



GD&T Training Course: Datums, Position Tolerancing and Tolerance Stack-Up



GD&T Training Course: Datums, Position Tolerancing and Tolerance Stack-Up

Introduction:

GD&T training covering datums, position tolerancing and tolerance stack-up is a 5-day course for design, manufacturing and quality engineering teams and dimensional inspectors that ends with a Case Assembly GD&T and Stack-Up File. Organisations pay for scrapped parts, assembly interference and inspection disputes when drawings rely on plus-minus coordinate limits, ambiguous datums and tolerances nobody has checked against the assembly gap. Nominees already read or issue mechanical drawings and learn through case studies on real part and assembly drawings, worked callout by callout. CoreConcept Training Center delivers this GD&T training course.

Course Objectives:

- Interpret feature control frames, basic dimensions and symbols on drawings to ASME Y14.5 and ISO GPS and explain the tolerance zone each one creates
- Establish datum reference frames from functional datum features and choose datum precedence that matches how parts locate in the assembly
- Specify form, orientation, position, profile and runout tolerances that protect fit and function without over-constraining manufacture
- Calculate bonus tolerance, virtual condition and resultant condition for features of size controlled at MMC or LMC
- Analyse assembly gaps with worst-case and statistical tolerance stack-up methods and reallocate tolerances where the loop fails
- Plan verification of geometric callouts with functional gauges, CMM datum alignment and hand gauges on a surface plate

Target Audience:

- Design engineering teams responsible for assigning tolerances and datums on part and assembly drawings
 - Manufacturing engineering teams responsible for fixturing, machining set-ups and process plans built from drawing datums
 - Quality engineering teams responsible for inspection plans, first article approval and supplier drawing queries
 - Dimensional inspection and metrology teams responsible for CMM programmes and hand gauge checks
 - Supplier quality and sourcing engineering teams responsible for resolving tolerance disputes with vendors
-

Course Outline:

Day 1: GD&T Foundations, Drawing Rules and Symbol Language

- Coordinate Plus-Minus Limits Versus Geometric Tolerance Zones
- ASME Y14.5 and ISO GPS Principles Compared Side by Side
- Feature Control Frame Anatomy, Basic Dimensions and Symbols
- Feature of Size Rules and the Envelope Principle
- Current-State Review of Legacy Part Drawing Callouts

Day 2: Datum Reference Frames, Form and Orientation Controls

- Datum Features, Datum Targets and Datum Feature Simulators
- Datum Reference Frame Construction and Degrees of Freedom
- Flatness, Straightness, Circularity and Cylindricity Form Tolerance Zones
- Perpendicularity, Parallelism and Angularity Orientation Controls Applied
- Datum Precedence Order and Its Effect on Part Set-Up

Day 3: Position, Profile, Runout and Material Condition Modifiers

- Position Tolerance Zones for Hole Patterns and Pins
- MMC, LMC and RFS Modifiers with Bonus Tolerance Calculation
- Virtual Condition and Resultant Condition Boundary Calculations
- Profile of a Surface and Profile of a Line Callouts
- Circular Runout and Total Runout on Rotating Shafts

Day 4: Tolerance Stack-Up Analysis, Functional Gauging and Inspection

- Worst-Case Tolerance Stack-Up Using Dimension Loop Charts
- Statistical Stack-Up with Root Sum Square and Monte Carlo
- Functional Receiver Gauge Design for MMC Position Callouts
- CMM Datum Alignment, Probing Strategy and Feature Construction
- Hand Gauge Inspection with Surface Plates and Dial Indicators

Day 5: Case Study: GD&T Specification and Stack-Up on a Case Assembly

- Case Assembly Functional Requirements and Datum Scheme Selection
 - Feature Control Frame Assignment Across Case Assembly Parts
 - Common Drawing Error Audit Against ASME Y14.5 Rules
 - Case Assembly Gap Stack-Up in Worst-Case and RSS
 - Case Assembly GD&T and Stack-Up File Completion and Peer Review
-

Skills You Will Gain:

- Feature Control Frame Interpretation
- Datum Scheme Design
- Geometric Tolerance Selection
- Material Condition Calculation
- Tolerance Stack-Up Analysis
- Functional Gauge Specification
- CMM Inspection Planning
- Drawing Error Detection

Why Attend This Course:

- Deliver a Case Assembly GD&T and Stack-Up File to the design engineering lead and the quality or inspection lead
- Decide whether a gap needs worst-case or statistical stack-up treatment and which part tolerance to tighten or release
- Avoid scrapped parts, rejected supplier lots and assembly interference caused by missing datums and unchecked tolerance loops
- Share a datum selection routine, stack-up worksheet and drawing review checklist with design and inspection colleagues

Conclusion:

Back at work, the participant hands the design engineering lead and the quality or inspection lead a Case Assembly GD&T and Stack-Up File containing a datum scheme, feature control frames, a gap stack-up and a gauging and CMM plan. Design leads use it to set a drawing review checklist for new parts, while quality leads use it to agree inspection set-ups with suppliers before first article approval. After the first drawing released under the file, the unit should review inspection queries, gauge rejections and assembly fit findings against the stack-up assumptions.

Frequently Asked Questions FAQ:

What should participants know before the GD&T training course on datums, position tolerancing and tolerance stack-up?

Participants should already read mechanical drawings and understand plus-minus dimensioning and basic workshop measurement. Prior use of geometric symbols is helpful but not required. Bringing a part or assembly drawing from their own unit lets them test each callout against familiar geometry.

How does GD&T training on datums, position tolerancing and tolerance stack-up differ from a CAD drafting or statistical process control course?

It teaches how to specify, interpret, stack up and verify geometric tolerances, whatever software draws them. CAD courses teach modelling and drafting commands, and process control courses monitor production variation with control charts rather than defining tolerance zones on the drawing.

Why does GD&T position tolerancing with an MMC modifier allow bonus tolerance?

When a hole departs from its maximum material size, more clearance exists for the mating pin, so the position zone grows by that departure. The part still assembles at its virtual condition boundary, and a functional gauge can confirm it in one check.

What do participants take back from the GD&T training course on datums, position tolerancing and tolerance stack-up?

Participants take back a Case Assembly GD&T and Stack-Up File with a datum scheme, feature control frames for each part, worst-case and statistical gap calculations, a corrected drawing error list and a gauging and CMM inspection plan ready to adapt to their own products.