

Essential Geometric Dimensioning & Tolerancing

ASME Y14.5Y-2018 Standard

INTRODUCTION

Geometric dimensioning and tolerancing is an international engineering drawing system that offers a practical method for specifying 3-D design dimensions and tolerances on an engineering drawing. Based on a universally accepted graphic language, as published in national and international standards, it improves communication, product design, and quality. Therefore, geometric dimensioning and tolerancing is accepted as the language of dimensional management and must be understood by all members of the dimensional management team. Some of the advantages of using GD&T on engineering drawings and product data sheets are that it:

- Removes ambiguity by applying universally accepted symbols and syntax.
- Uses datums and datum systems to define dimensional requirements with respect to part interfaces.
- Specifies dimensions and related tolerances based on functional relationships.
- Expresses dimensional tolerance requirements using methods that decrease tolerance accumulation.
- Provides information that can be used to control tooling and assembly interfaces.

Geometric dimensioning and tolerancing provides a comprehensive system for symbolically defining the geometrical tolerance zone within which features must be contained.

It provides an accurate transmission of design specifications among the three primary users of engineering drawings; design, manufacturing and quality assurance. This language contains all relevant details involved in fabrication, including dimensions, tolerances, definitions, rules and symbols that communicate a component's functional requirements, in alignment with ASME Y14.5M-2018 Standard.

Learning Outcome

This module is taught on the basis of achieving intended learning outcomes. On successful completion of the module, the participant will be expected to be able to demonstrate the following:

Knowledge &	Definition & control of each symbols.
-------------	---------------------------------------

understanding	<i>Basic GD&T symbols and its meaning.</i>
Intellectual, practical and transferable skills	<i>The used of Datum, Feature control frame & Basic Checking method & interpretation GD&T drawings. Selection an appropriate Datum, modifier etc.</i>

TARGET AUDIENCE

This course is ideal for anyone who has a need to read or interpret geometric tolerances on a product print.

Product Engineers, Purchaser, Manufacturing Engineers, CAD Designers, Quality Engineers & Inspectors and manufacturing personnel.

PREREQUISITES

Participants should have basic engineering blueprint reading skills.

TRAINING OUTLINE

MODULE 1	Introduction to GD&T ▪ What is GD&T ▪ Why we need GD&T ▪ The 2018 version and the changes.
MODULE 2	Datum (Definition, interpretation) ▪ Symbols ▪ 3-2-1 plane ▪ Datum simulation ▪ Placement & selection ▪ Coplanar & Coaxial
MODULE 3	Basic Dimension (Definition, interpretation) ▪ Symbol ▪ Why is important

MODULE 4	Feature Control Frame (Definition, interpretation) ▪ Symbols ▪ The role of FCF ▪ Type of FCF ▪ Composite FCF Case study & Exercise
MODULE 5	Symbols (Definition, interpretation) ▪ Form ❖ Straightness ❖ Flatness ❖ Circularity ❖ Cylindricity ▪ Orientation ❖ Parallelism ❖ Angularity ❖ Perpendicularity ▪ Runout ❖ Circular Runout ❖ Total Runout ▪ Location ❖ Position ❖ Symmetry (Removed in 2018 version) ❖ Concentricity (Removed in 2018 version) ▪ Profile ❖ Line profile ❖ Surface profile ▪ Other common symbols Case Study & Exercise
MODULE 6	Material Condition Modifier (Definition, interpretation) ▪ Regardless of Feature Size (RFS) ▪ Maximum Material Condition (MMC) ▪ Least Material Condition (LMC) ▪ Virtual Condition (VC) ▪ Taylor Principle ▪ Bonus Tolerance (BT) ▪ Datum Shift (MMB) ▪ Resultant Condition (RC) ▪ Boundary Case study & Exercise

MODULE 7	Tolerancing Q&A Assessment End

DURATION

2 Days x 8 hours / day.

TRAINER'S PROFILE

Yong WK (*MBA,UK*) Please refer to stand alone profile submitted