Chapter 3: 
CNC Programming Concepts
CNC

Learning objectives

- Format of a CNC Program
- Types of Tool Motion
CNC Program Overview

CNC Program

- A Sequence of Instructions
  - Start-up Instructions
    - Program Start
    - Program Number
  - Sequence of Machine Instructions for a Controller
    - Each instruction is comprised of CNC “machine specific codes” (commands)
      - The Controller uses individual Codes to move, position the spindle and machine a part
    - Instructions are listed sequentially in numbered blocks
  - Controller executes instructions sequentially
# CNC Program Overview

## Sample CNC Milling Program

<table>
<thead>
<tr>
<th>Instructions</th>
<th>Comments</th>
</tr>
</thead>
<tbody>
<tr>
<td>%</td>
<td>Program Start flag</td>
</tr>
<tr>
<td><code>:1002</code></td>
<td>Program #1002</td>
</tr>
<tr>
<td>N5 G90 G20 G40 G17</td>
<td>Block #5, Abs in inches</td>
</tr>
<tr>
<td>N10 M06 T3</td>
<td>Block #10, Tool Change to Tool #3</td>
</tr>
<tr>
<td>N15 M03 S1250</td>
<td>Spindle on CW at 1250 RPM</td>
</tr>
<tr>
<td>N20 G00 X1.0 Y1.0</td>
<td>Rapid over to X1.0, Y1.0</td>
</tr>
<tr>
<td>N25 Z0.1</td>
<td>Rapid down to Z0.1</td>
</tr>
<tr>
<td>N30 G01 Z-0.125 F5</td>
<td>Feed down to Z-0.125 at 5ipm</td>
</tr>
<tr>
<td>N35 M30</td>
<td>Stop</td>
</tr>
</tbody>
</table>

- **G-codes are preparatory functions involve actual tool moves:**
  - Rapid moves, radial feed moves
- **M-codes are Miscellaneous functions:**
  - Spindle on/of tool changes, coolant on/of program stop

Refer page 74

Refer pages 72 - 73
Writing a CNC Program

Process

1. Decide on measuring units
   - Inches or Metric

2. Decide on Coordinate System to use
   - Absolute
   - Incremental

3. Call up the tool

4. Turn on the spindle

5. Move tool rapidly to a point close to part

6. Start Machining

7. System Shutdown

Phase 1

Phase 2

Phase 3
CNC Program
Phase 1: Program Set Up Instructions

Program start flag
0<= Program number <= 9999
Use Abs units and Inch programming
Stop for tool change, use Tool #2
Turn the spindle on CW to 1200rpm

% :1001
N5 G90 G20
N10 M06 T2
N15 M03 S1200
N20 G00 X1.00 Y1.00
N25 Z0.125
N30 G01 Z-0.125 F5.0
N35 G01 X2.0 Y2.0
N40 G00 Z1.0
N45 X0 Y0
N50 M05
N55 M30
CNC Program
Phase 2: Material Removal

% 1001
N5 G90 G20
N10 M06 T2
N15 M03 S1200
N20 G00 X1.00 Y1.00
N25 Z0.125
N30 G01 Z-0.125 F5.0
N35 G01 X2.0 Y2.0
N40 G00 Z1.0
N45 X0 Y0
N50 M05
N55 M30

Rapid to distance (1,1) from origin
Rapid down to Z.1 just above the part
Feed down to Z-0.125 inches at 5 ipm
Feed diagonally to X2, Y2
Rapid up to Z1 (clear the part)
Rapid back home to X0, Y0

G00 in N20 (G01 in N35) remains active until overridden by G01 in N30 (G00 in N40):

Modal G Codes
CNC Program

Phase 3: Shut Down

% :1001
N5 G90 G20
N10 M06 T2
N15 M03 S1200
N20 G00 X1.00 Y1.00
N25 Z0.125
N30 G01 Z-0.125 F5.0
N35 G01 X2.0 Y2.0
N40 G00 Z1.0
N45 X0 Y0

Turn the spindle off
End of program

N50 M05
N55 M30

End of program
CNC Program Block

Block Format

Let’s examine the block format
- Order of Codes, Components

Sample block of a CNC Code:

N135 G01 X1.0 Y1.0 Z0.125 F5.0

Order:
- Block number, G-code, Coordinate, Special Function

Components:
- N135 - Shows current CNC block number
- G01 – Command that instructs the machine to do something (linear feed move – See page 74)
- X1.0 Y1.0 Z0.125 – Specifies the endpoint to move the spindle
- F5.0 – Any special function or related parameter is included here (feed rate of 5 inch per minute)
Rules for CNC Program Block

Restrictions

- Only one tool move allowed per block
- May contain any number of non-tool move G-codes, provided there is no conflict between the codes
- Only one feed rate per block
- May contain only one specified tool or spindle speed
- Block number should be sequential
- Program start flag and program number must be independent of other commands
- Order of components must conform to format in slide 9
- Only one M-code allowed per block
Preparing to Program
Start with a Drawing

- Locate “Program Zero” (Reference Point, i.e., origin):
  - For Milling, Reference Point is at lower left hand corner on top of work piece (refer to chapter 2 lecture notes)
  - For lathe, Reference point is always the center of the path in X and right-hand end of finished work piece in z-direction (refer to Chapter 2 lectures)

- Draw the finished work piece:
  - Locate all coordinates and distances from reference point (absolute programming) on drawing

- Record coordinates on an X,Y, Z “Coordinate sheet”