Computer Aided Manufacturing

- CNC Milling used as representative example of CAM practice.
- CAM applies to lathes, lasers, waterjet, wire edm, stamping, braking, drilling, etc.
- CAM derives process information from the CAD model (although this is sometimes interpreted and re-entered).

CNC Milling

- Advantages of CNC
  - high accuracy with limited verification time
  - higher metal removal rates
  - consistency and accuracy across feeds/speeds
  - operators able to manage several machines
  - inspections can be reduced
  - complex contouring and surfacing is possible

- Disadvantages of CNC
  - not as flexible as hand work
  - compensation systems need intelligence
  - high initial cost
  - higher maintenance costs
  - part programs are required

- Classification of CNC Machines
  - Machine Type: PTP vs Contouring
  - Controller: Hardware NC vs CNC/DNC
  - Control System: Closed loop vs Open loop

  - Point to Point
    - machine does not “cut” while positioning drives are in motion (e.g. drilling machine)
  - Contouring
    - cutting while drives are moving (e.g. milling)

- Open loop vs Closed Loop Control
  - Open Loop - consider stepper motor drives
    - signal from interpolator (# of pulses & pulse freq.)
    - stepper motor
    - gearing (normally required)
    - drive leadscrew, through to table or spindle axis
    - no feedback
    - limited ability to deal with variable forces, therefore usually used only in PTP machines.
    - BLU defined by one pulse as translated through geartrain.
  - Closed Loop Control
    - signal from interpolator
    - comparator evaluates error
    - DAC converts to analog signal
    - servo drive motor
    - gearing system
    - leadscrew or other drive
    - encoder or linear scale or other feedback sent to comparator.
    - Compensation possible in software.
• Requirements for CNC Machining
  – geometric information (usually direct from CAD). Sometimes requires adjustment to get “best” toolpaths.
  – Machining parameters (handbook data)
  – Process planning information (what to do first)
  – Any machine dependant information although this is often added after a generic CNC plan is produced (I.e. CL file or intermediate format)
  – Post processing for specific machine tool
  – appropriate tooling and fixturing

• Directions for future
  – more powerful processors, more memory
  – more intelligent software and CAD models
  – Adaptive Control
    • adjusting parameters on the fly
      – constraints: cutting force, machining power, cutting torque
      – parameters: feedrate, spindle speed
    • optimizing parameters based on solid modeling info
    • monitoring for tool wear and/or breakage
    • vision systems

• What is G-Code?
  – G-Code programming is a very robust and simple language. G-codes are what CNC Machines run on. A CNC control program processes the g-code line by line and sends this information to the machine. Precise machine movements result.

• Where does G-Code come from?
  – From a CAM program and various machining parameters and inputs.
  – It is post-processed for a particular machine

• G-Code List
  – G0 or G00 – Rapid Movement
    • The most rapid movement the CNC Machine can make to the next position. If moving in multiple axis, each axis will move as fast as they can independently of one another.
  – G1 or G01 – Linear Movement
    • A straight move with a speed defined by an “F.” [F=Feedrate] If moving in multiple axis, the machine will move in each axis until it reaches its defined position.

• Post Processors
  – A post processor is a specific to the machine and normally provided by the machine company or the CAM vendor.
  – There are standard and custom ones. If a standard one doesn’t work, most CAM software gives the option to customize the post processor

• Canned Cycles
  – There are G-Codes that do pertain to Canned Cycles. These are “subroutines” to complete more complex operations (e.g. peck drilling).

• G-Code List
  – G2 or G02 – Interpolation Clockwise
    • A circular movement in 2-axis in a clockwise motion. Will create an arc to a specified radius defined by an R or I/J combination.
  – G2 or G03 - Interpolation Counter Clockwise
    • A circular movement in 2-axis in a counter-clockwise motion. Will create an arc to a specified radius defined by an R or I/J combination.
  – G4 or G04 – Dwell
    • Machine will dwell once reached position to a user defined time, noted by a “P”

ENGI7962 Computer-Aided Engineering
• G-Code List
– G20 - Machine in inch
– G21 - Machine in MM
– G28 - Return to Reference Position
  • Normally machine home.
– G40 - Cutter Compensation Cancel
– G41 - Cutter Compensation Left
– G42 - Cutter Compensation Right
– G43 - Tool Length Compensation +
– G44 - Tool Length Compensation -

• G-Code List
– G54-G59 - Work Coordinate Systems
  • User defined, XYZ
– G80 - Cancel Canned Cycle
– G81 - Drilling Cycle
– G82 - Drilling Cycle
– G83 - Pecking Cycle
– G84 - Tapping Cycle
– G90 - Absolute Command
– G91 - Incremental Command

• 2" Square G-Code Example
– The numbers on the left of the G-Code are the sequence numbers of the code and the order it will be executed in. Notes are not usually included. Notes start with a "-" then continues. (i.e. a "-" means nothing in G-Code).

• What are M-Codes?
– M-Codes are other codes that are throughout your CNC Program. Think of them like the lesser cousin to G-Codes.
– M-Codes are usually very specific to the machine. For example, “M03” might stand for “Torch Off” on a CNC Plasma Cutter. It also might stand for “Spindle Turn Clockwise” on a CNC Mill or CNC Router.
• **M-Codes**
  - M0 - program stop
  - M1 - optional program stop
  - M2 - program end
  - M3 - turn spindle clockwise
  - M4 - turn spindle counterclockwise
  - M5 - stop spindle turning
  - M6 - tool change
  - M7 - mist coolant on
  - M8 - flood coolant on
  - M9 - mist and flood coolant off
  - M30 - program end, pallet shuttle, and reset
  - M48 - enable speed and feed overrides
  - M49 - disable speed and feed overrides