Advanced Manufacturing Technology (ME 702)

Module-3 <NC/CNC>

Dr. C. Chatteraj
Contents

Introduction
Types of NC Machines
Components of a NC Machine
Control Mechanisms
Interpolation
Software Components
HISTORICAL DEVELOPMENT

• 15th century - machining metal.
• 18th century - industrialization, production-type machine tools.
• 20th century - F.W. Taylor - tool metal - HSS

  Automated production equipment -
  Screw machines
  Transfer lines
  Assembly lines
  using cams and preset stops

Programmable automation -
  NC
  PLC
  Robots
NEW NCs or CNCs

• high speed spindle (> 20,000 rpm)
• high feed rate drive ( > 600 ipm)
• high precision ( < 0.0001" accuracy)
NC MACHINES

- Computer control
- Servo axis control
- Tool changers
- Pallet changers
- On-machine programming
- Data communication
- Graphical interface
NC MACHINES

MCU - Machine control unit

CLU - Control-loops unit

DPU - Data processing unit
NC MOTION-CONTROL

NC Program

Execution System

Dimensions

Interpolator & Servo-control Mechanism

Control Logic

Linear Motion

Commands

Translator

Relay

Solenoid

Power
NC MACHINE CLASSIFICATIONS

1. Motion control:
   point to point (PTP)
   continuous (contouring) path

2. Control loops:
   open loop
   closed loop

3. Power drives:
   hydraulic, electric, or pneumatic
NC MACHINE CLASSIFICATIONS

4. Positioning systems:
   incremental
   absolute positioning

5. Hardwired NC and softwired
   Computer Numerical Control (CNC)
POINT TO POINT

- Moving at maximum rate from point to point.
- Accuracy of the destination is important but not the path.
- Drilling is a good application.
CONTINUOUS PATH

• Controls both the displacement and the velocity.
• Machining profiles.
• Precise control.
• Use linear and circular interpolators.
COMPONENTS OF AN NC MACHINE TOOL

- Controller
- Servo drive
- Position transducer
- Machine table
- Leadscrew
- Gear box
- Tachometer
- Motor
- Magnetics control cabinet
NC MACHINE RATING
/Specification

- Accuracy
- Repeatability
- Spindle and axis motor horsepower
- Number of controlled axes
- Dimension of workspace
- Features of the machine and the controller.
ACCURACY

• Accuracy = control instrumentation resolution and hardware accuracy.

• Control resolution: the minimum length distinguishable by the control unit (BLU).

• Hardware inaccuracies are caused by physical machine errors.
HARDWARE INACCURACIES

Component tolerances:

- inaccuracies in the machine elements,
- machine-tool assembly errors,
- spindle runout,
- and leadscrew backlash.

Machine operation:

- Tool deflection (a function of the cutting force), produces dimensional error and chatter marks on the finished part.
HARDWARE INACCURACIES

Thermal error:

heat generated by the motor operation, cutting process, friction on the ways and bearings, etc. Use cutting fluids, locating drive motors away from the center of a machine, and reducing friction from the ways and bearings
REPEATABILITY

Repeatability

Avg. error

Test result

Programmed position
LEADScrews

Converting the rotational motion of the motors to a linear motion.

Pitch (p): the distance between adjacent screw threads

the number of teeth per inch (n):

\[ n = \frac{1}{p} \]

BLU: Basic Length Unit (machine resolution)

\[ \text{BLU} = \frac{p}{N} \]
CONTROL LOOPS

Open loop - No position feedback.

Use stepping motor.
CONTROL LOOPS

• A machine has 1 BLU = 0.001". To move the table 5" on X axis at a speed (feed rate) of 6 ipm.

• pulse rate = speed/BLU = 6 ipm/0.001 ipp = 6,000 pulse/min

• pulse count = distance/BLU

= 5/0.001 = 5,000 pulses
CLOSED LOOP

Closed-loop control mechanism
INTERPOLATION

Control multiple axes simultaneously to move on a line, a circle, or a curve.

