflections” should be suppressed.
6. To establish the ANSI standard for dimensioning a part, go to Tools/Options/Document properties and even if you find ANSI (which is the desired format under (Overall drafting standard), open the menu using the down arrow and select anything else (like ISO) and click OK.

7. Then return to Tools/Options/Document Properties and select ANSI and click OK. When you place a dimension you will find the correct format. **Note that you may be required to repeat this procedure later in the course on when you “extract” a part to a SolidWorks drawing.**

8. Describe the construction sequence used to create this part:

   (a) Choose a **sketch plane**... show planes in Model Tree

   (b) Make a **sketch**, exit the sketch and **extrude** or **revolve**

   (c ) **Repeat** (a) & (b) as required to build the part

9. View Model Tree & describe functionality ➔ to view or change previous work & auto update

10. View Tool Ribbon and focus on Sketch and describe **Drawing Tools** – line, circle, arc, rect.

11. View Tool Ribbon and focus on Feature and show **Extrude**(+ or -) & **Revolve** buttons

12. Use “View Orientation” to reposition the BLOCK in multiple positions

13. Depress Mouse wheel and rotate object

14. Rotate Mouse wheel to show Zoom (in & out)

15. Use “Display Options” and demo views- Shaded with Edges, Shaded and Wireframe
16. Re-establish the Shaded with Edges view

17. Note the “IPS” setting in the bottom Status Bar (most used IPS→inches & MMGS→millimeters)

18. Open Boss-Extrude 1 Sketch4 and note dimensions and “Fully Defined” (use zoom to reposition part)

19. Exit Sketch 4 – demo both right click and then select Exit Sketch in created window as well as select Exit Sketch in Tool Ribbon as alternative methods for exiting a sketch

20. Open Boss-Extrude 2 Sketch5 and note that “A” drawing tool (text used to create FRONT)

21. Double Click on FRONT, Add an “x” and click on green arrow to accept

22. Open all Features in Model Tree (click +) and move cursor down through tree-note Part highlights

23. Note that the SolidWorks HOT KEY Shortcuts are available on the SolidWorks1 page under “Introduce SolidWorks Hotkeys in PDF or Word”

24. Click on the plus to expand Cut-Extrude2 and select Sketch 5 and open with Edit Sketch. Double left click on the FRONT text to open the dialog text box and change FRONT to Student’s name (best if done using 2 lines), grab the DOT to reposition text block so name is on the object surface and Exit Sketch.

25. (Use this step or step 26, not both) Finally select the Front surface and then Edit Sketch and select the “A” text box and employ the default setting (Use Document Default) to create the label reading:
   Prof. SolidWorks
   Seat: Front

   Change font to around 22 points and accept with green arrow.
Position text by grabbing and moving the DOT. Under Display/Delete Relations select Add a Relation and click on FIX icon and accept with green arrow. Exit Sketch and select CTRL-F7 to zoom and place in ISO orientation. Print and submit the part file as an optional homework assignment. Have the class answer questions 1-4 below before proceeding. (Note that this DEMO is the only one where the results can be optionally submitted for the homework assignment)

26. (Use this step or step 26, not both) Optionally show the students the instructions for the SolidWorks Layout HW problem contained first part of SW-01 Homework found on the EEIC SolidWorks1 page. This HW uses the View Options Part used in this demo. Delete the SLDPRT file without saving it.

NOTE 2: Under Content on the EEIC website, there is a document called “SolidWorks Settings and Helpful Hints” which provides solutions to many of the issues new SolidWorks users experience. One of the most valuable hints describes how to reset the SolidWorks views, ie., make the current front view a side view, etc.

HOMEWORK QUESTIONS *

1. What was the shape of the block in Assignment 1 after the first extrusion (Boss-Extrude 1).
   
   ?

2. What feature of the design tree corresponds to the creation of the inclined plane?
   
   ?

3. Which feature corresponds to the word “BACK”?
   
   ?

4. How many surfaces of the object can you see from the bottom view?
   
   ?

*Instructor Note: Sweep the ? below the questions and change color to black
The following instructions are an optional replacement for the “2D Sketching” slides

Student + Instructor:

1. Open a new Part Drawing, set the scale to IPS and select “Sketch” to use the line tool to construct the shown sketch located at the origin on the front sketch plane (long vertical line should be about 2”)
2. Exit sketch and use Extruded Boss/Base using about 1” in both directions (click on Direction2 box for backward direction), then select on green arrow to show extruded construction.
3. Click on Undo arrow as shown to remove extrusion
4. Click on Revolved Boss/Base and select long vertical line to create revolved part
5. Delete the revolved part (including the sketch) to leave an empty part file.
6. Select the FRONT PLANE and click on Sketch (use Ctrl 8 if not already Normal To)
7. Describe the use of the Trim and Offset Tools
8. Describe the use of the various drawing tools
9. Using the Line tool, draw a ~2” square located at the origin (do not set dimensions!)
10. Demonstrate that the sketch can be reshaped by pulling any line with the mouse
11. Reshape to ~2” height and ~4” width
12. Draw a Center Point Arc on the left side and two circles on the right side
13. Use trim to remove the inner portion of the larger right side circle (trim to nearest) and then use delete to remove the extra lines.
14. Using the line tool, draw a centered ~1” square located as shown (using the top horizontal line as part of the square), use the Trim Tool (Trim to Nearest) to remove the extra line and the Sketch Fillet to establish the .25” radius fillets
15. (Use step 15 & 16 or step 17, not both) Exit the sketch and select the front plane and using Edit Sketch add a text box with Professor’s name and student’s seat number. Click on the DOT associated with the text box select the FIX icon. Select CTRL+8 to place sketch as "Normal To” orientation.
16. Save Select Zoom to Fit, print and submit object as an optional out of class assignment.

17. Optionally show the students the instructions for the 2D Sketching HW problem contained second part of SW-01 Homework (out of class assignment) found on the EEIC SolidWorks1 page. - due at the beginning of SolidWorks 2