Dimensioning with SolidWorks

Part 2

Alternate Instructor’s PowerPoint modified by R. Busick

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Engineering Education Innovation Center
Formal Drawings

Definition: Detailed multi-view representations of a finished part

Formal Drawing Components:
1. Dimensions Part 2
   • Size and Type of Features
   • Good Practices
Review 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

Dimension Line
Extension Line

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 centermark 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
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.
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 that are shared between views should be placed between those views. Generally all dimensions should be placed between views.

4. **Dimensioning Cylinders:**
   - 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 O is still required.
   - Dimension a **negative cylinder** (e.g. a hole) in the view where it appears as a **circle**.

5. **Hole and Arc Details:**
   - Diameters (for circular features of 360 degrees) are dimensioned with a numerical value preceded by the **diameter symbol** (Ø) e.g. Ø5.0.
   - 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.
   - 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.

6. Dimensions should be located **outside the boundaries** of the object whenever possible.

7. **Avoid dimensioning to a hidden line.**

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

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

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

11. 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.)

12. Center lines and center marks should not be placed on Isometric views.

* **OVERALL CLARITY is the ultimate goal.** However, if you do not follow any of the above guidelines, the results should be distinctly clearer than what you could achieve without violating the guidelines.

It is strongly recommended that you make a copy of the website word document and use it as a reference when you perform dimensioning.
Review-Order of Dimensioning

(After you have oriented the object so that the dominant FEATURES appear in the ORTHOGRAPHIC FRONT VIEW)

1. Insert all center lines and center marks
   (then they will be available for proper dimensioning!)

2. Set dimension units (inch/mm)

3. Dimension basic outside dimensions
   (height, width and depth)

4. Locate all the object’s features

5. Size all the object’s features
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)

NOTE: Per the homework instructions, do NOT implement the R4 fillets since then you will be unable to properly dimension the drawing.
Now complete the drawing started previously by adding all necessary dimensions and notes. (Hint: Notice Symmetry)
The Following 3 slides show how to:

1. Control Dimension Precision (how many decimal places)

2. Establish Proper Dimensioning Styles (ANSI)

3. Eliminate Tangent Edges

These slides should be shown in SLIDE SHOW MODE
Control of Dimension Precision in Drawings

(How many decimal places)

Set appropriate number of decimal places here.
Modifying the Drawing Template for Proper Dimensioning

If after you bring up the OSU template and try to dimension a hole you find the following incorrect format:

To correct it, go to Tools/Options/Document Properties and even if you find ANSI (which is the desired format), open the menu using the down arrow and select anything else (like ISO) and click OK.

Then return to Tools/Options/Document Properties and select ANSI and click OK. When you dimension the hole you find the correct format.
Eliminating Tangent Edges

If you find that your drawing is showing tangent edges

Right click on the tangent edge, hover over Tangent Edge in the dropdown box and then click Tangent Edges Removed to eliminate this occurrence.

To achieve this result

To eliminate future tangent edges, navigate to Tools/Options/System Options/Display Style and under Tangent edges in new views click on "Removed" and then "OK".