

SPC consulting group

GD&T

GEOMETRIC, DIMENSIONING & TOLERANCING

Objective:
The participant will learn the symbols and basic rules used in GD&T, the correct interpretation of engineering drawings and recognizing the importance of communication between customers and suppliers through a standardized framework.

Prerequisites
- Geometry basics

Duration:
16 hours

Introduction

Basic Concepts
- Introduction
- Definitions
- Basic rules
- Units of measure

General Tolerancing
- Basic tolerances
- General Tolerances
- Modifiers
- Feature Control Frame
- Maximum & Less Material Conditions

Datums
- Degrees of freedom
- Reference frames
- Order of preference
- Establishing Datums
- Multiple datums
- Specific datums

Tolerances of Form
- Straightness
- Flatness
- Circularity
- Cilindricity

Tolerances of Orientation
- Angularity
- Parallelism
- Perpendicularity

Tolerances of Location
- Position
- Concentricity
- Symmetry
- MMC, RFC, LMC

Tolerances of Profile
- Profile of a line
- Profile of a surface

Tolerances of Runout
- Circular runout
- Total runout