

Fundamentals of Geometric Dimensioning and Tolerancing

By QCS*

This course is intended for technical people who need to gain an understanding of GD&T and also for those needing a refresher. This is a one day course. The text currently used is intended to provide the attendee with an excellent reference source both during the sessions as well as in the future. This reduces the number of class hours and considers the retention of participants, in particular where not every aspect of GD&T is continually used. The company is billed for the texts at the cost paid by the instructor. Measurement techniques are discussed along with their limitations.

Quizzes are given for the benefit of the participant. There are no pass/fail criteria or scoring. The questions with answers are reviewed after the quizzes. All participants receive a certificate of attendance. The course generally runs from 8:30 AM to 4:30 PM.

Course Content

Introduction

- Functional relationships
- Rounding of numbers

Terminology

- Feature control frames
- Symbols
- Bonus tolerance
- MMC, RFS
- Primary two rules

Form

- Flatness
- Straightness
- Circularity
- Cylindricity

Datum

- The datum system
- Datum targets

Orientation

- Perpendicularity
- Angularity
- Parallelism

Position (True position)

- Advantages
- MMC application
- Virtual
- Projected tolerance zone
- Gages
- Fixed and floating fasteners

Concentricity/Symmetry

- Concentricity
- Symmetry

Runout

- Circular Runout
- Total Runout

Profile

- Profile tolerancing
- Profile of a line
- Profile of a surface

*Contact:

Nick Squeglia

nsqueglia@snet.net