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

**Student + Instructor:**

Demo how to start SolidWorks

1. Open SolidWorks and select Part to create the object shown above. Note that this demo uses vertical and horizontal construction lines to reduce the number of required geometric constraints for this part which exhibits symmetry left to right. No dimensions (except for the required filet radius) are employed. Near the end of the construction, the sketch is “pulled” in multiple directions to assure that all the needed geometric constraints have been applied.

2. Change the Units to MMGS

3. Edit sketch on the FRONT Plane and draw 2 Horizontal construction lines (CL) of infinite length, one through the origin and one at about half way up toward the top of the screen. Accept using Esc key or the green arrow and draw a Vertical construction line.

4. Draw 3 small circles, 2 on the top Horizontal construction line and 1 on the Vertical construction line positioned roughly as shown. Make all 3 circles equal by holding the Ctrl key down, selecting the circles and clicking on **EQUAL** in the Add Relations dialog box and finally accepting the relation using Esc or the green arrow.

5. Draw 3 larger circles on the center points (CP) of the 3 smaller circles. Make all 3 circles equal by holding the Ctrl key down, selecting the circles and clicking on **EQUAL** in the Add Relations dialog box and finally accepting the relation using Esc or the green arrow.

6. Make the top 2 circle CP’s symmetric about the Vertical CL by holding the Ctrl key down and first selecting the Vertical CL and then the smaller arc of the top 2 circles and clicking on **SYMMETRIC** in the Add Relations dialog box.
7. Using the CP Straight Slot tool, draw and accept a slot about the origin.

8. Using the Line tool draw the indicated pattern noting that where the line is slanted, SolidWorks did not provide the usual “dashed” line to help in locating the correct point to “go down” to the lower circles outer left edge. Start the Line tool on the left side of the upper left circle at the Yellow diamond and finishing at the yellow diamond on the left side of the lower centered circle. Note that SolidWorks provides “help” in maintaining the first vertical and horizontal segments aligned, but no “help” in determining the edge on the lower circle so it is left drawn at any angle and will be adjusted once the outline is completed.

9. Continue with the Line tool as shown starting at the Yellow diamond on the right side of the lower circle and completing on the Yellow diamond on the right side of the upper right circle, again noting that the usual “dashed” line help was not provided where the sketch has slanted lines.

10. Continue with the Line tool as shown starting at the Yellow diamond on the left side of the upper circle and completing on the Yellow diamond on the right side of the upper left circle, again noting that the usual “dashed” line help was not provided where the sketch has slanted lines.

11. Select all the slanted lines with the Ctrl key depressed and then click on Vertical in the Add Relations dialog box and accept the constraints.
12. Desired
Trim away portions of those 3 circles and inspect all the intersection vertical lines with the circular arcs noting that some intersections have a TANGENCY constraint and some do not. The RED arrows indicate intersections without the tangency constraint. Apply missing tangency constraints holding the Ctrl key down and selecting the paired arcs and Vertical lines and selecting TANGENT in the Add Relations dialog box.

13. Select Sketch Fillet / Sketch Fillet, set the radius to 10mm and apply to the corners as shown in Yellow and accept the fillet.

14. Since we are not dimensioning this object, it is not possible to have the part “Fully Defined”, but you can check for the presence of full geometric constraints by using the mouse to “tug” on each of the lines and points to see if any unexpected distortions occur. One expects that the part will “flex” but maintain its SYMMETRY if all the geometric constraints are present. This “tugging” technique revealed that the part is missing an equal constraint on the left and right side vertical line segments as shown in RED or alternatively is missing alignment of the 2 bottom horizontal lines as shown in GREEN and can be corrected either by holding the Ctrl key down and selecting first the left vertical segment and then the right vertical segment and then selecting EQUAL in the Add Relation dialog box and accepting the constraint or by holding the Ctrl key down and selecting first the left lower horizontal segment end point and then the right lower horizontal segment end point and then selecting HORIZONTAL* in the Add Relation dialog box and accepting the constraint.

*Note that points or a point and a line can be forced into either horizontal or vertical alignment using the HORIZONTAL OR VERTICAL constraint. Line to line alignment can be accomplished using the COLLINEAR constraint.
15. A copy of the desired object was obtained by opening Geometric_Constraints_without_dimensions.SLDPRT and performing the Normal To function and “Snipping” and pasting the image. The Current part was adjusted slightly by tugging on various points, lines or arcs to produce proportions closer to the desired object. This adjusting step need not be performed since once DIMENSIONS are placed on the part, it would match the Desired Object, but is does show that the object can be reshaped as desired.

16. Click on any line on the sketch, select Feature and select Extrude Boss/Base and extrude sketch outward for 20mm. Save the created part as Geometric_Constraints_Demo.SLDPRT if the Student was able followed along and completed the construction. If not, do not save if only partially complete. In the following Dimensioning Demo, the student can use the supplied Dimensional_Constraints_without_dimensions.SLDPRT instead of the student created Geometric_Constraints_Demo.SLDPRT.

17. The part is now complete. It would have been possible to construct the part without using the horizontal CL to locate the circles and the use of the SYMMETRIC constraint, but construction would have required many more constraints. It is strongly recommended that this strategy be employed to complete the assigned out of class HW.