



GD&T in SOLIDWORKS

Who should attend:

Application Engineer
R&D Engineer
Product Designer or Engineer
Industrial Engineer

Prerequisites:

Attended Introduction to GD&T (Theory)

Basic understanding of part design and drafting

Mechanical Design Experience

Duration:

1 Day

Methodology:

Practical hands-on with using computers, lecturing, discussions and case studies

Introduction

This GD&T in SOLIDWORKS training course is designed as a further in-depth course to help understand, interpret and apply the principles of GD&T. It is based on the ASME Y14.5 – 2009 standard. Formally the ASME Y14.5M - 1994 standard.

By definition, GD&T establishes uniform practices for stating and interpreting dimensions, tolerancing, and evaluate machined parts for assembly form & fit.

Objective

At the end of this program participants are expected to:

- Understand about the basic functions involved in a CAD system.
- Apply different dimensioning schemes such as size & location dimension, datum and geometric tolerance.
- Auto-dimensioning scheme for GD&T and check for complete definition of the manufacturing product.
- Understand the basic flow chart involved for Stack-up analysis; TolyAnalyst.

IME CADCAM TRAINING CENTRE SDN BHD

PJ: 22A, Jln. PJS 1/46, Tmn. Petaling Utama, 46000 PJ, Selangor, Malaysia.

Tel: +603-7783 6866 | Fax: +603-7781 1185

PG: 68, Jln. Perai Jaya 4, Bandar Perai Jaya, 13700 Pulau Pinang, Malaysia.

Tel: +604-399 5571 | Fax: +604-399 5572

JB: No. 25-01, Jalan Bestari 8/2, Taman Nusa Bestari, 79150, Iskandar Puteri, Johor.

Tel: +607-238 7978 | Fax: +607-238 2162

training@cadcam.com.my | www.cadcam.com.my



GD&T in SOLIDWORKS

Key Topics

Day 1

Course Outline:

Lesson 1: Introduction to GD&T

- Learn about the definition, application and different formats for GD&T
- Familiarize with basic terminology used in GD&T such as datum.

Lesson 2: Introduction to DimXpert

- Understand the CAD function known as Dimxpert and how is it apply to GD&T.
- Learn about the different categories of DimXpert functions such as size and location and dimension, datum, geometric tolerance.

Lesson 3: Coordinate Tolerance for size and location dimensions

- Using common part model, learn how to define dimension tolerance for common manufacturing features such as bosses, simple hole, notch and etc.
- Learn to define location dimensions with standard tolerance settings such as symmetric style.
- Learn to combine multiple location dimensions for pattern manufacturing features such as simple holes.

Lesson 4: Geometric Tolerance

- Learn to define feature control panel to illustrate the manufacturing tolerance such as flatness, parallel, perpendicularity and etc.
- Using both a prismatic and turned machined parts, define both the feature control and datum symbol for complete manufacturing detailing.

Lesson 5: Auto-dimensioning & Show Tolerance Status

- Understand about the automated function to define external manufacturing features for both prismatic and turned parts.
- Learn a CAD tool to check whether the manufacturing features have been defined by using GD&T.

Lesson 6: Introduction to TolAnalyst

- Understand both the definition and application of assembly stack-up analysis.
- Learn a common workflow to perform a standard assembly stack-up analysis to evaluate the form and fit of the assembly model.
- Learn to modify the GD&T for machined features and re-evaluate the assembly stack-up analysis.

IME CADCAM TRAINING CENTRE SDN BHD

PJ: 22A, Jln. PJS 1/46, Tmn. Petaling Utama, 46000 PJ, Selangor, Malaysia.

Tel: +603-7783 6866 | Fax: +603-7781 1185

PG: 68, Jln. Perai Jaya 4, Bandar Perai Jaya, 13700 Pulau Pinang, Malaysia.

Tel: +604-399 5571 | Fax: +604-399 5572

JB: No. 25-01, Jalan Bestari 8/2, Taman Nusa Bestari, 79150, Iskandar Puteri, Johor.

Tel: +607-238 7978 | Fax: +607-238 2162

training@cadcam.com.my | www.cadcam.com.my

