

## 1. What is GD&T?

Geometric Dimensioning & Tolerancing (GD&T) is a symbolic language for researching, refining, and encoding the function of each feature of a part. In addition to enabling unambiguous decoding to communicate design intent to manufacturing and quality assurance, GD&T enables scientific tolerance stack-up analysis, and is therefore in a position to absolutely guarantee the assemblability of in-tolerance mating parts. It consists of concepts, tools, rules, and processes, which are described in various military, national and ISO standards, and are set forth in this document in abbreviated form. This pocket guide is based on the ASME Y14.5M 1994 Standard, plus certain *SmartGD&T* enhancements noted as such.

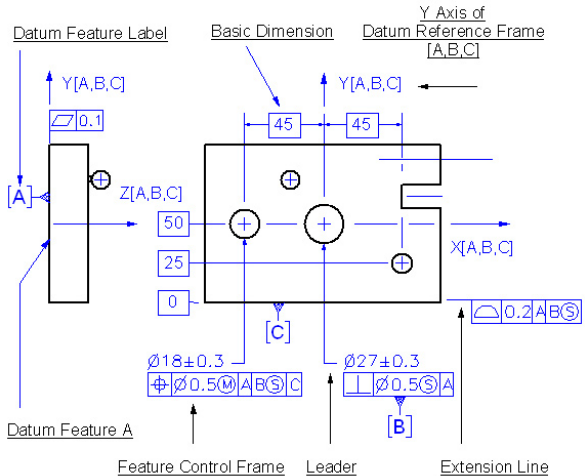


Figure 1/1 A GD&T Encoded Drawing

The symbolic language of GD&T creates a perfect imaginary world consisting of Tolerance Zones, Tolerance Values, Datums, Coordinate Systems and Basic Dimensions, which impose limits on the imperfect form, size, orientation and location of real world, machine part Features (e.g. planar surfaces, bores and shafts) and their Components (surface points, axes, mid-planes). Figure 1/1 above illustrates a partially GD&T encoded object, Figure 1/2 below illustrates the resulting collection of Tolerance Zones and the controlling Datum Reference Frame (DRF) whose axes are labeled

X[A,B,C], etc.. (Note: These DRF axis labels are *SmartGD&T* enhancements.)

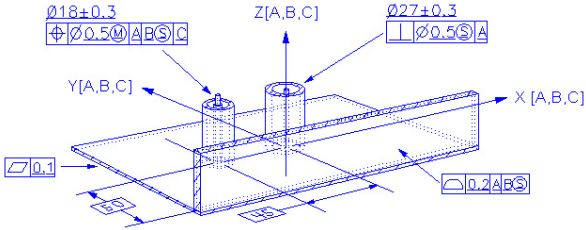


Figure 1/2 GD&T defined Tolerance Zones

All points on the controlled surface of the feature, or the entire bounded component of the feature, must lie within the specified Tolerance Zone.