Point-to-point control path

\[ V_x = 6 \frac{(10-3)}{\sqrt{(10-3)^2 + (5-2)^2}} = 6 \frac{7}{\sqrt{49 + 9}} = 5.5149 \]

\[ V_y = 6 \frac{(5-2)}{\sqrt{(10-3)^2 + (5-2)^2}} = 6 \frac{3}{\sqrt{49 + 9}} = 2.3635 \]
INTERPOLATORS

- Most common: linear and circular
- Since interpolation is right above the servo level, speed is critical, and the process must not involve excessive computation.
- Traditional NC interpolators: Digital Differential Analyzer (DDA)
- Higher order curves, such as Bezier's curve, use off-line approximation algorithms to break the curves into linear or circular segments.
COORDINATE SYSTEMS

- Right hand rule
- Z axis align with the spindle - +Z moves away from the workpiece or the spindle.
- X axis - Lathe: perpendicular to the spindle.
- Horizontal machine: parallel to the table.
- Vertical machine: +X points to the right.
MACHINE COORDINATES

X - Primary Feed axis
Z - Spindle axis
Y - Remaining axis
PROGRAM STORAGE

• Paper tape
  Paper or Mylar coated paper.

• Diskettes

• From other computers through RS 232 or local area network (LAN)
SYMBOLIC CODES

• ASCII or ISO, use even parity
• EIA - Binary Coded Decimal (BCD), RS 244A standard, use odd parity.
INPUT FORMATS

• EIA RS-274 standard

• Fixed sequential format
  0010 01 07500 06250 00000 00000 612

• Tab sequential format
  T0010 T01 T07500 T06250 T T T612

• Word-address format
  N0010 G01 X07500 Y06250 S612
Agenda

• Introduction
• Absolute and Incremental Programming
• Elements of NC Program
• NC Words (G, M, T, S, … Codes)
• Examples
• Cutter Compensation and Offsets
• Examples
• Conclusions
Introduction to NC programming

• Manual part programming
• Computer-assisted part programming
• Formats
  – Fixed-Address
  – Tab-Sequential
  – Word-Address
Manual NC programming

- Absolute Programming
- Incremental Programming
Basic Elements of NC Program

- Blocks of Commands
- NC Words
- NC Function ~ NC word(s)
- Several Functions in one block
- When viewing, a block is same as a line of text
- Pre-defined Terminator
- Optional Blocks
NC WORDS

• A G-code program consists the following words:

• An EIA standard, RS-273 defines a set of standard codes.
Basic Elements of NC Program

a. Preparatory functions: which unit, which interpolator, absolute or incremental programming, which circular interpolation plane, cutter compensation, etc.

b. Coordinates: three translational, and three rotational axes.

c. Machining parameters: feed, and speed.

d. Tool control: tool diameter, next tool number, tool change.

e. Cycle functions: drill cycle, ream cycle, bore cycle, mill cycle, clearance plane.

f. Coolant control: coolant on/off, flood, mist.

g. Miscellaneous control: spindle on/off, tape rewind, spindle rotation direction, pallet change, clamps control, etc.

