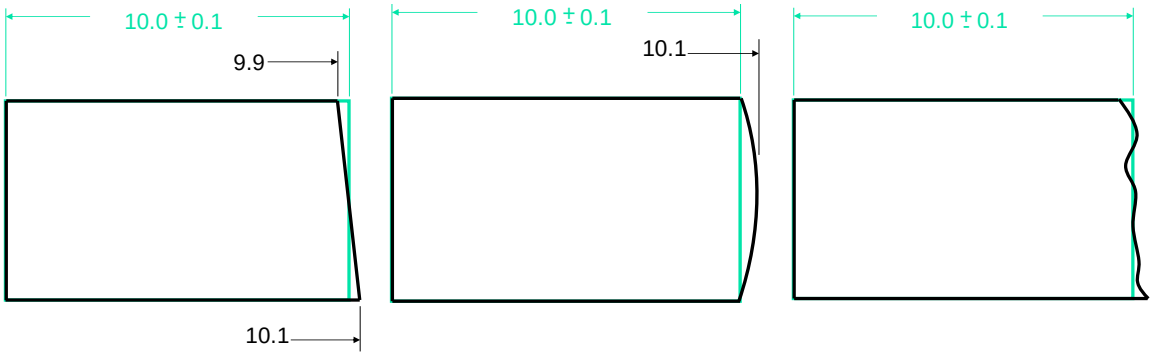
# Geometric Dimensioning and Tolerancing (GD&T)

## Background

- Limit Tolerancing, which is done by having maximum and minimum limits for dimensions, is likely what you have been using and is one of the more commonly known tolerancing, but has major fallbacks.
- Some issues that Limit Tolerancing faces is that it allows for a high amount of variation within its limits. Things such as straightness, perpendicularity and concentricity are all things that aren't specified nor controlled when using Limit Tolerancing which can lead to ambiguity in the design.
- Geometric Dimensioning and Tolerancing (GD&T) is a system of standardized drawing symbols that is used to reduce the uncertainty and obscurity of the desired geometric shape.
- The GD&T system is used internationally and is language universal, as it consists of standard symbols approved by the International Organization of Standardization (ISO) and agreed upon internationally.
- The following depict some variation in the geometry of a feature that can be seen in limit tolerancing, which GD&T attempts to control and lessen.




## When to Use GD&T?

- GD&T does not need to be used for every dimension in the drawing. It should mainly be used for critical features, such as for:
    1. Features that are critical to the function
    2. Features that come into contact with other parts or surfaces, especially when close tolerances are used for the concerned features
- Other reasons GD&T should also be used are when:
    1. Functional gaging techniques are desirable
    2. Datum references are required for consistency
    3. It allows for better machining process choices during production

## Using GD&T

- Geometric tolerances are done by using symbols and numbers placed into rectangular frames called *feature control frames*.
- When doing geometric tolerancing, state the following details into the compartments of the *feature control frame*.
    1. The geometric symbol which defines what characteristic of the feature is being toleranced. A table of these symbols and their use is featured below.
    2. The Tolerance Value (The difference between the maximum and minimum limits).
        - Preceded by:
            i. Φ if it is circular or cylindrical
            ii. SΦ if it is spherical
        - Followed by GD&T symbols for conditions shown in Fig …. If applicable
    3. The datum letter(s) that the geometric references. (If there are any)
        - Note: Each letter gets a separate compartment.

## Form Control

- These are geometric features that control the form of the feature, and do not require any datum references.

| Type of tolerance | Characteristics to be toleranced | Symbol | Datum needed | Applications |
|---|---|---|---|---|
| Form | Straightness |  | No | A straight line. The edge or axis of a feature. |
| | Flatness |  | No | A plane surface. |
| | Roundness |  | No | The periphery of a circle.Cross-section of a bore, cylinder, cone or sphere. |
| | Cylindricity |  | No | The combination of circularity, straightness and parallelism of cylindrical surfaces. Mating bores and plungers. |
| | Profile of a line |  | No | The profile of a straight or irregular line. |
| | Profile of a surface |  | No | The profile of a straight or irregular surface. |

## Orientation Control

| Type of tolerance | Characteristics to be toleranced | Symbol | Datum needed | Applications |
|---|---|---|---|---|
| Orientation | Parallelism |  | Yes | Parallelism of a feature related to a datum. Can control flatness when related to a datum. |
| | Perpendicularity |  | Yes | Surfaces, axes, or lines positioned at right angles to each other. |
| | Angularity |  | Yes | The angular displacement of surfaces, axes, or lines from a datum. |
| | Profile of a line |  | Yes | The profile of a straight or irregular line positioned by theoretical exact dimensions with respect to datum plane(s). |
| | Profile of a surface |  | Yes | The profile of a straight or irregular surface positioned by theoretical exact dimensions with respect to datum plane(s). |

## Location Control

| Type of tolerance | Characteristics to be toleranced | Symbol | Datum needed | Applications |
|---|---|---|---|---|
| Location | Position |  | See note below | The deviation of a feature from a true position. |
| | Concentricity and coaxiality |  | Yes | The relationship between two circles having a common centre or two cylinders having a common axis. |
| | Symmetry |  | Yes | The symmetrical position of a feature related to a datum. |
| | Profile of a line |  | Yes | The profile of a straight or irregular line positioned by theoretical exact dimensions with respect to datum plane(s). |
| | Profile of a surface |  | Yes | The profile of a straight or irregular surface positioned by theoretical exact dimensions with respect to datum plane(s). |

http://www.cnccookbook.com/GD&T/GD&TTutorialHome.html

PPT]Geometric Dimensioning & Tolerancing - UF MAE

*www2.mae.ufl.edu/designlab/Online%20Resources_files/GDT%20tutorial.ppt*

*GD&T* is a means of dimensioning & tolerancing a drawing which considers the function of the ... For *example*, if large number of parts are being made – *GD&T* can reduce or eliminate inspection of some features. .... Datums on a drawing of a part are represented *using* the symbol shown below. ... UNRESTRICTED *FREE*.