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

Student + Instructor:

1. Click on the files on the EEIC website containing the Sections.SLDPRT, the Sections.SLDDRW and Sections_Demo_Base.SLDDRW place both on the Z drive. Open SolidWorks and using File/Open, bring both the part and the drawings into SolidWorks. Select the drawing for display, maximize the window and then use Zoom to Fit maximize the image.

2. The drawing is organized to be discussed from top to bottom and right to left for the Full and Half sections:
   - Discuss the "Use Section Views" text and then the "Section Views Components"
   - Discuss the "Full Section" noting the 3 labeled components (red/orange/blue)
   - Discuss the "Full Section" crosshatching and the section label, noting that if SCALE information appears, it should be removed (in title block). The section line is drawn within the Section View function as will be demonstrated later.
   - Discuss the "Half Section" noting that you need to activate the "Sketch" to draw the desired section lines. First while under View Layout, select Section View and click on Half Section in the dialog box. You can then click on the type of half section desired from the 8 available choices and then click on the location to position the section... in this case the center of the circle.

   - Left click on the screen in the area of the “OFFSET” section view new the bottom of the drawing and zoom in using the mouse wheel. Discuss the application of the offset section view when the desired features cannot be sectioned using a straight section lines. Note that where the offset section line creates a cut through solid material (see the vertical section lines), that cut does not produce a solid vertical line in the drawing since such a line would be confusing to the viewer. Use Zoom to Fit to restore the full drawing view.
Repeat the previous step for Broken, Revolved and Removed sections. Note that in the Revolved and Removed sections, the cutting plane is perpendicular to the element being section, but the produced section view is rotated 90 degrees prior to placement in the drawing.

3. This concludes the discussion of the drawing which should now be deleted from SolidWorks.

4. View the Sections_Demo_Base.SLDDRW drawing.

5. Select View Layout / Section View, verify the ‘Section’ is high-lighted in the dialog box and move the pencil icon to the RIGHT image and hover / hunt for the extension of the horizontal center line. Left click the mouse to select the starting location of the Section View and select the green arrow to accept. Pull the cursor down until the desired location of Section View is reached and then left click the mouse to anchor. (If the direction arrows ever appears in the incorrect direction select ‘Flip Direction’ in the dialog box to correct)

6. Select Annotation / Centerline and sequentially left click on the left and right sides of all the holes and finally accept the Centerlines by clicking on the green arrow or by hitting the Esc key. The Full section is now complete.

7. Again select View Layout / Section View, select the ‘Half Section’ in the dialog box and move the pencil icon to the RIGHT image and hover / hunt for the center point of the circle. Left click the mouse to select the starting location of the Section View and pull the cursor down until the desired location of Section View is reached and then left click the mouse to anchor.

7. Again select Annotation / Centerline and sequentially click on the left and right sides of the right side holes the Centerline by clicking on the green arrow or by hitting esc.
8. Since there is no hole available on the left side of the object and because we need to establish the left hole center line we will grab and extend the Center Mark downward as shown.

9. Next under Sketch/Line/Center line select ‘Vertical’ and ‘Construction’, draw the 2 short lines and accept it and then restore the location of the Center Mark to complete the Half Section.

10. Even though we placed two different sections on the same drawing for this demo, one would not do this for a standard section view drawing. Select Zoom to Fit, print and submit object as optional in class assignment.

11. Either edit the title block to add the students and instructor name and date and submit as the alternate In-class assignment or describe the In-class assignment (SW-21) shown below from the EEIC website which uses the provided SolidWorks part drawing.

4. SW-21 In-Class Activity - PDF and SLDPRF - due before midnight Friday