h. Interpolators: linear, circular interpolation
# NC WORDS – G codes

## G codes

<table>
<thead>
<tr>
<th>N code. sequence number example: N0010</th>
<th>G code. preparatory word.</th>
</tr>
</thead>
<tbody>
<tr>
<td>g00 Rapid traverse</td>
<td>g40 Cutter compensation - cancel</td>
</tr>
<tr>
<td>g01 Linear interpolation</td>
<td>g41 Cutter compensation - left</td>
</tr>
<tr>
<td>g02 Circular interpolation, CW</td>
<td>g42 Cutter compensation - right</td>
</tr>
<tr>
<td>g03 Circular interpolation, CCW</td>
<td>g70 Inch format</td>
</tr>
<tr>
<td>g04 Dwell</td>
<td>g71 Metric format</td>
</tr>
<tr>
<td>g08 Acceleration</td>
<td>g74 Full circle programming Off</td>
</tr>
<tr>
<td>g09 Deceleration</td>
<td>g75 Full circle programming On</td>
</tr>
<tr>
<td>g17 X-Y Plane</td>
<td>g80 Fixed cycle cancel</td>
</tr>
<tr>
<td>g18 Z-X Plane</td>
<td>g81 -9 Fixed cycles</td>
</tr>
<tr>
<td>g19 Y-Z Plane</td>
<td>g90 Absolute dimension programming</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 9.1 G codes</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>g00 Rapid traverse</td>
<td>g40 Cutter compensation - cancel</td>
</tr>
<tr>
<td>g01 Linear interpolation</td>
<td>g41 Cutter compensation - left</td>
</tr>
<tr>
<td>g02 Circular interpolation, CW</td>
<td>g42 Cutter compensation - right</td>
</tr>
<tr>
<td>g03 Circular interpolation, CCW</td>
<td>g70 Inch format</td>
</tr>
<tr>
<td>g04 Dwell</td>
<td>g71 Metric format</td>
</tr>
<tr>
<td>g08 Acceleration</td>
<td>g74 Full circle programming Off</td>
</tr>
<tr>
<td>g09 Deceleration</td>
<td>g75 Full circle programming On</td>
</tr>
<tr>
<td>g17 X-Y Plane</td>
<td>g80 Fixed cycle cancel</td>
</tr>
<tr>
<td>g18 Z-X Plane</td>
<td>g81 -9 Fixed cycles</td>
</tr>
<tr>
<td>g19 Y-Z Plane</td>
<td>g90 Absolute dimension programming</td>
</tr>
</tbody>
</table>
NC WORDS- BLU

• X, Y, Z, A, B, C Codes. coordinate positions of the tool.
  The coordinates may be specified in decimal number (Decimal Programming), or integer number (BLU Programming).

• BLU programming: leading zero, trailing zero.
  – In the leading zero format:
  • X00112 Y002275 Z001
  – In the trailing zero format, the program looks like:
  X11200 Y22750 Z10000
NC WORDS – Circular Interpolation

Circular Interpolation:

N0100 G02 X7.000 Y2.000 I5.000 J2.000

(5.000,4.000)

sequence no. ? destination center

(7.000,2.000)

N0100 G02 X7.000 Y2.000 I5.000 J2.000

Cut from (5.000,4.000) to (7.000,2.000) CW

(5.000,2.000)
NC WORDS- F and S Codes

- F Code. feed speed. inch/min (ipm), or ipr.

- F code must be given before either G01, G02, or G03 can be used.

- Example:
  N0100 G02 X7.000 Y2.000 I5.000 J2.000 F6.00

- S Code. cutting speed code. Programmed in rpm.

- S code **does not** turn on the spindle, spindle is turned on by a M code.

- N0010 S1000
NC WORDS- T and R Codes

- T Code. tool number.
  Actual tool change does not occur until a tool change M code is specified.

- R Code. cycle parameter.
  The cycle may be programmed in one block, such as (cycle programming is vendor specific.):

  N0010 G81 X1.000 Y2.000 Z0.000 R 1.300
## NC WORDS – M Codes

M Code. miscellaneous word.

### Table 9.2. M codes

<table>
<thead>
<tr>
<th>M Code</th>
<th>Description</th>
<th>M Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>m00</td>
<td>Program stop</td>
<td>m06</td>
<td>Tool change</td>
</tr>
<tr>
<td>m01</td>
<td>Optional stop</td>
<td>m07</td>
<td>Flood coolant on</td>
</tr>
<tr>
<td>m02</td>
<td>End of program</td>
<td>m08</td>
<td>Mist coolant on</td>
</tr>
<tr>
<td>m03</td>
<td>Spindle CW</td>
<td>m09</td>
<td>Coolant off</td>
</tr>
<tr>
<td>m04</td>
<td>Spindle CCW</td>
<td>m30</td>
<td>End of tape</td>
</tr>
</tbody>
</table>
Example 9.1
Machined from a 5" x 4" x 2" workpiece. low carbon steel.

The process plan:
1. Set the lower left bottom corner of the part as the machine zero point (floating zero programming).
2. Clamp the workpiece in a vise.
3. Mill the slot with a 3/4" four flute flat end mill made of carbide. From the machinability data handbook, the recommended feed is 0.005 inch/tooth/rev, and the recommended cutting speed is 620 fpm.
4. Drill two holes with a 0.75" dia twist drill. Use 0.18 ipr feed and 100 fpm speed.
PART DRAWING

All dimensions in inches. All tolerances ±0.001"
SOLUTION TO EXAMPLE

Solution:

The cutting parameters need be converted into rpm and ipm.

