

# MAC 122 CNC Turning

## COURSE DESCRIPTION:

Prerequisites: MAC 121; DMA 010, 020, 030; DRE 096

Corequisites: None

This course uses Mastercam programming, setup, and operation of CNC turning centers. Topics include programming formats, control functions, program editing, part production, and inspection. Upon completion, students should be able to manufacture simple parts using Mastercam and CNC turning centers. Course Hours per Week: Class, 1. Lab, 3. Semester Hours Credit, 2.

## LEARNING OUTCOMES:

Upon completion of this course, the student will be able to:

- a. Understand the procedures and concepts of programming, set up and operation of a CNC Lathe.
- b. Identify and understand the programming codes.
- c. Create geometry and toolpaths from the specifications on a blueprint for simple parts using programming software.
- d. Identify and define the functions of the CNC machine control.
- e. Set up the CNC machining center for manufacturing parts
- f. Manufacture parts on the CNC machining center.

## OUTLINE OF INSTRUCTION:

- I. CNC Lathe
  - A. Machine CNC system description
  - B. Axis designation/coordinate system
  - C. Code description/data format
    1. G codes
    2. M codes
    3. S and F codes
  - D. Tooling and setup procedures
    1. Z length and diameter set
    2. Reference, tool change and special point set locations
    3. Tooling safety
    4. Tooling menus
  - E. Control orientation
    1. Start-up procedures
    2. Diagnostic checks
    3. Menu format
  - F. Operation procedures
    1. Manuscript reading
    2. Program editing
      - a. speed/feed overrides

- b. dimensional changes
- 3. Manual data input (MDI)
- 4. Set-up tooling procedures
- 5. Operation of CNC lathes and Turning Centers

**REQUIRED TEXTBOOK AND MATERIAL:**

The textbook from MAC 121 Intro to CNC will be used in this course.

The instructor will use various handouts generated by Durham Tech for operation and programming of CNC mill.