CNC PROGRAMMING

93

CNC Programming

- Manual
  - Write code directly
- Computer-assisted
  - Draw cutter path
- CAD/CAM
  - Draw the part
  - Cutter path is generated

CNC PROGRAMMING

- **Offline programming** linked to CAD programs.
- **Conversational programming** by the operator.
- **MDI** ~ Manual Data Input.
- **Manual Control** using jog buttons or `electronic handwheel'.
- **Word-Address Coding** using standard G-codes and M-codes.

95

Basics of NC Part Programming:

During secondary motion, either the tool moves relative to the workpiece or the workpiece moves relative to the tool. In NC programming, it is always assumed that the tool moves relative to the workpiece no matter what the real situation is.

96

The position of the tool is described by using a Cartesian coordinate system. If (0,0,0) position can be described by the operator, then it is called *floating zero*.

97

In defining the motion of the tool from one point to another, either *absolute positioning* mode or *incremental positioning* mode can be used.

98

1. *Absolute positioning*. In this mode, the desired target position of the tool for a particular move is given relative to the origin point of the program.

2. *Incremental positioning*. In this mode, the next target position for the tool is given relative to the current tool position.

99

Structure of an NC Part Program:

Commands are input into the controller in units called *blocks* or *statements*.

Block Format:

1. Fixed sequential format
2. Tab sequential format
3. Word address format

100

EXAMPLE:

Assume that a drilling operation is to be programmed as:

1. The tool is positioned at (25.4,12.5,0) by a rapid movement.
2. The tool is then advanced -10 mm in the z direction at a feed rate of 500 mm/min., with the flood coolant on.
3.The is then retracted back 10 mm at the rapid feed rate, and the coolant is turned off.

101

1. Fixed sequential format

```
0050 00 +0025400 +0012500 +0000000 0000 00
0060 01 +0025400 +0012500 -0010000 0500 08
0070 00 +0025400 +0012500 +0000000 0000 09
```

2. Tab sequential format

```
0050 TAB 00 TAB +0025400 TAB +0012500 TAB +0000000 TAB TAB
0060 TAB 01 TAB TAB TAB -0010000 TAB 0500 TAB 08
0070 TAB 00 TAB TAB TAB -0000000 TAB 0000 TAB 09
```

3. Word address format

```
N50 G00 X25400 Y125 Z0 F0
N60 G01 Z-10000 F500 M08
N70 G00 Z0 M09
```

102

Modal commands: Commands issued in the NC program that will stay in effect until it is changed by some other command, like, feed rate selection, coolant selection, etc.

Nonmodal commands: Commands that are effective only when issued and whose effects are lost for subsequent commands, like, a dwell command which instructs the tool to remain in a given configuration for a given amount of time.

103

INFORMATION NEEDED by a CNC

1. Preparatory Information: units, incremental or absolute positioning
2. Coordinates: X,Y,Z, RX,RY,RZ
3. Machining Parameters: Feed rate and spindle speed
4. Coolant Control: On/Off, Flood, Mist
5. Tool Control: Tool and tool parameters
6. Cycle Functions: Type of action required
7. Miscellaneous Control: Spindle on/off, direction of rotation, stops for part movement

This information is conveyed to the machine through a set of instructions arranged in a desired sequence – Program.

104

BLOCK FORMAT

Sample Block

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

- Restrictions on CNC blocks
- Each may contain only one tool move
- Each may contain any number of non-tool move G-codes
- Each may contain only one feedrate
- Each may contain only one specified tool or spindle speed
- The block numbers should be sequential
- Both the program start flag and the program number must be independent of all other commands (on separate lines)
- The data within a block should follow the sequence shown in the above sample block

105

WORD-ADDRESS CODING

Example CNC Program

- N5 G90 G20
- N10 M06 T3
- N15 M03 S1250
- N20 G00 X1 Y1
- N25 Z0.1
- N30 G01 Z-0.125 F5
- N35 X3 Y2 F10
- N40 G00 Z1
- N45 X0 Y0
- N50 M05
- N55 M30

Each instruction to the machine consists of a letter followed by a number.

Each letter is associated with a specific type of action or piece of information needed by the machine.