Milling: 
\[
RPM = \frac{12 \cdot V}{\pi D} = \frac{12 \times 620 \text{ fpm}}{\pi \times 0.75 \text{ inch}} = 3,157 \text{ rpm}
\]

Drilling:
\[
RPM = \frac{12 \cdot V}{\pi D} = \frac{12 \times 100 \text{ fpm}}{\pi \times 0.75 \text{ inch}} = 509 \text{ rpm}
\]

\[
V_f = f \cdot RPM = 0.018 \text{ ipr} \times 509 \text{ rpm} = 9.16 \text{ ipm}
\]
CUTTER LOCATIONS

The coordinates of each point (cutter location) are calculated below:

\[
\begin{align*}
p1': (1.75+0.375, -0.1-0.375, 4.00) &= (2.125, -0.475, 4.00) \\
p1: (2.125, -0.475, 2.000-0.500) &= (2.125, -0.475, 1.500) \\
p2: (2.125, 4.000+0.100, 1.500) &= (2.125, 4.100, 1.500) \\
p3: (3.000-0.375, 4.100, 1.500) &= (2.625, 4.100, 1.500) \\
p4: (2.625, 1.375, 1.500) \\
p5: (3.000, 2.000-1.000+0.375, 1.500) &= (3.000, 1.375, 1.500) \\
p6: (3.000, 2.625, 1.500) \\
p7: (3.000, 2.000, 1.500) \\
p8: (2.625, 2.000, 1.500) \\
p9: (2.625, -0.100, 1.500) \\
p9': (2.625, -0.100, 4.000)
\end{align*}
\]
# PART PROGRAM

<table>
<thead>
<tr>
<th>Part program</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>N0010 G70 G 90 T08 M06</td>
<td>Set the machine to inch format and absolute dimension programming.</td>
</tr>
<tr>
<td>N0020 G00 X2.125 Y-0.475 Z4.000 S3157 Rapid to p1'.</td>
<td></td>
</tr>
<tr>
<td>N0030 G01 Z1.500 F63 M03</td>
<td>Down feed to p1, spindle CW.</td>
</tr>
<tr>
<td>N0040 G01 Y4.100</td>
<td>Feed to p2.</td>
</tr>
<tr>
<td>N0050 G01 X2.625</td>
<td>To p3.</td>
</tr>
<tr>
<td>N0060 G01 Y1.375</td>
<td>To p4.</td>
</tr>
</tbody>
</table>
## PART PROGRAM

<table>
<thead>
<tr>
<th>Part program</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>N0070 G01 X3.000</td>
<td>To p5.</td>
</tr>
<tr>
<td>N0080 G03 Y2.625 I3.000 J2.000</td>
<td>Circular interpolation to p6.</td>
</tr>
<tr>
<td>N0090 G01 Y2.000</td>
<td>To p7.</td>
</tr>
<tr>
<td>N0100 G01 X2.625</td>
<td>To p8.</td>
</tr>
<tr>
<td>N0110 G01 Y-0.100</td>
<td>To p9.</td>
</tr>
<tr>
<td>N0120 G00 Z4.000 T02 M05</td>
<td>To p9', spindle off, tool #2.</td>
</tr>
<tr>
<td>N0130 F9.16 S509 M06</td>
<td>Tool change, set new feed speed.</td>
</tr>
<tr>
<td>and</td>
<td></td>
</tr>
<tr>
<td>N0140 G81 X0.750 Y1.000 Z-0.1 R2.100 M03</td>
<td>Drill hole 1.</td>
</tr>
<tr>
<td>N0150 G81 X0.750 Y3.000 Z-0.1 R2.100</td>
<td>Drill hole 2.</td>
</tr>
<tr>
<td>N0160 G00 X-1.000 Y-1.000 M30</td>
<td>Move to home position, stop the machine.</td>
</tr>
</tbody>
</table>
CNCS VERIFICATION

