Dimensioning part 1 with SolidWorks

ENGR 1182
SolidWorks 08
Today’s Objectives

- Formal Drawing Components:
  - Dimensioning with SolidWorks

- SW08 In-Class Activity

- SW08 Out-of-Class Homework Assignment
Formal Drawings

Definition: Detailed multi-view representations of a finished part

Formal Drawing Components:

1. Extracted Drawings
   - Extracted Views
   - Detailed Features
   - Title Block

2. Dimensions
   - Size and Type of Features
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.
Basic Dimensioning

Overall Size

Features Size and Location

.75
2
2
.75
2
.25
1
2
Basic Dimensioning: Arcs and Circles

Overall Size

Features Size and Location

R1

Ø1
Baseline Vs. Chain Dimensioning

- There are many ways to use dimensions to **locate features**.
  - Chain
  - Baseline

(Both techniques are acceptable, however baseline dimensions are preferred.)
SolidWorks: Dimensional Standards

- There are multiple dimensioning standards used in manufacturing and technical drawings.
  - ANSI – EEIC preferred Standard

Open the SolidWorks options menu
SolidWorks: Dimensional Standards

- ANSI – EEIC preferred Standard

- Under Document Properties
  - ANSI is listed as the Overall drafting standard but usually needs reset
  - Reset System:
    - Switch to ISO, then hit OK
    - Then go back and switch it back to ANSI
SolidWorks: Adding Dimensions

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

Note that sometimes the Isometric needs to be re-scaled to allow adequate drawing space which requires a NOTE.

Notes can be added to reduce the number of dimensions by stating symmetry or identifying fillet radii.
Dimensioning Wrap Up

**Basic Rules of Dimensioning**

1. Overall size in all 3 dimensions
2. Size and location of all features in all 3 dimensions

**Homework Assignment SW08-OUT:** Model part and create 2D Drawings with all necessary formatting (dimension in next class)
SolidWorks: Adding Dimensions
(for In_Class Activity)

Dimensions can be added using the “Smart Dimension” feature under “Annotation”
In-Class Activity

Open the following document entitled “Dimensioning Part1 – InClass Activity.slddrw” located in the Zip File under “In_Class Activity” on the EEIC website and add dimensions to the following 4 shapes. UnZip the folder using 7-Zip Extract Here
Important Takeaways

- Extracted drawings are used to show 3D parts as 2D drawings.
- Titles blocks are used for identification and informative purposes.
- Dimensioning is used to define an object, including the overall size with 3 dimensions and the location and size of part features.
What’s Next?

- Due Next Class: SW08 Out of Class HW
- Before next class, you will read about dimensioning in detail.
- Take SolidWorks 9 Quiz on readings