Letters used in Codes

N,G,X,Y,Z,A,B,C,I,J,K,F,S,T,R,M

106

G & M Codes

Example CNC Program

- N5 G90 G20
- N10 M06 T3
- N15 M03 S1250
- N20 G00 X1 Y1
- N25 Z0.1
- N30 G01 Z-0.125 F5
- N35 X3 Y2 F10
- N40 G00 Z1
- N45 X0 Y0
- N50 M05
- N55 M30

- **G-codes**: Preparatory Functions involve actual tool moves.
- **M-codes**: Miscellaneous Functions – involve actions necessary for machining (i.e. spindle on/off, coolant on/off).

107

G Codes

- **G00 Rapid traverse**
- **G01 Linear interpolation**
- **G02 Circular interpolation, CW**
- **G03 Circular interpolation, CCW**
- **G04 Dwell**
- **G08 Acceleration**
- **G09 Deceleration**
- **G17 X-Y Plane**
- **G18 Z-X Plane**
- **G19 Y-Z Plane**
- **G20 Inch Units (G70)**
- **G21 Metric Units (G71)**

- G40 Cutter compensation – cancel
- G41 Cutter compensation – left
- G42 Cutter compensation- right
- G70 Inch format
- G71 Metric format
- G74 Full-circle programming off
- G75 Full-circle programming on
- G80 Fixed-cycle cancel
- G81-G89 Fixed cycles
- G90 Absolute dimensions
- G91 Incremental dimensions

108

Modal G-Codes

- Most G-codes set the machine in a “mode” which stays in effect until it is changed or cancelled by another G-code. These commands are called “modal”.

109

Modal G-Code List

- G00 Rapid Transverse
- G01 Linear Interpolation
- G02 Circular Interpolation, CW
- G03 Circular Interpolation, CCW
- G17 XY Plane
- G18 XZ Plane
- G19 YZ Plane
- G20/G70 Inch units
- G21/G71 Metric Units
- G40 Cutter compensation cancel
- G41 Cutter compensation left
- G42 Cutter compensation right
- G43 Tool length compensation (plus)
- G43 Tool length compensation (plus)
- G44 Tool length compensation (minus)
- G49 Tool length compensation cancel
- G80 Cancel canned cycles
- G81 Drilling cycle
- G82 Counter boring cycle
- G83 Deep hole drilling cycle
- G90 Absolute positioning
- G91 Incremental positioning

110

M Codes

- M00 Program stop
- M01 Optional program stop
- M02 Program end
- M03 Spindle on clockwise
- M04 Spindle on counterclockwise
- M05 Spindle stop
- M06 Tool change
- M08 Coolant on
- M09 Coolant off
- M10 Clamps on
- M11 Clamps off
- M30 Program stop, reset to start

111

N Codes

- Gives an identifying number for each block of information.
- It is generally good practice to increment each block number by 5 or 10 to allow additional blocks to be inserted if future changes are required.

112

X,Y, and Z Codes

- **X, Y, and Z** codes are used to specify the coordinate axis.
- Number following the code defines the coordinate at the end of the move relative to an incremental or absolute reference point.

113

I,J, and K Codes

- **I, J, and K** codes are used to specify the coordinate axis when defining the center of a circle.
- Number following the code defines the respective coordinate for the center of the circle.

114

F,S, and T Codes

- **F-code**: used to specify the feed rate
- **S-code**: used to specify the spindle speed
- **T-code**: used to specify the tool identification number associated with the tool to be used in subsequent operations.

115

Application of Some Codes
G01 Linear Interpolation

Format: N_ G01 X_ Y_ Z_ F_

- Linear Interpolation results in a straight line feed move.
- Unless tool compensation is used, the coordinates are associated with the centerline of the tool.

