

Lecture Objectives



- At the end of the lecture, you will be able to:
 - Understand the meaning of important terminology connected with word address programming
 - Explain what comprise a program
 - Understand the meaning of a fixed or canned cycle
 - Identify the different codes of information required to program a fixed cycle
 - Describe the general sequence of operations to be followed in programming hole operations

Terminology



- Programming character – is an alphanumeric character or punctuation mark: N G ;
- Address – is a letter that describes the meaning of the numerical value that follows the address: G00, G90, X-12
- Words – are built by characters and are composed of two main parts: an address followed by a number in order to describe machine motions and dimensions in a program.
- B lock – is a complete line of information to the CNC machine. Comprise of a word or an arrangement of words

Terminology



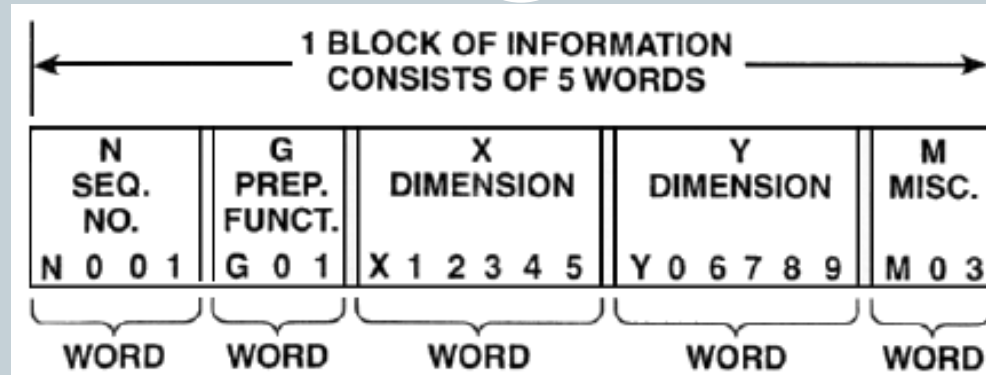
- Thus coming up with the



PROGRAM

where the MCU execute it block by block

Block of Information



- CNC information is generally programmed in blocks of five words. Each word conforms to the EIA standards and they are written on a horizontal line. If five complete words are not included in each block, the machine control unit (MCU) will not recognize the information, therefore the control unit will not be activated. Using the example shown in Fig. 17, the five words are as follows:
- N001 represents the sequence number of the operation.
- G01 represents linear interpolation
- X12345 will move the table 1.2345 in. in a positive direction along the X axis.
- Y06789 will move the table 0.6789 in. along the Y axis.
- M03 Spindle on CW.

Program codes



- Program Number (O)
- Sequence Number (N)
- Preparatory Function (G)
 - Modal vs Nonmodal
 - Standardize vs Specialized
 - Machining Center vs Lathe

- Dimension Words (X,Y,Z)
- Feedrate (F)
- Spindle Speed (S)
- Miscellaneous Machine Functions (M)
- Automatic Tool Changing (M6)
- Tool Length Offset & Cutter Radius Compensation (H, D)
- Comments (L&R parenthesis)

Modal/Non Modal Command



- Certain command remain active until cancelled by another codes – Modal Command
- e.g. G00, G01 and Canned Cycle
- A Non Modal Command is only active in the block in which it is issued
- e.g. M00, M01

Canned Cycle



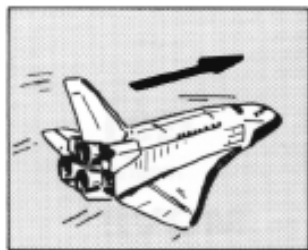
Canned Cycle such as G81 are routine, built into the control to perform standard operation.

e.g. drilling, boring and tapping – G82, G83, G84, G65

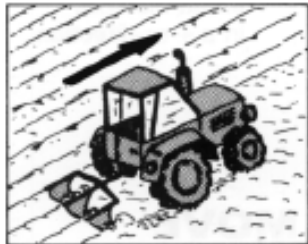
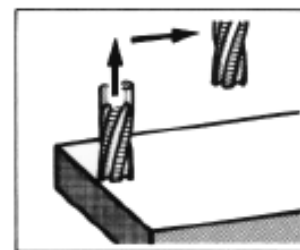
Each coordinate written will cause the spindle to repeat the cycle

