Dimensioning part 2 with SolidWorks

ENGR 1182
SolidWorks 09
Today’s Objectives

- Review Basic Rules of Dimensioning
- Extend Basic Rules for more Clarity
- SW09 In-Class Activity
- SW09 Out-of-Class Homework Assignment
**Formal Drawings**

Definition: Detailed multi-view representations of a finished part

**Formal Drawing Components:**
- Dimensioning
  - Size and Type of Features
  - Good Practices
Review of Basic Dimensioning

Dimensioning is used to define an object so that it could be manufactured and must:

- Define the overall size of the part in all 3 dimensions.
- Define the size and location of the features of the part in all 3 dimensions.
Review of Basic Dimensioning

1. Overall Size
2. Features Size and Location
Rule 1: Appropriate View

Each dimension should be placed in a descriptive or characteristic view without dimensioning to hidden lines.
Rule 2: Clarity

Dimensions should be located outside the boundaries of the object whenever possible and should be spaced far enough apart to be read clearly.
Rule 3: Location of Dimensions

Dimensions that are shared between two views should be located between those views.
Rule 4: Cylinders

- 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.
- A diameter symbol is always required before a numerical value.
Rule 5: Amount of Dimensions

Do not over-dimension or under-dimension an object
Solid Modeling Dimensioning

Hole and Arc Details

- 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: Adding Dimensions

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

Notes can be added 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

Homework Assignment

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

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

Now complete the drawing started previously 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 Out of Class HW
- Before next class, you will read about Implementing Section Views, Working Drawings and Rapid Prototyping.