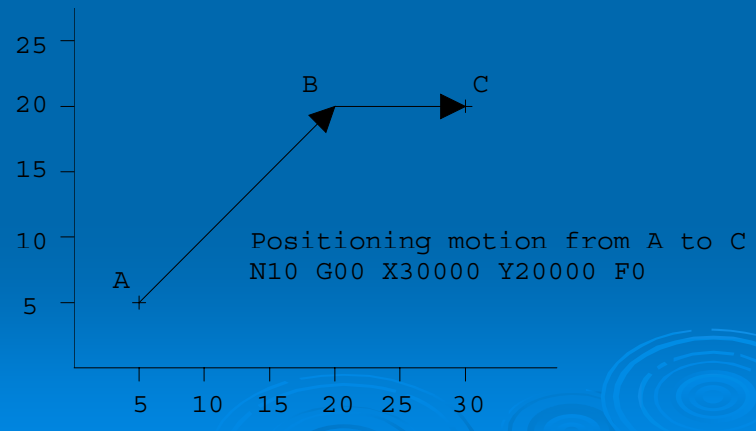
116

Application of Some Codes
G01 Linear Interpolation

- . As an example, for the motion that occurs in x-y plane with the same maximum speed for the x- and y-axis, initial motion is at an angle of 45o to the axes until motion in one of
- the axes is completed and then the balance of the motion occurs in the other axis. This is called *point-to-point motion*.

117

Application of Some Codes
G01 Linear Interpolation



118

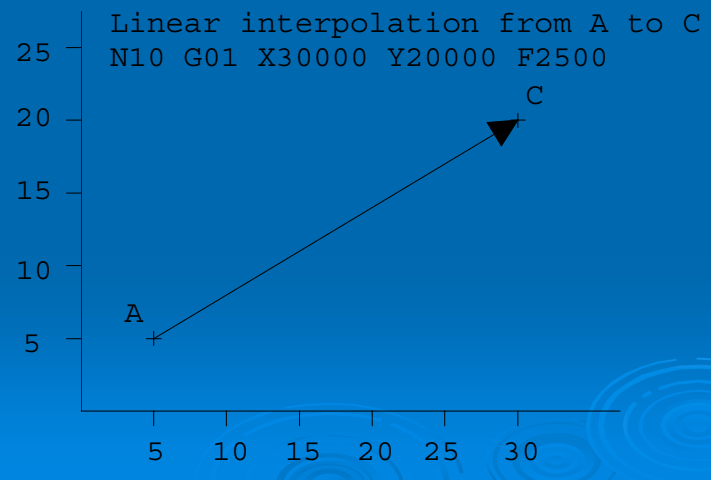
Application of Some Codes
G01 Linear Interpolation

G01 is another preparatory function to specify that the tool should be moved to a specified location along a straight line path. It is referred to as *linear interpolation*.

This function is typically used to specify machining of straight features such as turning a cylindrical surface in turning, cutting a slot in milling, etc.

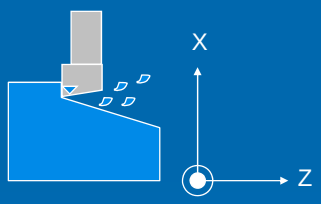
119

Application of Some Codes
G01 Linear Interpolation



120

G01 Linear Interpolation



N10 G00 X1 Z1
N15 Z0.1
N20 G01 Z-0.125 F5
N25 X2 Z2 F10

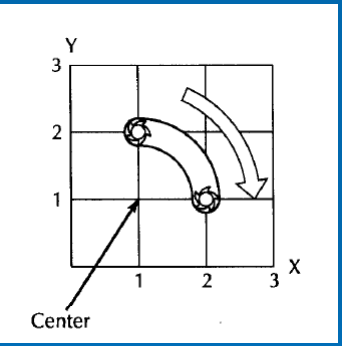
121

G02 Circular Interpolation

- G02 is also a preparatory function to specify that the tool should be moved to a specified location along a circular path in a clockwise direction. In order to specify the path to the MCU, the end point of the arc and the location of the center of the arc should be specified. Within the block in which the G02 code is programmed, the center of the arc is given by specifying its location relative to the start of the arc.

122

G02 Circular Interpolation (CW)



- The G02 command requires an endpoint and a radius in order to cut the arc.
- I,J, and K are relative to the start point.

