

Geometric Dimensioning and Tolerancing (GD&T) Course Outline

 DURATION/TIME 3 Days 9:00 am - 5:00 pm	 METHODOLOGY Instructor led, presentation, exercises and discussion
 PREREQUISITES Technical or engineering background	 TARGET Application Engineer, R&D Engineer, Product Designer or Engineer, and Industrial Engineer

INTRODUCTION

Geometrical Dimensioning and Tolerancing (GD&T) provides you the knowledge in current standards and the ability to use this knowledge in your drawings or 3D models. Members will understand the difference between ASME Y14.5M and ISO 1101. At the end, members will understand their usage for production and inspection and able to apply the learned knowledge in case studies.

OBJECTIVE

At the end of this program participants are expected to:

- Understanding the difference between ASME Y14.5M and ISO 1101 and the used TOLERANCE PRINCIPLE
- Understand the DATUM REFERENCE and the FORM, POSITION, ORIENTATION and RUN notifications
- Able to interpret the MODIFIER symbol and their usage for production and inspection
- Able to APPLY the learned knowledge in CASE STUDIES

CONTACT

IME CADCAM RESOURCE CENTRE SDN BHD
Certified ISO 9001:2015 QMS

 www.training.cadcam.com.my

 training@cadcam.com.my

 +6019-659 2186



Geometric Dimensioning and Tolerancing (GD&T)

Course Outline:

Day 1: GD&T Notification

- GD&T vs. dimensional tolerances
- Tolerance principles and the difference between ISO and ASME
- Interpretation of DATUM reference system
- Basic of FORM, PROFILE, ORIENTATION, LOCATION and RUN tolerances
- Interpretation of FORM, PROFILE and ORIENTATION tolerances
- Interpretation of LOCATION and RUN tolerances

Day 2: MODIFIER and DESIGN aspect

- MODIFIER and the effect of the tolerance zone
- Calculation of MODIFIER
- Inspection methods

Day 3:

- CASE STUDIES for HIGH and LOW volume production

CONTACT

IME CADCAM RESOURCE CENTRE SDN BHD
Certified ISO 9001:2015 QMS

 www.training.cadcam.com.my

 training@cadcam.com.my

 +6019-659 2186