Machining Time: 1.62 min
CNCS 3D DRAWING

Machining Time: 1.62 min

N0100 G01
G1 X2.62
N0110 G01
G1 Y-0.1
N0120 G00
G0 Z4.00
N0130 F9.16E1
N0140 G81
G81 X0.7
N0150 G81
G81 X0.7
Job compl
Offsets

• Fixture
  – G10, G54, G54.1

• Diameter

• Tool
  – Length compensation
  – Part-Edge compensation

• Cutter Compensation – Next Slides

• Others Discussed in Lab Exercises (Simulators)
Tool Radius Compensation

• Cutter Compensation
  Shifting tool path so that the actual finished cut is either moved to the left or right of the programmed path.

• Normally, shifted exactly by tool radius

• Tool Entry and Exit Issues
Tool Radius Compensation

Start of Compensation.

G41 (or G42) and G01 in the same block ramp takes place at block N0010.

N0010 G01 G42 X0.500 Y1.700
N0020 G01 X1.500

(a) G41                                                  (b) G42
G41
G42

G41 (or G42) and G01 in separate blocks the compensation is effective from the start.

N0010 G41
N0020 G01 X0.500 Y1.700
N0030 G01 X1.500
Tool Radius Compensation

Inside Corner.

Cutter path is inside a corner, stops at the inside cutting point

N0010 G41
N0020 G01 X1.500 Y2.000
N0030 G01 X0.000 Y1.600

Use of M96 and M97.

Cutting tool that is larger than the height of the step, M97 must be used

N0010 G41
N0020 G01 X1.000 Y1.000
N0030 G01 Y0.800 M97
N0040 G01 X2.000
TOOL-RADIUS COMPENSATION

Cancel Tool Compensation.

G40 in the same block ramp off block.

N0060 G40 X2.000 Y1.700 M02

G40 in a block following the last motion, the compensation is effective to the end point (2.000,1.700).

N0060 X2.000 Y1.700

N0070 G40 M02
EXAMPLE

A square 2.0 in. x 2.0 in. is to be milled using a 1/2 in. end milling cutter. Write an NC part program to make the square.

Solution

Let us set up the lower left corner of the square at (6.0,6.0). Using tool-radius compensation, the square can be produced.
# PART PROGRAM

<table>
<thead>
<tr>
<th>Part Program</th>
<th>Explanation</th>
</tr>
</thead>
<tbody>
<tr>
<td>N0010 G41 S1000 F5</td>
<td>Begin compensation, set feed and speed, spindle on</td>
</tr>
<tr>
<td>N0020 G00 X6.000 Y</td>
<td>Move to lower left corner</td>
</tr>
<tr>
<td>N0030 G01 Z-1.000</td>
<td>Plunge down the tool</td>
</tr>
<tr>
<td>N0040 Y8.000</td>
<td>Cut to upper left corner</td>
</tr>
<tr>
<td>N0050 X8.000</td>
<td>Cut to upper right corner</td>
</tr>
<tr>
<td>N0060 Y6.000</td>
<td>Cut to lower right corner</td>
</tr>
<tr>
<td>N0070 X6.000</td>
<td>Cut to lower left corner</td>
</tr>
<tr>
<td>N0080 Z1.000</td>
<td>Lift the tool</td>
</tr>
<tr>
<td>N0090 G40 M30</td>
<td>End compensation, stop the machine</td>
</tr>
</tbody>
</table>
TURNING

Part design

Cutter path

Tool
TURNING

Programming tool point

No compensation needed.

Surfaces cut

IMAGINARY TOOL POINT

Programmed tool path

Surface created

58
COMPUTER ASSISTED PART PROGRAMMING

Machine-oriented languages - machine specific

General-purpose languages - use post processors to generate
  machine specific code

Translate input symbols

Arithmetic calculation

Cutter offset calculations

Post processing

Part program

Language Processor

CL data

Post Processor

N-G code

CL
BCL
RS-494

RS-273 59
Advantages

• NC can reduce machining skill
• NC can reduce the time required to machine a part
• NC provides sophisticated contour capability
• NC is a flexible method for manufacture of sophisticated machined components