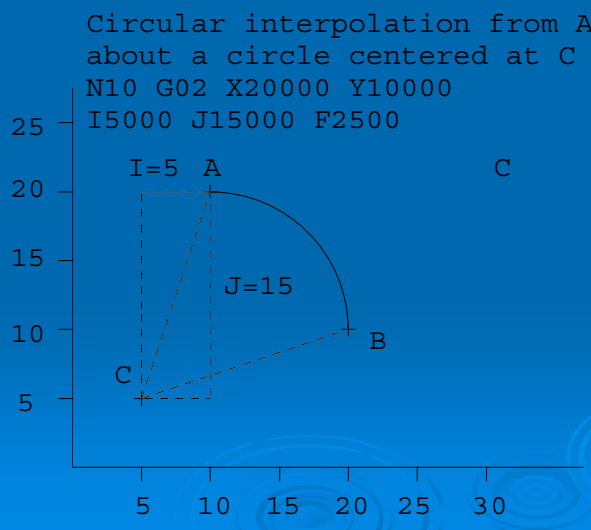
N_ G02 X2 Y1 I0 J-1 F10

or

N_ G02 X2 Y1 R1

123

G02 Circular Interpolation (CW)



124

Canned Cycles

The sequence of some machining operations is may be the same for any part and for any machine. For example, drilling a hole involves the following steps:

Position the tool above the point where the hole will be drilled

Set the correct spindle speed

Feed the tool into the workpiece at a controlled feed rate to a predetermined depth

Retract the tool at a rapid rate to just above the point where the hole started

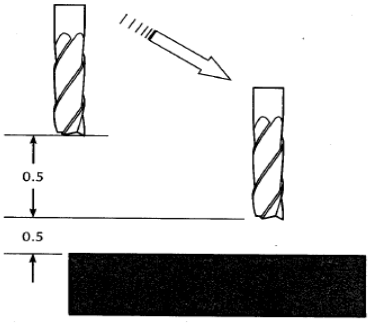
125

Some Commonly Used Canned Cycle				
Code	***Function***	***Down feed***	***At bottom***	***Retraction***
G81	Drilling	Continuous feed	No action	Rapid
G82	Spot face, counterbore	Continuous feed	Dwell	Rapid
G83	Deep hole drilling	Peck	No action	Rapid
G84	Tapping	Continuous feed	Reverse spindle	Feed rate
G85	Through boring(in & out)	Continuous feed	No action	Feed rate
G86	Through boring(in only)	Continuous feed	Stop spindle	Rapid

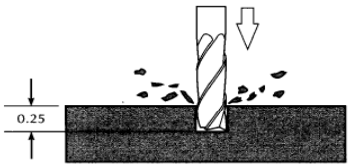
126

G81 ILLUSTRATION

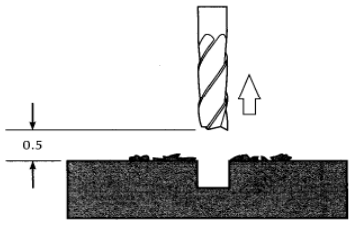
```
N5 G00 X0 Y0 Z1
N10 X1 Y1 Z0.5
N15 G81 Z-0.25 R0.125 F5
```



(a) The first move establishes the Z initial plane. The Z initial plane is the height the tool is at before the G81 cycle (0.5 in.) It is a good idea to move the tool an intermediate distance from the part so that less time is wasted rapiding in and out. The tool will rapid back to the initial plane after each hole is drilled.



(b) The tool rapids from the Z initial plane to the retract plane (0.125 in.). Starting at the Z retract plane (0.125 in.) the tool feeds down to the Z depth (–0.25 in.) at the specified feedrate.



(c) The tool then rapids up to the Z initial plane (0.5 in.). At this point, it would go on to the next hole.

