# MET-2000: CAD/CAM PROCESSES

## Cuyahoga Community College

**Viewing: MET-2000 : CAD/CAM Processes**
**Board of Trustees:**
2018-05-24

**Academic Term:**
Fall 2018

**Subject Code**
MET - Mech Eng/Manuf Ind Eng Tech

**Course Number:**
2000

**Title:**
CAD/CAM Processes

**Catalog Description:**
Using Mastercam and other Computer-Aided Design/Computer-Aided Manufacturing (CAD/CAM) software to graphically model parts; graphic display manipulation; geometrical analysis; graphic and data files management; exchange and conversion of graphic files to formats readable by Mastercam or given CAD/CAM software; generating codes, post processing to G-codes interpretable by given computer numerical controller; verification and validation of tool-paths by graphical simulation; downloading path programs to machine; tooling and setting up parts on CNC lathe and milling machines; operating CNC machines to produce parts.

**Credit Hour(s):**
3

**Lecture Hour(s):**
2
**Lab Hour(s):**
3

## Requisites

**Prerequisite and Corequisite**
MET-1400 CNC Programming and Operation or concurrent enrollment.

## Outcomes
**Course Outcome(s):**
Apply the fundamentals of CAD/CAM applications to design and manufacture engineering parts.

**Essential Learning Outcome Mapping:**
Critical/Creative Thinking: Analyze, evaluate, and synthesize information in order to consider problems/ideas and transform them in innovative or imaginative ways.

**Objective(s):**
1. Recognize the safety precautions to be observed in the programming and operations of a CAD/CAM system.
2. Differentiate between direct and graphic based CNC programming.
3. Interpret parts models.
4. Select appropriated tooling and determine the desired spindle speed, feed-rate, and other machining parameters required to accomplish CNC code generation.
5. Generate G-codes and post process them for a given CNC controller type FANUC, MAZAK, OKUMA, etc.
6. Verify path programs by simulation.
7. Download path program to the machine control unit (MCU).
8. Set up parts for machining, determining tools? offsets, establishing work ?zero,? and tools change positions.
9. Run CNC milling machine or lathe to produce part.

**Methods of Evaluation:**
1. Lab reports, models inspection, simulation analysis
2. Quizzes and assignments
3. Midterm examination
4. Final examination

**Course Content Outline:**
1. Concepts
   a. Introduction to Mastercam
      i. Starting Mastercam
      ii. Standard Menus and Toolbars
      iii. Function Keys and Quick Keys
      iv. Coordinate System
      v. Creating Simple Geometry
         1. Circles
         2. Lines
         3. Rectangle
         4. Save new geometry
         5. Open existing file
         6. Customizing Mastercam
         7. Getting Help.
   b. Creating 2D Geometry
      i. Steps to Creating 2D Geometry
      ii. Open a new file
      iii. Create a Rectangle
      iv. Use Screen-Zoom tool
      v. Delete geometry
      vi. Create a round (fillet)
      vii. Create and trim the bottom notch
      viii. Create a Point, Arc (Circle)
      ix. Create Point Patterns using the Bolt Circle and Grid options
      x. Xform tools
      xi. Add dimensions
      xii. Analyze the part
   c. Creating 3D Geometry
      i. Steps to Creating 3D Geometry
      ii. Change perspective of the part
      iii. Translate geometry in the Z direction
      iv. Change the Cplane
      v. Create geometry in the side and top Cplanes
      vi. Translate entities without linking
      vii. Create Pocket geometry
      viii. Create new Cplane
      ix. View new Cplane with Gview
      x. Create circle on new Cplane
      xi. Translate circle in the Z direction
      xii. Trim the circle
      xiii. Translate arc to complete angled slot
      xiv. Create 3D dimensions
   d. Creating Surface Geometry
      i. Steps to Creating Surface Geometry
      ii. Create Planar surface using existing wireframe geometry
      iii. Trim Planar surface
      iv. Remove existing boundaries
      v. Create a Lofted surface
      vi. Create a surface using the Sweep tool
      vii. One last sweep
   e. Importing Existing Geometry

        i. Steps to Importing Existing Geometry
       ii. Pull IGES file off internet
      iii. Open IGES file
      iv. Translate the imported part to Mastercam Axis System
       v. Rotate part 90 degrees
      vi. Shade surfaces.

