Dimensioning part 1 with SolidWorks

ENGR 1182
SolidWorks 08
Today’s Objectives

- Formal Drawing Components:
  - Dimensioning with SolidWorks
- SW08 Activity
- SW08 Application
**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.
Review Basic Dimensioning

Overall Size

Features Size and Location
Basic Dimensioning: Arcs and Circles

Overall Size

Features Size and Location

OBJECT IS SYMMETRICAL TOP TO BOTTOM

WITHOUT THE SYMMETRY NOTE PRESENT, THIS DIMENSION WOULD BE REQUIRED
When dimensioning objects with Center Marks it is usually advantageous to establish the Center Marks location early in the dimensioning process.

Note that by dimensioning the Center Mark offset early, it becomes readily apparent that by using it, in combination with the radius (R1), the width of the object is fully specified.
Review 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.)
Review Dimensioning Guidelines

10 BASIC DIMENSIONING GUIDELINES *

1. Do not over-dimension or under-dimension an object. Every object must be completely and uniquely described, so that any distance or angle related to the object can be found, but only in one way.

2. Each dimension should be placed in a descriptive or characteristic view where the corresponding component may be seen clearly.

3. Dimensions should be located outside the boundaries of the object and placed between views whenever possible.

4. Dimension lines should be aligned and grouped where possible to promote clarity and uniform appearance.

5. Do not cross dimension lines with extension lines or other dimension lines (note that extension lines may cross other extension lines).

6. When symmetry is used to reduce dimensioning complexity, a note must be added describing precisely the location of the axis of symmetry.

7. Avoid dimensioning to a hidden line.

8. Hole and Arc Details:
   A. Diameters (for circular features of 360 degrees) are dimensioned with a numerical value preceded by the diameter symbol (Ø) e.g. Ø5.0.
   B. Radii (for circular features of less than 360 degrees) are dimensioned with a numerical value preceded by the radius symbol (R) e.g. R5.0.
   C. A small cross is always used to locate the center of a circle. It is also used when the center of curvature for an arc needs to be located.

9. Dimensioning Cylinders:
   A. Dimension a positive cylinder (e.g. a rod) in a view perpendicular to where it appears as a circle, i.e. where the side of the cylinder appears as a rectangle. The symbol Ø is still required.
   B. Dimension a negative cylinder (e.g. a hole) in the view where it appears as a circle.

10. There should be a visible gap between the extension line and the feature being dimensioned. (Note that if the feature is interior to the object, there is no break in the extension line where it crosses the object boundary.)

* OVERALL CLARITY is the ultimate goal. However, if you break any of the above guidelines, the results should be distinctly clearer than what you could achieve without breaking the guidelines.
SolidWorks: Setup Dimensioning Standards

- There are multiple dimensioning standards used in manufacturing and technical drawings.
- ANSI – EED preferred Standard
- Go to Options Icon and click on Options

Open the SolidWorks options menu
SolidWorks: Setup Dimensioning Standards

- ANSI – EED preferred Standard

- Under Document Properties
  - ANSI Modified listed as the Overall drafting standard but this usually often to be reset
  - Reset System:
    - Switch to ISO, then hit OK
    - Repeat the sequence and switch it back to ANSI Modified
Dimensioning in SolidWorks

- Recall the process we used for Hand Drawings
- In SolidWorks we need to add 3 more steps to the process

After creating the Extracted Drawing, Complete the Title Block

Check for any missing Center Marks and Center Lines

Check for Symmetry and add an appropriate Note

Add Dimensions to: Establish Overall Size

Add Dimensions to: Features Size and Location
- We applied a **limited set** of the Guidelines earlier to simple Hand Drawings.

- Now let's look at applying **many of the Guidelines** to a more complex SolidWorks part.
Extracted SolidWorks Drawing with completed Title Block

Guideline 2 - Characteristic View
Guideline 3 - DIM Between Views

Check for Symmetry and add note

Add any missing Center Marks (none) and Center Lines

DIMENSION HEIGHT - STEP 1

OBJECT IS SYMMETRICAL LEFT TO RIGHT AND FRONT TO BACK
Guideline 2 - Characteristic View
Guideline 3 - DIM Between Views
Guideline 8 - Radi (R)
Guideline 9 - Positive Cylinder where appears as a Rectangle
Guideline 2 - Characteristic View
Guideline 3 – DIM Between Views
Without the SYMMETRY note many more dimensions would be required in the Right Side View. We know the corresponding element on the right also has a depth of 2.0. Since the cylinder has a diameter of 3.0, then the total “gap” is equal to 

\[10 - 3.0 - 2*2.0 = 3.0\]

and each individual gap is equal to 

\[3.0 / 2 = 1.5\].

Also the location of the center of the cylinder would have required 2 dimensions in the Top View.

So what appeared to be fairly complicated is now complete!
SolidWorks: Adding Dimensions

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

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 multiple fillet radii.
SolidWorks Dimensioning Wrap Up

**Basic Rules of Dimensioning**

1. Complete Title Block
2. Check for Missing Center Marks and Center Lines
3. Check for Symmetry and add Note
4. Overall size in all 3 dimensions
5. Size and location of all features in all 3 dimensions

**ACTIVITY SW08:**
Model part and create 2D Drawings with all necessary formatting (dimension in next class)
SolidWorks: Adding Dimensions
(Activity)

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

A SYMMETRY note can be added using A Note under “Annotation.”
Open the SolidWorks file associated with **Activity** located in the Zip File under in Carmen 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.
- Some Center Marks and Center Lines will need to be added.
- A Symmetry note may need to be added.
- 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 Application
- Before next class, you will read about Dimensioning in detail, Implementing Section Views and Working Drawings in SolidWorks
- Take SolidWorks 9 Quiz on readings