Dimensioning part 2
with SolidWorks

ENGR 1182
SolidWorks 09
Today’s Objectives

- Practice Basic Rules of Dimensioning *
- SW09 Activity
- SW09 Application

* The student is encourage to practice using the provided DEMOS under both SW 08 and SW 09.
Each dimension should be placed in a descriptive or characteristic view (Guideline 2) without dimensioning to hidden lines (Guideline 7) and without Over Dimensioning (Guideline 1).
Dimensions should be located outside the boundaries of the object (Guideline 3) whenever possible and should be spaced far enough apart to be read clearly since OVERALL CLARITY is the ultimate goal of Dimensioning.
Dimensions that are shared between two views should be located between those views (Guideline 3) with each feature being dimensioned in its CHARACTERISTIC VIEW (Guideline 2), Negative Cylinders being dimensioned where they appear as CIRCLES (Guideline 9) and avoiding dimensioning to HIDDEN LINES (Guideline 7). If SYMMETRY exists, a NOTE should be used (Guideline 6).
GUIDELINE 8
• A diameter symbol is always required before a numerical value.

GUIDELINE 9
• Positive cylinder (e.g. a rod) – dimension in a view where the side of the cylinder appears as a rectangle.
• Negative cylinder (e.g. a hole) – dimension in the view where the cylinder appears as a circle.
Do not over-dimension or under-dimension an object (Guideline 1) and if SYMMETRY exists, a NOTE should be used (Guideline 6).
Solid Modeling Dimensioning

Hole and Arc Details – Guideline 8

- Circular features of 360° - include the diameter symbol followed by a numerical value, e.g. Ø5.0
- Circular features of < 360° - include the radius symbol followed by a numerical value, e.g. R5.0

A center mark represents the center of an arc or circle and is used for dimensioning feature locations.

Centerlines are used in the view where the circular feature appears as a rectangle.

Symmetry
When symmetry is used to reduce complexity a note must be added.

Options include:
Symmetrical from left to right, front to back, top to bottom.
SolidWorks: A reminder on how to add Dimensions

Dimensions can be added using the “Smart Dimension” feature under “Annotation”.

Notes can be added using A Note under “Annotation” to reduce the number of dimensions by stating SYMMETRY or identifying fillet radii.
Dimensions Wrap Up

Rules of Dimensioning

1. Appropriate View
2. Clarity
3. Location of Dimensions
4. Cylinders
5. Amount of Dimensions

Application SW09:

Now dimension the part that was created for Extracted Drawings Application.

(Hint: Use Notes for Symmetry and fillet radii)
In-Class Activity

Now complete the drawing started previously (SW_06) by adding all necessary dimensions and notes. (Hint: Notice Symmetry)
Important Takeaways

- Dimensioning is used to define an object, including the overall size with 3 dimensions and the location and size of part features.
- There are important rules for common features of parts including circular features.
What’s Next?

- Due Next Class: SW09 Applications