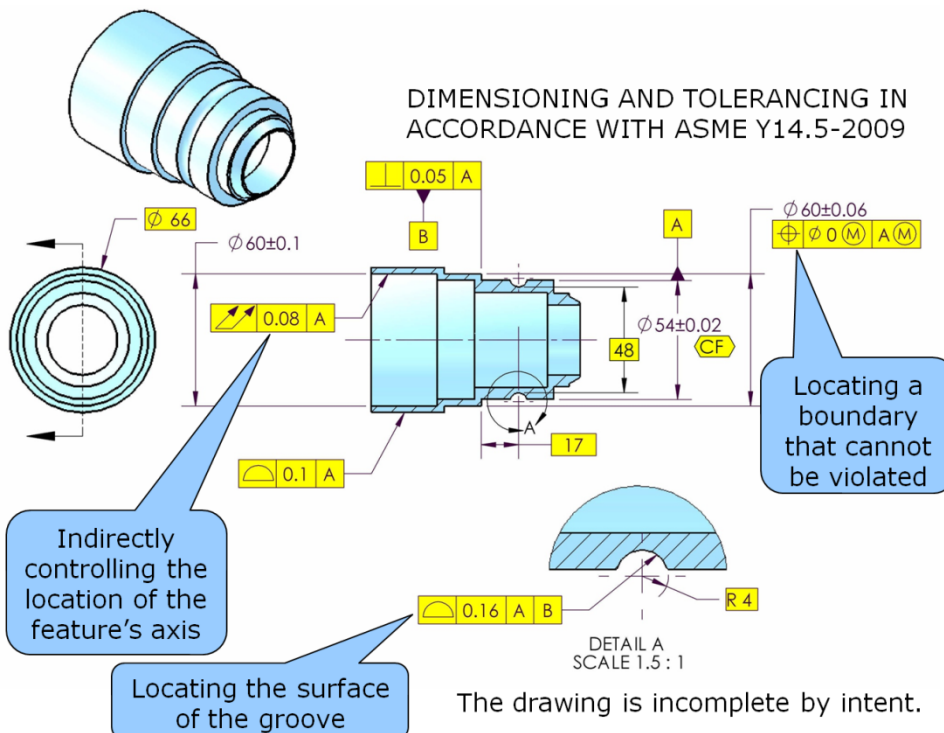# January 2012
Tip-of-the-Month
Stop Locating Features with
Plus/Minus Toleranced Dimensions!

    I am constantly being asked questions regarding the tolerancing applied to drawings. Recently many of these drawings use plus/minus toleranced dimensions to locate features. There have been so many of these drawings lately that my head is about to explode. These directly toleranced dimensions are great for controlling the size of a feature. We have Rule #1 that explains how that works. However, other than the little used dimension origin symbol, **there is no support in the Y14.5 standard for locating features with dimensions that use plus/minus tolerances**. Features are located using BASIC dimensions. Other than some datum features, all features on a part must be controlled for location. The geometric tolerances that control location of features are shown in the table.

Feature Being Located	Appropriate Geometric Tolerance
Any surface	⌒
The center of a feature of size	⊕
One boundary of a feature of size	⊕ modified with Ⓜ or Ⓛ
A cylinder coaxial to a datum axis	⊕, ↗, ⚡ or ⌒

    Symmetry and concentricity may also locate but are of such little value that many companies have outlawed their use.

The drawing drives the entire process. When the drawing is unclear it affects purchasing, tooling, quality plans, assembly, inspection software, tolerance analysis software, etc. The Y14.5 standard has been proven to have only one meaning. To continue to ignore this Standard is irresponsible.

This drawing shows the three main ways to tolerance the location of features.

The drawing is incomplete by intent.

Please email us any suggestions or topics that you would like to see covered in our
Tip-of-the-Month series.
www.tec-ease.com