Design Dimensioning and Tolerancing
J. M. McCarthy
Fall 2003

- Overview of dimensions
- Dimensioning various features
- Limits of size
- Classes of fits
- Summary
This standard establishes uniform practices for defining and interpreting dimensions, and tolerances, and related requirements for use on engineering drawings.

The figures in this presentation are taken from Bruce Wilson’s *Design Dimensioning and Tolerancing*. 
An engineering drawing is not an illustration. It is a specification of the size and shape of a part or assembly. The important information on a drawing is the dimension and tolerance of all of its features.
Dimensioning Guidelines

The term “feature” refers to surfaces, faces, holes, slots, corners, bends, arcs and fillets that add up to form an engineering part.

Dimensions define the size of a feature or its location relative to other features or a frame of reference, called a datum.

The basic rules of dimensioning are:
1. Dimension where the feature contour is shown;
2. Place dimensions between the views;
3. Dimension off the views;
4. Dimension mating features for assembly;
5. Do not dimension to hidden lines;
6. Stagger dimensioning values;
7. Create a logical arrangement of dimensions;
8. Consider fabrication processes and capabilities;
9. Consider inspection processes and capabilities.
## Symbols Used in Dimensioning

<table>
<thead>
<tr>
<th>CURRENT PRACTICE</th>
<th>ABBREVIATION IN NOTES</th>
<th>PARAMETER</th>
</tr>
</thead>
<tbody>
<tr>
<td>ø</td>
<td>DIA</td>
<td>DIAMETER</td>
</tr>
<tr>
<td>Sø</td>
<td>SPHER DIA</td>
<td>SPHERICAL DIAMETER</td>
</tr>
<tr>
<td>R</td>
<td>R</td>
<td>RADIUS</td>
</tr>
<tr>
<td>CR</td>
<td>CR</td>
<td>CONTROLLED RADIUS</td>
</tr>
<tr>
<td>SR</td>
<td>SR</td>
<td>SPHERICAL RADIUS</td>
</tr>
<tr>
<td>CBORE or SFACE</td>
<td>CSK</td>
<td>COUNTERBORE</td>
</tr>
<tr>
<td></td>
<td>DP</td>
<td>SPOTFACE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>COUNTERSINK</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DEEP</td>
</tr>
<tr>
<td></td>
<td></td>
<td>DIMENSION ORIGIN</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SQUARE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>REFERENCE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>PLACES, TIMES</td>
</tr>
<tr>
<td></td>
<td></td>
<td>ARC LENGTH</td>
</tr>
<tr>
<td></td>
<td></td>
<td>SLOPE</td>
</tr>
<tr>
<td></td>
<td></td>
<td>CONICAL TAPER</td>
</tr>
</tbody>
</table>

---

[195 EPD engineering project development]
Layout of Dimensions

Dimension feature contours

Figure 3-24. Features are most clearly dimensioned in the view that shows the feature contour.

Place dimensions between views

Figure 3-25. Dimensions are placed between views when possible.
Arrangement of Dimensions

- Keep dimension off of the part where possible.
- Arrange extension lines so the larger dimensions are outside of the smaller dimensions.
- Stagger the dimension value labels to ensure they are clearly defined.

![Diagram]

Figure 3-26. Dimensions are normally placed off the object. Avoid placing dimensions on the object.

Figure 3-29. Staggered positions for dimension values make it easier to read the dimensions.
Dimensioning Holes

- Dimension the diameter of a hole.
- Locate the center-line.
- Use a notes and designators for repeated hole sizes.
Dimensioning the Radius of an Arc

Dimension an arcs by its radius.
Locate the center of the radius or two tangents to the arc.
Drilled Holes, Counterbores and Countersinks

- Use the depth symbol to define the depth of a drilled hole.
- Use the depth symbol or a section view to dimension a counterbore.
- Countersinks do not need a section view.
Angles, Chamfers and Tapers

- Dimension the one vertex for an angled face, the other vertex is determined by an intersection.
- Chamfers are generally 45° with the width of the face specified.
Rounded Bars and Slots

- The rounded end of a bar or slot has a radius that is 1/2 its width.
- Use $R$ to denote this radius, do not dimension it twice.
- Locate the center of the arc, or the center of the slot.

Figure 4-25: The overall length of round-ended bars is dimensioned.

Figure 4-26: Slotted holes are used to allow greater tolerance for the location of fastener holes and to permit adjustment.
Limits of Size

- All dimensions have minimum and maximum values specified by the tolerance block.
- Tolerances accumulate in a chain of dimensions.
- Accumulation can be avoided by using a single baseline.
Fit Between Parts

- Clearance fit: The shaft maximum diameter is smaller than the hole minimum diameter.
- Interference fit: The shaft minimum diameter is larger than the hole maximum diameter.
- Transition fit: The shaft maximum diameter and hole minimum have an interference fit, while the shaft minimum diameter and hole maximum diameter have a clearance fit.

**Clearance Fit**

**Interference Fit**

**Transition Fit**

Figure 3-42. Proper calculation of a clearance fit will result in limits of size that provide clearance between the features for all possible size continuations.

Figure 3-43. A negative allowance value between two features indicates that an interference fit exists.

Figure 3-44. A transition fit results in a clearance fit at one extreme of the applied tolerance limits, and an interference fit at the other extreme.
Classes of Fit

The limits to sizes for various types of fit of mating parts are defined by the standard ANSI B4.1

There are five basic classes of fit:
1. Running and sliding clearance (RC)--there are type of RC fits, RC1-RC9;
2. Location clearance (LC)--there are eleven types of LC fits;
3. Location transition (LT)--there are six types of LT fits;
4. Location interference (LN)--there are three LN fits;
5. Force fits (FN)--there are five FN fits.
Summary

- The most important information on an engineering drawing are the dimensions.
- Dimensions specify the size and location of features that make the part useful.
- All dimensions have tolerances defined either explicitly or in a tolerance block.
- The relative size ranges of mating parts defines whether they fit with or without allowable relative movement (clearance or interference).