Chapter 5:
CNC Milling
CNC
Learning objectives

- Hands-on programming for milling operations
- Linear and circular interpolation programming
- Cutter diameter compensation
- Letter address command for the CNC Mill
Command Line
Letter Addresses (Characters)

G-codes and M-codes Command Lines contain

- specific characters for designated functions of the spindle:
  - D ...... Z excluding: L, O, Q, U, V, W

- Diameter offset register number (D)
  - Used to call the specified offset register for cutter diameter compensation

- Z-axis definition (Z)
  - Designates a coordinate along the Z-axis

Refer to pages 72 -73 for list of CNC Milling letter addresses
G-codes are preparatory functions involve actual tool moves:

- Rapid moves, radial feed moves
- G00 – Position in rapid
- G01 – Linear Interpolation
- ..................
- Most G-codes remain active until another code is issued (Modal codes)

Refer to pages 74 for list of CNC Milling G-codes
Rapid Move

G00: To Position Spindle in Rapid Movement

- Use the G00 command to move the tool linearly and quickly from one point to another \textit{without} cutting
  - Tool positioning
- General Format:
  - \texttt{N\_ G00 X\_ Y\_ Z\_}
  - \(X, Y\) and \(Z\) represent respective axis along the width, length and thickness of the work piece (the material)
- Standard Format for most CNC Machines:
  - Adopt two separate moves
    - Rapid move in XY plane
    - Rapid Z move
    - Which move should be executed first?
      - If \(Z\)- value represents a cutting move in the negative direction, the \(X\) and \(Y\) axes should be executed first
      - If the \(Z\)- value represents a move in the positive direction, the \(X\) and \(Y\) axis should be executed last
Rapid Move
G00 Demo

- Let’s go to the Simulator:
  - Open file: G00EX1.MiLL
  - First cycle thru’ and then step thru’ the Blocks
- What is the Work piece size?
- What Tool number is used?
- What is the Tool Start Position?
- What observations can you make about the following Blocks:
  - N20 and N25?
  - N50 and N55?
  - Edit the code and re-run the simulator
  - Save the edited file with a different name (myg00.mill)
Linear Interpolation
G01

- Use the **G01** command for linear removal of material from a work piece in any combination of X, Y, or Z axis:
  - Linear interpolation: specifies X-Y (no Z value)
  - Multiaxis (all diagonal): specifies X-Y-Z axis

- **General Format:**
  - \[ N_\_ G01 X_\_ Y_\_ Z_\_ F_\_ \]
    - X, Y and Z represent respective axis along the width, length and thickness of the work piece (the material)
    - F specifies a feedrate in inches or millimeters per minute
Linear Interpolation
G01 Demo

- Let’s go to the Simulator:
  - Open file: G01EX2.MiLL
  - First cycle thru’ and then step thru’ the Blocks
- What is the Work piece size?
- What Tool number is used?
- What is the Tool Start Position?
- What observations can you make about the following Blocks:
  - N20 and N25?
  - N30 and N35?
  - Edit the code and re-run the simulator
  - Save the edited file with a different name (myg01.mill)
Circular Interpolation (Clockwise - CW)  

G02

- Use the G02 command for all clockwise radial (or arc) feed moves:
  - Quadratic arcs, partial arcs, complete circles
  - Moves must be in a 2-d plane
  - Multiaxis (all diagonal): specifies X-Y-Z axis

**General Format:**

But adopt two separate moves

- \[N_ \_ G02 \_ X_ \_ Y_ \_ Z_ \_ I_ \_ J_ \_ K_ \_ F_ \]
- \[N_ \_ G02 \_ X_ \_ Y_ \_ Z_ \_ R_ \_ F_ \]

or

- Let \((x_0, y_0, z_0), (x_1, y_1, z_1), (x_2, y_2, z_2)\) rep the coord of the arc’s center-point, start-point and end-point respectively then:
  - \[G02 \_ X_ \_ Y_ \_ Z_ \_ I_ \_ J_ \_ K_ \_ F_ \] = \[G02 \_ X_{x_2} \_ Y_{y_2} \_ Z_{z_2} \_ I_{(x_0 - x_1)} \_ J_{(y_0 - y_1)} \_ K_{(z_0 - z_1)} \_ F_ \]

- R specifies the radius (i.e. the incremental distance from the starting-point to the center-point)
- F specifies a feed-rate in inches or millimeters per minute
Circular Interpolation (Clockwise CW) G02 Demo

- Let’s go to the Simulator:
  - Open file: G02EX3.MiLL
  - First cycle thru’ and then step thru’ the Blocks
- What is the Work piece size?
- What Tool number is used?
- What is the Tool Start Position?
- What observations can you make about the following Blocks:
  - N35?
  - N45 and N50?
  - Edit the code and re-run the simulator
  - Save the edited file with a different name (myg02.mill)
Circular Interpolation (Counter Clockwise - CCW)

G03

- Use the **G03** command for all **counter clockwise** radial (or arc) feed moves:
  - Quadratic arcs, partial arcs, complete circles
  - Moves must be in a 2-d plane
  - Multiaxis (all diagonal): specifies X-Y-Z axis

