

Fundamentals of Tolerance Analysis Using Tolerance Analysis Extension for PTC® Creo®

Training Syllabus

Session 1:

Chapter 1: Tolerance Analysis Fundamentals

Chapter 2: Tolerance Analysis Extension Modeling Overview

Chapter 3: Analysis and Results

Offline Student Homework

Complete exercises in Chapters 1-3 ~ 2 hours

Session 2:

Chapter 4: TAE Modeling Considerations

Chapter 5: Integrated Workflow Tools

Offline Student Homework

Complete exercises in Chapters 4 & 5 ~ 2 hours

Optional Follow-up Session

Address any questions with Chapter 4 & 5 exercises



Course Topics — Fundamentals of GD&T

What is GD&T?

- Avoiding Ambiguity in Specifications

What is ASME Y14.5?

Types of Geometry

Legal Issues Pertaining to Part Definition and GD&T

Part Definition, GD&T and Client / Supplier Relationships

Engineering Drawing Dimensioning & Tolerancing Philosophy

Functional Dimensioning and Tolerancing and Interchangeability

GD&T Symbology and Tools in ASME Y14.5-2009 & Earlier Versions

Directly-Toleranced Dimensions & Tolerances (Plus & Minus)

- Uses, Drawbacks, Ambiguity, and Limitations

Interpretation of Limits and Measurement Process Issues

Feature Types

- Regular Features of Size, Bounded Features, Complex Features

Rule #1 and Rule #2

Fundamental Rules and Definitions

Material Condition, Material Boundary, & Applicable Modifiers

Basic Dimensioning Practices

Defaults for Tolerancing Screw Threads, Splines and Gears

Actual Mating Envelopes

- Unrelated – Oriented to the Feature
- Related – Oriented to a Datum Reference Frame

Form Tolerances

- Flatness, Circularity, Cylindricity, and Straightness

Virtual Condition and Resultant Condition

- Boundaries, Calculations, and Formulas

Orientation Tolerances

- Perpendicularity, Parallelism, and Angularity

Datums and Datum Reference Frames

- Definitions
- Degrees of Freedom and Constraint
- Selecting Functional Datum Features
- Specifying Datum Features
- Comparing Types of Datum Features, Datum, Datum Feature Simulators, and Constraint
- Datum Feature Precedence
- Tolerancing Datum Features
- Effect of Modifiers on Datum Feature Simulation
 - o RMB, MMB, LMB and Datum Feature Shift
- Datum Targets and Target Systems
- Datum Reference Frame Examples
- Datum Targets and Target Systems
- Constructing Datum Reference Frames and Coordinate Systems

Location Tolerances

- Positional Tolerancing
 - o Comparison with Plus and Minus Tolerancing
 - o Surface vs. Center Geometry
 - o RFS, MMC, LMC Examples
 - o Additional (Bonus) Tolerance
 - o Positional Tolerancing and Functional Gaging Examples
 - o Boundary Concepts
 - o Projected Tolerance Zones
 - o Floating Fastener and Fixed Fastener: Examples & Calculations
 - o Simultaneous Requirements and Separate Requirements
- Concentricity & Symmetry
 - o What they mean and don't mean

Runout Tolerances

- Full Indicator Movement, Circular Runout, Total Runout and Comparison of Circular and Total Runout

Profile Tolerances

- Profile of a Line and Profile of a Surface
- The Power of Profile
- Uniform and Non-Uniform Profile Tolerances
- Profile Tolerancing in Context
- Specifying Limited Length or Area of Application
- Defining the Extents of Application
- Equal-Bilateral Profile Tolerancing
- Unequal Bilateral Profile Tolerancing
- o Unilateral Profile Tolerancing
- o All-Around Profile Tolerancing
- o Specifying Profile Between Points