f. Creating a Job Setup
        i. Steps in Creating a Job Setup
       ii. Know and understand all part requirements
      iii. Review the part and plan the milling process
      iv. Create the Raw Stock

g. Creating Facing Toolpaths
        i. Steps in Creating Facing Toolpaths
       ii. Define the Facing Plane
      iii. Define the Tooling Parameters
      iv. Tooling operations
       v. Modify an existing Toolpath
      vi. Verify the Facing Toolpath

h. Creating Contouring Toolpaths
        i. Steps to Creating the Contour Toolpath
       ii. Check the Facing Toolpath
      iii. Create the basic Contour Toolpath
      iv. Modify the Contour Toolpath

i. Creating Pocket Toolpaths
        i. Steps to Creating Pocket Toolpaths
       ii. Create the basic Pocket Toolpath
      iii. Create the open Pocket Toolpath
      iv. Create the Island Pocket Toolpath

j. Creating Drill/Surface Toolpaths
        i. Steps to Creating Drill Cycle Toolpaths
       ii. Create the Bolt Circle Drill Toolpaths
      iii. Create the Grid Drill Toolpath
      iv. Create the Surface Toolpath
       v. Final Toolpath - face off excess material
      vi. Final analysis
     vii. Post it
    viii. The Setup Sheet

k. Introduction to Mastercam Lathe
        i. Creating Lathe Toolpaths
       ii. Steps to Creating the Hammer Handle using Mastercam Lathe
      iii. Start the Mastercam Lathe
      iv. Download Hammer Handle.MC9 file
       v. Open the Hammer Handle.MC9 file
      vi. Create Lathe Job Setup
     vii. First lathe operation - facing the part; create a Center Drill operation
    viii. Modify the Job Setup
     ix. Create a Quick Rough Toolpath
      x. Create the Finish Cut
     xi. Next Roughing Toolpath
    xii. Create Threads.

2. SKILLS
   a. Perform the four arithmetic operations on whole numbers, arithmetic fractions, and decimal fractions;
   b. Identify the relationships in geometric figures;
   c. Convert between the metric and English measurement systems and convert within each system;
   d. Read and write "G- and M-codes" Fanuc;
   e. Compute feeds and speeds for appropriate materials;
   f. Set up and operate a CNC mill in a safe and efficient manner
   g. Operate milling machine

    h. Read blueprints
    i. Model parts in 2D, 3D wireframe , 3D surfaces, solid geometry
    j. Setup models for CNC Toolpaths generation
    k. Machine parts by simulation
    l. Verify and Validate Toolpaths by solid graphical simulation
   m. Troubleshoot and Edit Toolpaths
    n. Generate and postprocess Toolpath readable by FANUC controllers.

3. ISSUES
    a. Equipment sufficiency
    b. Safety in Operation of CNC machines
    c. CNC interface program updates
    d. Networking for direct Programs transfer

## Resources

Rehg, James and Henry Kraebber. *Computer Integrated Manufacturing.* 3rd Ed. Upper Saddle River, NJ., 2005.

Chang, Tien-Chien, Richard Wysk and Hsu-Pin Wang. *Computer-Aided Manufacturing.* 3rd Ed. Upper Saddle River, NJ., 2006.

In-House Solutions. *Mastercam 2017 Beginner Training Tutorial.* 2017. 2017.

Verma, Gaurav and Matt Weber. *Mastercam 2017 for SolidWorks Black Book.* 2017. CADCAMCAE Works, USA., 2017.

Groover, Mikell. *Automation, Production Systems, and Computer-Integrated Manufacturing.* 4th ed. Boston, MA: Pearson, 2015.

Katz, Joseph. "Journal Of Manufacturing Science and Engineering" Monthly. 2008-10-01 00:00:00.0.

### Resources Other

1. *Journal Of Manufacturing Science and Engineering* http://manufacturingscience.asmedigitalcollection.asme.org/journal.aspx  2018.
2. Mastercam V9 Workbook  by Richard Cozzens
3. Software: Mastercam
4. Machine and software manuals.
5. Handouts.

Top of page
Key: 2903