127

Three Main parts of a CNC program

Part 1- Program Petup

- N5 G90 G21 (Absolute units, metric)
- N10 M06 T2 (Stop for tool change, use tool # 2)
- N15 M03 S1200 (Turn the spindle on CW to 1200 rpm)

128

Three Main parts of a CNC program

Part 2- Chip Removal

- N20 G00 X1 Y1 (Rapid to X1,Y1 from origin point)
- N25 Z0.125 (Rapid down to Z0.125)
- N30 G01 Z-0.125 F100 (Feed down to Z-0.125 at 100 mm/min)
- N35 G01 X2 Y2 (Feed diagonally to X2,Y2)
- N40 G00 Z1 (Rapid up to Z1)
- N45 X0 Y0 (Rapid to X0,Y0)

129

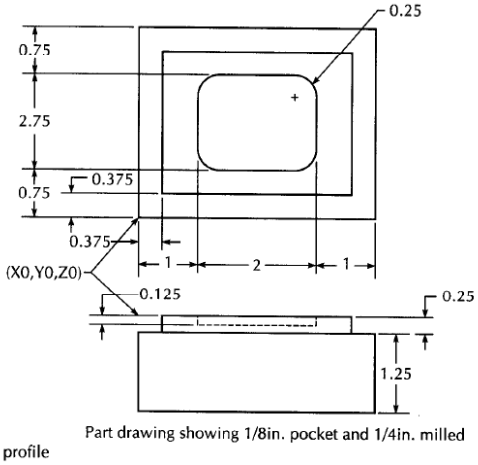
Three Main parts of a CNC program

Part 3- System Shutdown

- N50 M05 (Turn the spindle off)
- N55 M00 (Program stop)

130

EXAMPLE OPERATION on CNC MILLING MACHINE



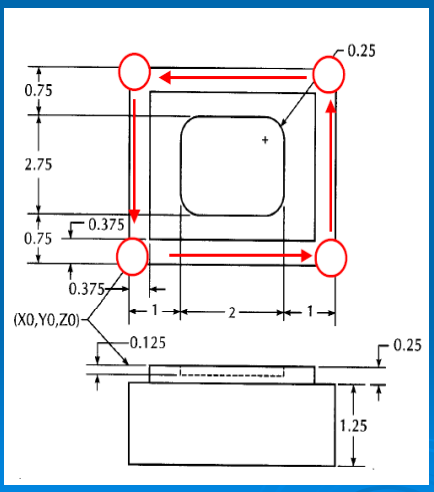
Part drawing showing 1/8in. pocket and 1/4in. milled profile

Material: 1020 St'l

Units: Inches

131

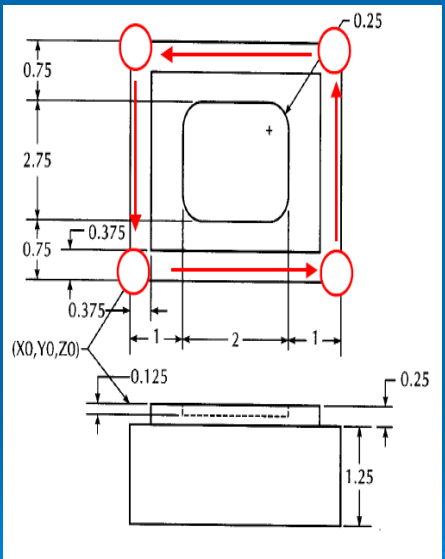
G-CODE PROGRAM



- First pass : conventional mill to a depth of 0.125 around edge profile. Tool 1 is a ½ inch dia. end mill.

```
%
:1002
N5 G90 G20
N10 M06 T1
N15 M03 S1200
N20 G00 X0.125 Y0.125
N30 Z0.125
N35 G01 Z-0.125 F5
N40 X3.875
N45 Y4.125
N50 X0.125
N55 Y0.125
```

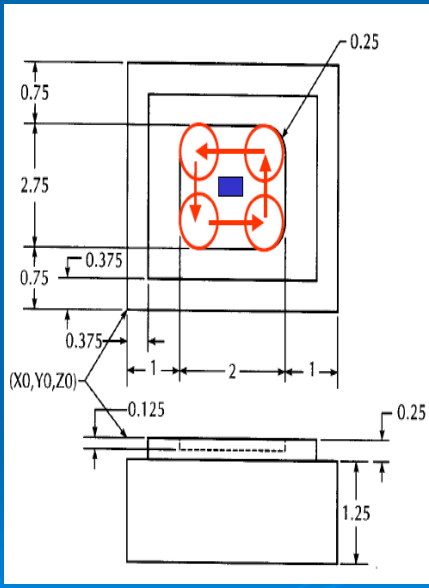
132



- Second pass: conventional mill to a depth of 0.25 around edge profile.

