



Geometric Dimensioning & Tolerancing

Start Date: 5/9/2019

End Date: 5/9/2019

At the end of this course, the student should be able to:

- Recognize standard GD&T symbols
- Define the Datum Reference Frame
- Understand the difference between traditional tolerancing and GD&T
- Understand material condition modifiers

The GD&T Process

- What is GD&T
- When to use GD&T
- Commonly used GD&T terms and symbols
- Datums and Features
- The Datum Reference Frame

How the Geometric System Works

- Plus/Minus Tolerancing
- Geometric Tolerance Zones
- Material Condition Modifiers
- The Feature Control Frame

- Form and Profile Tolerances
- Orientation, Location and Runout Tolerances
- General Rules of GD&T

- Rule #1
- Rule #2

Limit Tolerancing vs. Geometric Tolerancing

Lunch is provided

Location(s)

Gadsden ATN Center