Canned Cycle must be cancelled after each operation
G80

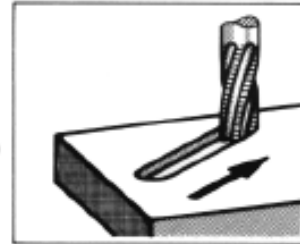
Functions of a Few Common G-Codes



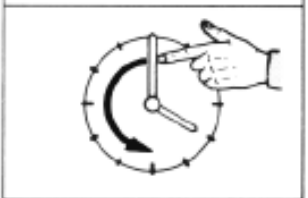
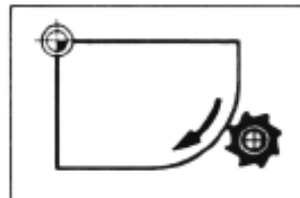
G00
RAPID TRAVERSE



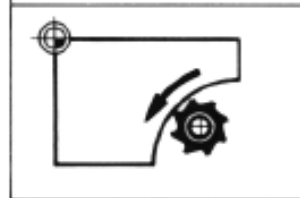
G01
LINEAR INTERPOLATION
(STRAIGHT LINE MOVEMENT)



G02
CIRCULAR INTERPOLATION
(CLOCKWISE)



G03
CIRCULAR INTERPOLATION
(COUNTERCLOCKWISE)



Common G-codes Used in CNC Programming



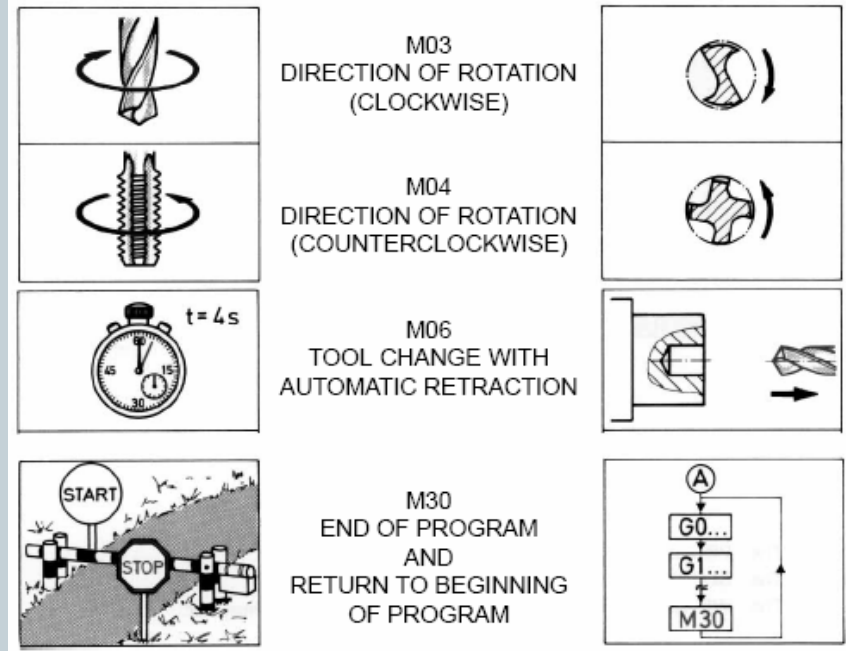
<u>Group</u>	<u>Code</u>	<u>Function</u>
01	G00	Rapid positioning
01	G01	Linear interpolation
01	G02	Circular interpolation clockwise (CW)
01	G03	Circular interpolation counterclockwise (CCW)
06	G20*	Inch input (in.)
06	G21*	Metric input (mm)
	G24	Radius programming (**)
00	G28	Return to reference point
00	G29	Return from reference point
	G32	Thread cutting (**)
07	G40	Cutter compensation cancel
07	G41	Cutter compensation left
07	G42	Cutter compensation right
08	G43	Tool length compensation positive (+) direction
08	G44	Tool length compensation minus (-) direction
08	G49	Tool length compensation cancel
	G84	Canned turning cycle (**)
03	G90	Absolute programming
03	G91	Incremental programming

(*) - on some machines and controls, these may be G70 (inch) and G71 (metric)

(**) - refers only to CNC lathes and turning centers.