**General Format:**

- N_ G03 X_ Y_ Z_ I_ J_ K_ F_
- N_ G03 X_ Y_ Z_ R_ F_
  
  or

- Let \((x_0, y_0, z_0), (x_1, y_1, z_1), (x_2, y_2, z_2)\) rep the coord of the arc’s center-point, start-point and end-point respectively then:
  - \(G03 \ X_ Y_ Z_ I_ J_ K_ F_ = G03 \ X_{x_2} \ Y_{y_2} \ Z_{z_2} \ I(x_0-x_1) \ J(y_0-y_1) \ K(z_0-z_1) \ F_\)

- **R** specifies the radius (i.e the incremental distance from the starting-point to the center-point)
- **F** specifies a feed-rate in inches or millimeters per minute
Circular Interpolation (Counter Clockwise \textbf{CW})

G03 Demo

- Let’s go to the Simulator:
  - Open file: \texttt{G03EX4.MiLL}
  - First cycle thru’ and then step thru’ the Blocks
- What is the Work piece size?
- What Tool number is used?
- What is the Tool Start Position?
- What observations can you make about the following Blocks:
  - N40, N50 and N60?
  - Edit the code and re-run the simulator
  - Save the edited file with a different name (myg03.mill)
Use the **G04** command to pause all axis movement for a specified time while spindle continues to revolve at the specified rpm

- Ideal for clearance of chips after drilling

**General Format:**

- **N_ G04 P_**

  - The letter P represents time period to pause, and is followed by time measured in seconds (e.g., for 2 seconds: **P2**)

---

Dwell

**G04**
Dwell
G04 Demo

- Let’s go to the Simulator:
  - Open file: **G04EX5.MiLL**
  - First cycle thru’ and then step thru’ the Blocks
- What is the Work piece size?
- What Tool number is used?
- What is the Tool Start Position?
- What observations can you make about the following Blocks:
  - N30 and N45?
  - Edit the code and re-run the simulator
  - Save the edited file with a different name (myg04.mill)
The machine tool can move in the horizontal (X-Y) and vertical (Z) directions:

- Use the **G17** command to (re)set system to **XY** plane
  - `N_ G17`
  - Demo file: **G17EX6.MiLL**

- Use the **G18** command to (re)set system to **XZ** plane
  - `N_ G18`
  - Demo file: **G18EX7.MiLL**

- Use the **G19** command to set system in **YZ** plane
  - `N_ G19`
  - Demo file: **G19EX8.MiLL**
Return to Reference Point (Via Intermediate point)

G28

- Use the G28 command to move tool from a starting point to a predefined reference point via an intermediate position.

- General Format:
  - N_ G28
  - N_ G28 X_ Y_ Z_ (Xk Yn Zm rep radial dist of intermediate point)

- Demo file: G28EX111.MiLL
  - What observations can you make about block:
    - N35
Return from Reference Point (Via Intermediate point)

G29

- Use the **G29** command to move tool from the reference point to the intermediate and finally to the end point
- **General Format:**
  - \[ \text{N}_n \text{ G29} \]
  - \[ \text{N}_n \text{ G29 X}_n \text{ Y}_n \text{ Z}_n \] (Xk Yn Zm rep radial dist of intermediate point)

**Demo file:** G29EX112.MiLL

- **What observations can you make about block:**
  - N35
More G-codes

Simulator Runs

- Run and observe simulator runs for the following G-codes:
  - N_ G40 (Cutter Compensation cancel) Program: G40EX11.MiLL
  - N_ G41 D_ (Cutter Compensation Left) Program: G41EX12.MiLL
  - N_ G42 D_ (Cutter Compensation Right) Program: G42EX13.MiLL
  - N_ G44 H_ (Tool Length Compensation: Minus) Program: G44EX15.MiLL
  - N_ G54 ..G59
    - (Workpiece coordination system) Program: G54EX19.MiLL
  - N_ G73 Z_ R_ Q_ F_ (High Speed Deep Hole Drilling Cycle) Program: G73EX20.MiLL
  - N_ G81 X_ Y_ Z_ R_ F_ (Drilling Cycle) Program: G81EX18.MiLL
  - N_ G82 X_ Y_ Z_ R_ P_ F_ (Spot Drilling Cycle) Program: G82EX19.MiLL
  - N_ G83 X_ Y_ Z_ R_ Q_ F_ (Deep Hole Drilling Cycle) Program: G83EX20.MiLL
Absolute Positioning

G90

- Use the **G90** command to set absolute coordinates as the default
  - Coords will be measured from the origin (X0, Y0, Z0) and expressed in X, Y and Z distances
  - Insert Command at start of program

**Format:**
- N_ G90

- Simulator Program: G90EX21.MiLL
- Run Simulator and observe System Status Window
Incremental Positioning

G91

- Use the **G91** command to set incremental coordinates as the default
  - Coords will be measured from previous point and expressed in X, Y and Z distances
  - Insert Command at start of program

**Format:**

- **N_ G91**

- Simulator Program: G91EX22.MiLL
- Run Simulator and observe System Status Window
Reposition the Origin Point

G92

- Use the **G92** command to reposition the origin point

**Format:**

- `N_ G92 X_ Y_ Z_`

- Simulator Program: G92EX23.MiLL

- Run Simulator and observe System Status Window