Dimensioning Precision and Tolerancing Strategies

IET 265, Solid Modeling using SolidWorks, by Chris Scarlett

Precision:

- The degree of accuracy necessary to ensure the intended function.
- Precision in mechanical drawings typically refers to the significant digits in the dimension used for measuring. The more digits used to describe a dimension the more precise it may be.
- In SolidWorks it also refers to the digits to the right of the decimal. This will have the same definition as the previous one involving significant digits, if the largest value digit is a thousandth of a unit.
    - Precision is important when regarding **mating surfaces** in machined parts, such as a pin inserted into a slot. It is important that the precision of the dimension referencing some common geometry between the two mating surfaces (such as an axis or cylindrical surface) is such that a specific fit can be obtained on a consistent basis at a reasonable cost.
    - Precision is less important with **Non-mating surfaces**. These surfaces will be exposed to the ambient environment and will have minimal precision requirements.

Tolerance:

- The allowable variation for any given size and provides a means of controlling the precision required.
    - For **Mating surfaces** on small parts, tolerances may be as small as a millionth of an inch (0.000001) or around 6 digits after the decimal. For practical reasons tolerances on typical parts range between 0.01 and 0.001 of an inch and precision of 3 to 4 significant digits.
    - For **Non-mating surfaces** on small parts tolerances vary around 0.01 inches.
    - On large parts the decimal moves to the right but the significant digits remain roughly the same.
    - Examples of allowable tolerances for the for various part sizes follow:

English	Metric	Typical Tolerance		Significant Digits
0 – 6”	0-150mm	+ / - 1/64” or 0.015”	0.4mm	3 to 4
6 – 18”	150-500mm (0.5 meter)	+ / - 1/32” or 0.03”	0.8mm	3 to 4
18” to 36’	0.5 meter to a meter	+ / - 1/16 or 0.06”	1.5mm	3 to 4

Dimensions and Drawings:

- Dimension extension lines should not touch or “cover” the outline of the object that you model. Use strategies introduced in class to eliminate this.
- Leaders: Text leaders should be drawn at an angle other than the horizontal or vertical such as 30, 45 or 60 degrees.
- Carefully choose the size and location of dimensions to make them readable and easily understood.
- 2 different dimension types follow:
    - Size type dimensions show the size of something such as the diameter of a hole or length of amn edge.
    - Location type dimensions show the location of an entity in regard to some reference geometry such as ordinate or polar dimensions.
- Redundant Dimensions: These are duplicate dimensions on your drawings and may refer to the same object in a different view. They should be avoided as they may cause update errors and clutter up your drawing.
- Centerlines: Are used to show axial symmetry and frequently reduce the need for location dimensions.
    - Centerlines should be visible on the drawing as the appropriate line type and size of a series of short and long dashes.
- Hidden Lines: Are used to show the location of some geometry (edge or hole) behind a solid surface.
    - Hidden lines should be visible on the drawing as the appropriate line type of short dashes.

Helpful Hints:

- Create different layers for your different line types; center, hidden and dimensions.
- Never cover another drawn object with a dimension extension line.
- Avoid the crossing of two dimension lines if possible.
- Avoid dimensioning hidden lines unless there is no other reasonable way to display this information.
- Bring your centerlines out slightly and symmetrically from the edge of your drawn object, they should never terminate on a model edge.
- Bring out or extend your section arrows so they do not touch your model.