Functions of a Few Common M-Codes

- M or miscellaneous codes are used to either turn ON or OFF different functions which control certain machine tool operations
- M-codes are not grouped into categories, although several codes may control the same type of operations such as M03, M04, and M05 which control the machine tool spindle.



Common M-codes used in CNC programming







<u>Code</u>	<u>Function</u>
M00	Program stop
M02	End of program
M03	Spindle start (forward CW)
M04	Spindle start (reverse CCW)
M05	Spindle stop
M06	Tool change
M08	Coolant on
M09	Coolant off
M10	Chuck - clamping (**)
M11	Chuck - unclamping (**)
M12	Tailstock spindle out (**)
M13	Tailstock spindle in (**)
M17	Toolpost rotation normal (**)
M18	Toolpost rotation reverse (**)
M30	End of tape and rewind
M98	Transfer to subprogram
M99	End of subprogram

(**) - refers only to CNC lathes and turning centers.

Example of CNC Programming for Milling

CNC Programming Hints - MILLING

-  Machine reference point (maximum travel of machine)
-  Machine X Y zero point (could be tool change point)
-  Part X Y zero point (programming start point)
-  Indicates the tool change position. A G92 code will reset the axis register position coordinates to this position.

For a program to run on a machine, it must contain the following codes:

- M03 To start the spindle/cutter revolving.
- Sxxx The spindle speed code to set the r/min.
- Fxx The feed rate code to move the cutting tool or workpiece to the desired position.

ANGLES:

The X Y coordinates of the start point and end point of the angular surface plus a feed rate (F) are required.

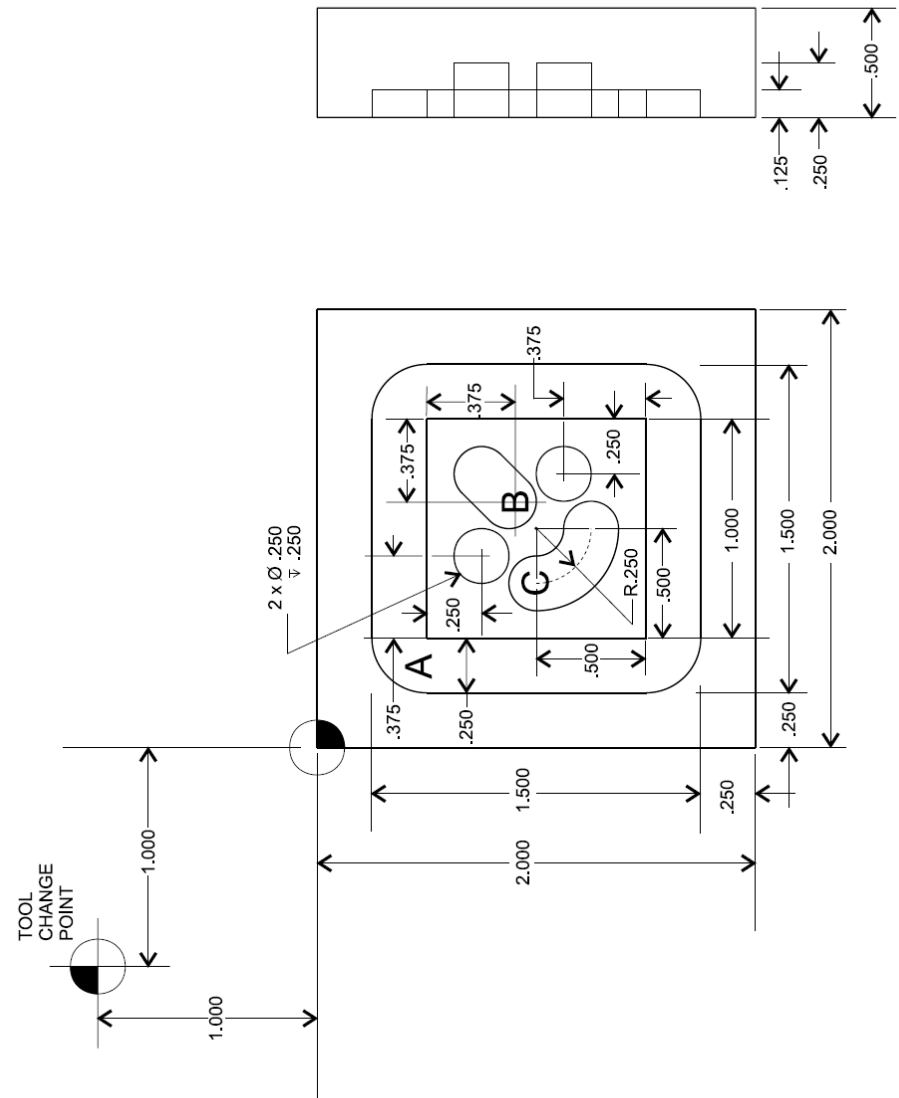
Z CODES:

