Basic Dimensioning and Tolerancing

Dr. Hodge Jenkins
MAE 205
Important Elements of Dimensioning
Dimension locations

A  
B  .500  
C  .375  
D  .125

E   F   G   H
.500 .500 .375 .125

2.000

Dimensions and all text are ALWAYS horizontal
Size and Location of Features

Dimension features in the view best seen.
Slotted holes may be dimensioned in several ways.
Dimension Line Spacing

**Figure 4.9 Minimum Dimension Line Spacing**
Standard practice for the spacing of dimensions is 10 mm from the view and 6 mm between dimension lines.
Dimension Locations

NO

Dimension out part boundary

YES

Dimensions and all text are ALWAYS horizontal

NO

Unidirectional Current standard

YES

Aligned Old standard

NO
Grouping of Dimensions

Figure 4.10  Group Dimensions
In standard practice, dimensions are grouped on a drawing. Do not use object lines as extension lines for a dimension.
Grouped dimension

Figure 4.11  Stagger Dimension Text
The general practice is to stagger the dimension text on parallel dimensions.

Figure 4.12  Extension Line Practice
Extension lines should not cross dimension lines, are not broken when crossing object or other extension lines, and are broken when crossing arrows.
Diameter Features

Figure 4.17 Radial Leader Lines
Leader lines used to dimension holes must be radial.
Diameter Features

Figure 4.21 Dimensioning Concentric Circles
Concentric circles are dimensioned in the longitudinal view.
Dimensioning Slots

Several methods are appropriate for dimensioning slots.
Figure 4.22 Dimensioning Arcs
Arcs of less than half a circle are dimensioned as radii, with the R symbol preceding the dimension value.

Figure 4.23 Avoid Overdimensioning
Double dimensioning can cause problems because of tolerancing.

Figure 4.24 Dimension the Most Descriptive View
Dimensions are placed in the most descriptive or contour view.
4.5.4 Single Limit Dimensions

When other elements of a feature will determine one limit dimension, MIN or MAX is placed after the other limit dimension. Items such as depth of holes, length of
Figure 4.29 Toleranced Parts and the Important Terms
Figure 4.30 Clearance and Interference Fits between Two Shafts and a Hole
Shaft A is a clearance fit, and shaft B is an interference fit.
Figure 4.25
A system is two or more mating parts.

Figure 4.26 Representing Tolerance Values
Tolerances are represented as direct limits or as tolerance values.

Figure 4.27 Geometric Tolerance System Used to Dimension Parts
Threads and threaded fasteners

**Figure 4.36** Standard Thread Note for English Dimension Fasteners
4.37 Standard Thread Note for Specifying Tap Drill Size