



# Engineering Graphics

## Dimensioning Rules

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### Dimensioning Rules

- § Dimensions should **always** be placed in the profile view.
- § **Do not** duplicate dimensions in other views.
- § Holes should always be located and called out in the circular view.
- § Cylindrical shapes should be dimensioned in the rectangular view.
- § You should **never** dimension to hidden lines or features.
- § You should also **never** cross a dimension line with any geometry.
- § **You may** however cross extension lines with other extension lines or with leaders for diameter/radius dimensions.
- § When placing leaders for diameter/radius dimensions or hole callouts, do not make the leader line horizontal or vertical. You should try to shift leaders at least 15° from vertical/horizontal.
- § **AutoCAD Only>>** All dimensions **must be associative**, and large point deductions will be made for ignoring this rule. Under no circumstances may you override the text value of a dimension except for reference dimensions. If a prefix or suffix is needed, do so via the **Dimension Style Manager**, using the options within the **Text** tab, or use the text editor and add annotation in front or after the <>. Some examples are:
  - %%C<>        puts a diameter symbol (∅) in front i.e. ∅.750
  - <>%%D        puts a degree symbol (°) after i.e. 45°
  - <>TYP        for a typical dimension i.e. 1.250TYP
  - (<>)         reference dimension with value in parenthesis i.e. (.25)
  - 2x<>        2 places i.e. 2xR.125
- § All dimensions in your drawings must reflect the **same number of decimal places** as the drawing problem given, including angular dimensions. There will usually be a varied number of decimal places in a single drawing. **AutoCAD Only>>** Use **Dimension Style Manager, Override, Primary Units, Precision**, to change decimal places, or right click after gripping a dimension(s) and change precision. Point deductions will be made for dimensions with the wrong number of decimal places.
- § Dimension **spacing should be .5 inches or 12 mm apart**. Use a grid to judge distances. Deductions will be made for crowded or inconsistent dimension spacing. If you need to expand the area of the grid, use the **Limits** command.
- § The empty space between dimensioned views should be **approximately 1.00” or 25 mm**. Move your views after dimensioning, and make sure that Ortho or Polar is on to maintain view alignment.
- § Tolerance and limit dimensions must be inserted using dimension variables, do not use the text command or suffixes and prefixes to add the extra values. **AutoCAD Only>>** Use **Dimension Style Manager, Override, Tolerances, Method**, to represent tolerances, or use **Modify Properties** to add the tolerance information after inserting the dimension.
- § Be aware that inch drawings typically have no leading zeros (before the decimal place), and metric drawings always do, when distances fall below 1 mm. **AutoCAD Only>>** Use **Dimension Style Manager, Modify, Primary Units, Zero Suppression**, if you need to suppress zeros.