N35 Z-0.250
N40 X3.875
N45 Y4.125
N50 X0.125
N55 Y0.125
N60 Z0.125

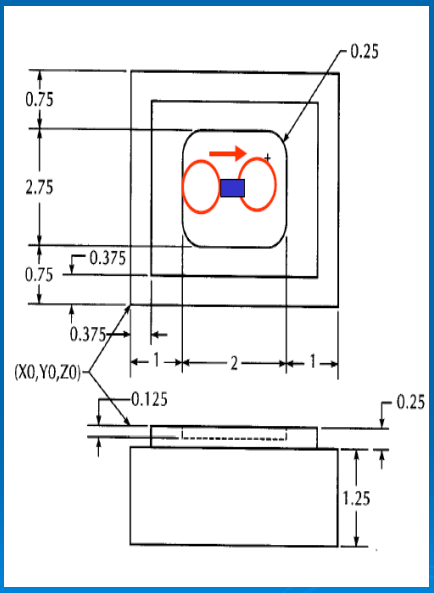
133



- Third pass: conventional mill to a depth of 0.125 around pocket profile.

N65 G00 X1.25 Y1.0
N70 G01 Z-0.125 F5
N75 X1.75
N80 Y2.5
N85 X1.25
N90 Y1.0
N95 Z0.125

134



- Fourth pass: climb mill to a depth of 0.125 across remaining material.

N100 Y2.125
N105 X2.625
N110 Z0.125
N115 G00 X-5 Y-5 Z5
N120 M05
N125 M30

135

Advanced features:

- Execution of the part of the program in a rotated or mirrored position.
- Ability to scale the program and produce larger or smaller programs.
- Three dimensional circular interpolation which produces a helical shape.
- Parabolic and cubic interpolation.

136

Program Loading:

- Through keyboard
- Through punched tape reader
- Through diskette drive
- Through RS 232 serial port
- Through network interface card

137

Direct Numerical Control (DNC):

- A system in which a central computer downloads the NC programs block by block to many NC machine tools simultaneously is called *Direct Numerical Control* (*DNC*) system.

138

Direct Numerical Control (DNC):

- This system used to work with the early NC machine tools which can not read more than a block of information at a time. The central computer feed the program information one block at a time. When the machine execute the information, the next block of information would be fed.

139

Distributed Numerical Control (DNC):

- Distributed NC is known by the same acronym as Direct Numerical Control (DNC). After the introduction of CNC, the machine tools have had the capability of storing large amount of information. Therefore, there have been no need to have drip feed information system, like, Direct Numerical Control. Instead, Distributed Numerical Control is introduced. In such a system, a host computer communicate with many CNC machine tools via networks and download or upload programs.

140

Distributed Numerical Control (DNC):

- With Distributed Numerical Control systems, it is possible to monitor the activities in individual CNC machine tools on host computer.
- Therefore, better shop floor control can be achieved.

141

Computer Aided Part Programming:

- NC program preparation may be tedious and difficult if the part to be machined has a complex geometry. The main difficulty is to find out the cutter locations during the machining. Computers may be used to assist the programmers in preparing the NC codes.

142

Advantages of applying computer-aided part programming include the following:

- 1. It reduces the manual calculations involves in determining the geometric characteristics of the part.
- It provides the cutter path simulation.
- It provides tool collision checking.
- It shortens the program preparation time.
- It makes the program preparation easier.

143

- The Aerospace Industries Association sponsored the work that led to the first part programming language, developed in MIT in 1955.
- This was called: *Automatically Programmed Tools* (APT).
- APT is an English like simple programming language which basically produce the *Cutter Location* (CL) data.
- Using the cutter location data, the program can generate the actual NC codes by using a postprocessor .

144

CAD/CAM Based Part Programming:

- The output of any CAD package include the geometric data of the part to be machined. Therefore, many CAD/CAM package can produce cutter location (CL) data to be used for NC code generation.
- There is still to be a process planning module for a workable NC code generation.
- Some of the CAD/CAM packages that have the NC code generation capabilities are Computervision, CATIA, CADAM, ProEngineer, MechanicalDesktop (Auto Desk).

145