

# Geometrical Dimensioning and Tolerancing

## ASME Y14.5-2009 baseline

This training is for anyone who comes into contact with geometrical tolerancing, both its application and interpretation. We often see a great deal of diversity in the application and interpretation of geometrical tolerancing. This training session strives constructively towards uniform interpretation, in order to get all of the participants on the same page.

### Training objective

The main objective of the training session is to get everyone on the same page with regard to the application and interpretation of geometrical tolerancing. This training session includes time for class discussions. This fine-tunes communication and helps get everyone achieve a common understanding.

By the end of the two-day training session, participants will:

- Be able to interpret indicated geometrical tolerancing correctly
- Be able to apply geometrical tolerancing functionally themselves in the formulation of product specifications and preparation of drawings
- Have an understanding of the differences between the ASME and ISO standards

### Target group

The training is designed for a mixed target group. Anyone who comes into contact with geometrical tolerancing (technical drafters, planners, etc.) is encouraged to attend.

### Training materials

- Manual in English: "Fundamentals of GD&T ASME Y14.5-2009", 419 pages.
- English: "The Ultimate GD&T Pocket Guide ASME Y14.5-2009", 121 pages.
- Syllabus: 221 pages, in English.

### General

- Maximum of 8 participants per session, in order to promote class discussion and interaction.
- The client will provide a projector and whiteboard.
- The training can be given in either Dutch or English.
- After completion, each participant will receive a certificate.

### Contents of the training

#### Introduction

- GD&T - Coordinate dimensioning, comparison
- Terminology
- GD&T modifiers and symbols
- Basic rules: Envelope principle, MMC, LMC and RFS
- Basic dimensions, Virtual condition and Bonus tolerance

#### Datum reference

- The datum system, Planar and Datum targets
- Datum reference frame and sequences
- Feature of size datum specifications, datum shift

#### GD&T symbols

- Flatness, Straightness, Circularity and Cylindricity
- Perpendicularity, Angularity and Parallelism
- Circular runout, Total runout
- Profile of a surface, Profile of a line
- Concentricity, Symmetry Controls
- Tolerance of Position Controls
- Tolerance of Position Controls special applications
- Functional gages applications
- Projected tolerance zone, tangent plane modifier, translation modifier
- Zero tolerance at MMC, non-uniform tolerance zone
- Composite tolerancing, Multiple single segment controls
- Pattern tolerancing, customized datum reference frame

#### ASME – ISO comparison

Theoretical and Practical exercises

### Duration of the training

The entire "Geometrical dimensioning and tolerancing" training session lasts for two days, but the contents can be adapted to a particular project or product group.

### Price

Please feel free to contact me for your customised quotation.

