The following set of instructions are an optional replacement for the “SolidWorks Dimensional Constraint Modeling” slides. This demo should help prepare the students for the Out of Class HW Discuss that most manufactured objects have **specified precise geometrical and dimensional constraints** so that they can perform their desired function (your desk, computer & monitor and all their internal parts, the overhead lights in the room, the pen / pencil you use to take notes and almost everything you see in this classroom).

1. Open simple demo sketch (Dimensional_Constraints.SLDPR) and note that it has no dimensions and the warning “Under Defined” is displayed
2. Stress that before starting dimensioning, make sure that the correct units (IPS/MMGS) are set
3. Dimensions include SIZE & LOCATION, thus the circle must have a diameter and an X & Y location
4. Point out the 3 kinds of dim...**linear, radial** (diameter / radius), **angular** features on the sketch. Note that you should apply geometrical constraints prior to applying dimensional constraints
5. Note that the Geometric_Constraints_Demo.SLDPRT file supplied with this demo can be used to show the properly dimensioned part for students to compare with their results in step 14.
6. Open the Smart Dimension tool:

   **Shows use of dimensional constraints which will result in the part being FULLY DEFINED since the necessary horizontal and vertical geometric constraints have already been applied.**

7. Demonstrate **linear** dim.
   a. Use **line to line** to dim. **left side (2”)**
   b. Use **point to point** to dim. **bottom (3”)**
8. Demonstrate **radial** dim. To **SIZE** the circle (**diameter = 1”)**

9. **LOCATE** the circle center point using the **arc & left side line (1”)** and then the arc & bottom line (**1”)**

10. Demonstrate **angular** dim. Using line to line to establish the **60 degree angle**

11. Since the object was geometrical constrained initially (vert., hor. & perpendicular), once we applied all the necessary dimensional constraints, the SolidWorks bottom ribbon indicated **FULLY DEFINED**
12. Demonstrate the use of algebraic formulas in dimensioning by changing the dimension on the bottom line to **1.5 times the left side** as shown in the following instruction:
   a. Move the mouse around the various dimensions to show the assigned label, noting that the left vertical line is "D1Sketch1".
   b. Double click on the 3.00 inch dimension for the bottom horizontal line.

Constraints can be added in the form of equations by double clicking on the bottom line:

I. Entering “=” (the “=” is **needed retain** the formula for future calculations)

II. Entering a value or by left clicking on a dimension (select the 2.0 dimension) causing the label D1Sketch1 to be entered in the equation.

III. Using operations +, -, *, / ( = “D1Sketch1” * 1.5 ) and accept by clicking the green arrow. Note that there is now the ∑ summation symbol displayed in front of the dimension.

IV. Repeat for all dimensions using a multiplier of 0.5 so all will display the ∑ summation symbol except the left vertical line. No formula is required for the 60 degree angle.

V. Resize the left vertical line to 4 and note that the object size is now twice the original size and the proportions are maintained. Reorient object by sequentially selecting Isometric Orientation→Zooms( Ctrl + 7) & Normal To( Ctrl + 8) and dragging the new dimension to the left.

13. Have the students open either the completed **SW_3_IN_CLASS_OPT_1.SLDPRT** or the supplied **Solidworks_Geometric_Constraints_Demo....SLDPRT** for student not completing demo 1, use Ctrl + 8 or View Orientation Isometric to orient the part and then Edit Sketch1. Dimension using Smart Dimension tool. Note that if **SW_3_IN_CLASS_OPT_1.SLDPRT** is used, any **text block sketch (name,etc) should be deleted**.

14. Using either your completed sketch from the Geometric Constraints Demo (or the un-dimensioned part file on the EEIC web site) place all the dimensions shown in the figure below until the sketch is **FULLY DEFINED**. Note that usually you dimension distances between center points to make the part easier to manufacture. There would also be a note designating that the object is symmetrical left to right and top to bottom.

**Shows use of dimensional constraints to a more complex part which will result in the part being FULLY DEFINED since all the necessary geometric constraints have already been applied.**
16. Once the sketch has been FULLY DEFINED via dimensioning, exit the Sketch, highlight Sketch1, select Feature and then Extruded Base/Boss and extrude 20mm in each direction. Accept the extrusion by clicking on the green arrow and finally click anywhere on the SolidWork’s window to remove the construction lines to produce the finished part.

Usually on complex parts it is a good practice to initially draw and FULLY CONSTRAIN (Geometric & Dimensional) a major feature, like the small circles to the left, to establish the correct sizing. This tends to reduce future difficulties like drawing distortions or having the sketch “fold” on itself. Then additional sketch details can be added using “primarily” geometric constraints which will reduce the number of dimensional required later to FULLY DEFINE the sketch. Point out that there are usually multiple ways of geometrically constraining a part.
17. (Use this step or discuss the second part of the in class assignment on the EEIC website) 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.

1. 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 the object to the Isometric orientation. Save the files as SW_3_IN_CLASS_OPT_2.SLDPRT, print and submit object as optional in class dimensional constraints assignment.

The following suggestions will make the students Dimensional Constraint HW much easier:

A. Establish scale as IPS before starting the drawing
B. Establish vertical and horizontal construction lines through the origin (called Centerline in SolidWorks in the line drawing tool)
C. Draw appropriate circles on the horizontal construction lines, making them equal where applicable.
D. Establish dimensions NOW in the Dimensional Constraints HW (circle sizes and separation [76 inches])
E. Draw the outlines, taking care to keep the top and bottom lines Horizontal as indicated.
   CAUTION: Do not connect the diagonal lines to the circle midpoints
F. Add tangent constraints to diagonals intersecting arcs.
G. Trim circles where required
H. Re-establish any lost tangencies
I. Establish all EQUALITIES
J. Establish SYMMETRY about both construction lines.
K. Add DIMENSIONS in the Dimensional Constraints HW
L. Verify that your Sketch is FULLY DEFINED in the Dimensional Constraints HW
M. If not, tug on drawing to determine missing constraints and add them. For the Geometric Constraints HW, the sketch should “flex” but maintain its SYMMETRICAL orientation.