

DD 160 - TOLERANCING AND GD&T

Course Description

This course first presents conventional tolerancing terminology, expressions, and accumulations in both inch and metric formats. Next, Geometric Dimensioning and Tolerancing (GD&T) presents an international system of symbols used to dimension products or tooling components. The course is based on the current ASME Y14.5M2009 Dimensioning and Tolerancing standard. Engineers, designers, drafters, cost estimators, machinists, and inspectors must understand this system. Group 2 course.

Credit Hours

3

Contact Hours

3

Lecture Hours

3

Required Prerequisites

DD 101

General Education Outcomes supported by this course

Critical Thinking - Direct

Course Learning Outcomes

Knowledge:

- Use English and metric dimensions and tolerance expressions.
- Complete tolerance accumulations.
- Interpret mechanical drawings using conventional and geometric dimensioning and tolerancing.
- Interpret size/shape tolerance zones.

Application:

- Use engineering drawings to interpret product requirements.

Integration:

- Describe engineering tolerances.
- Evaluate a manufactured workpiece to verify its compliance.

Human Dimension:

- Work cooperatively to achieve a common goal.

Caring - Civic Learning:

- Reflect upon the importance of accuracy in product creation.

Learning How to Learn:

- Utilize Tolerancing and GDT skills in real-world applications.