

1-DAY GD&T PRIMER

COURSE DESCRIPTION

Intended for the professional or manager seeking an awareness of dimensioning and tolerancing who does not need to further their knowledge with a more in-depth course. Designed to provide you with an awareness of GD&T fundamentals, this course focuses on the basic requirements of engineering drawings, numeric dimensions and tolerances and geometric dimensions and tolerances.

LEARNING OBJECTIVES

- Calculate minimum and maximum allowable values for dimensions considering numeric tolerances.
- Learn how geometric tolerances supplement conventional tolerances to specify limits on allowable variation.
- Recognize the symbols used with the geometric system of tolerances.
- Recognize basic dimensions on prints and understand their meaning.
- Identify and properly read feature control frames.
- Recognize when bonus tolerance is available and calculate geometric tolerance values when bonus tolerance applies.
- Recognize datum feature identifiers applied to features with size and surfaces.
- Analyze the datum reference frame including order of precedence.
- Identify and read geometric controls on location, orientation, form, profile & runout.

WHO SHOULD ATTEND

This course is designed for engineering managers and supervisors, mechanical engineering technicians, manufacturing engineering technicians, quality assurance technicians, and supplier quality engineers.



DYNAMIC DESIGN CONSULTING, LLC

www.dynamicdesignconsulting.com

1-DAY GD&T PRIMER

COURSE OUTLINE

- **The Basics of Manufacturing Prints:** What manufacturing prints are and why they are used. Who uses them and how. The manufacturing cycle and purpose of prints. The basic requirements of technical drawings.
- **Dimensions:** Fundamental dimensioning and tolerancing rules. Symbols associated with dimensions. Linear, diametral, radial, and angular dimensions. Dimensioning of common features.
- **Numeric Tolerances:** The ways numeric tolerances are indicated on prints. Determining tolerance values for numeric tolerances given implicitly, explicitly, symbolically, and by limit dimensioning.
- **Geometric Tolerance Fundamentals:** How geometric tolerances work. Thinking of parts as collections of features not dimensions. Features with size and surfaces. Maximum Material Condition and Least Material Condition. What bonus tolerance is and when it applies.
- **The Datum Reference Frame Concept:** The purpose of a datum reference frame. Establishing datums from various datum features. What datum shift is and when it applies.
- **Interpretation of Geometric Controls:** The geometric characteristic symbols used to impose geometric requirements.

Prerequisites

None.

Course Format

8 hours: 1 in-person session or 2 online sessions



DYNAMIC DESIGN CONSULTING, LLC

www.dynamicdesignconsulting.com