- A Z dimension raises the cutter above the work surface.
- A Z- dimension feeds the cutter into the work surface.
- Z.100 is the recommended retract distance above the work surface before a rapid move (G00) is made to another location.

RADII / CONTOUR Requirements:

- The *start* point of the arc (XY coordinates)
- The *direction* of cutter travel (G02 or G03)
- The *end* point of the arc (XY coordinates)
- The *center* point of the arc (IJ coordinates) or the arc radius)

Fig 26: A sample flat part used for CNC programming and machining



Program Notes: (Fig. 26)

- Program in the absolute mode starting at the tool change position at the top left corner of the print.
- The material is aluminum (300 CS), feedrate 10 in./min.
- The cutting tool is a .250 in. diameter high speed steel 2-flute end mill.
- Mill the 1 in. square slot.
- Drill the two .250 in. diameter holes, .250 in. deep.
- Mill the .250 in. wide angular slot, .125 in. deep.
- Mill the .250 in. wide circular groove, .125 in. deep.
- After the job is completed, return to the tool change position.

Programming:

% (rewind stop code / parity check)

2000 (program number)

N5 G92 X-1.000 Y1.000 Z1.000

G92 programmed offset of reference point (tool change position)
X-1.000 tool set at 1.000 to the left of the part.
Y1.000 tool set at 1.000 above the top edge of the part.
Z1.000 the end of the cutter is 1.000 above the top surface of the part.

N10 G20 G90

G20 inch data input.
G90 absolute programming mode.

N15 M06 T01

M06 tool change command.
T01 tool no. 1 (.250 diameter, 2-flute end mill).

N20 S2000 M03

S2000 spindle speed set at 2000 r/min.
M03 spindle on clockwise.

N25 G00 X0 Y0 Z.100

G00 rapid traverse rate to X0 Y0 at the top left corner of the part.
Z.100 tool rapids down to within .100 of the work surface.

Machining the square groove

N30 X.375 Y-.375

tool rapids to position A.

N35 G01 Z-.125 F10

G01 linear interpolation.
Z-.125 tool feeds .125 below the work surface.
F10 feed rate set at 10 in./min.

N40 X1.625 Y-.375

X1.625 top groove cut to the right hand end.
Y-.375 measurement did not change because it was set in block N30.

N45 Y-1.625

Y-1.625 right hand side of the groove cut.

N50 X.375

X.375 bottom groove cut to the left side.

N55 Y-.375

Y-.375 left-hand side of groove cut; this completes the groove.

N60 G00 Z.100

G00 rapid traverse mode.
Z.100 tool rapids to .100 above work surface.

Hole Drilling

N65 G00 X.875 Y-.750

tool rapids to the top left hole location.

N70 G01 Z-.250 F10

tool feeds .250 into work at 10 in./min. to drill the first hole.

N75 G00 Z.100

tool rapids out of hole to .100 above work surface.

N80 X1.250 Y-1.125

tool rapids to second hole location.

N85 G01 Z-.250 F10

tool feeds .250 into work at 10 in./min. to drill the second hole.

N90 G00 Z.100

tool rapids out of hole to .100 above work surface.

Machining the Angular Slot

N95 X1.125 Y-.875 (location B)

tool rapids to the start of the angular slot.

N100 G01 Z-.125 F10

G01 linear interpolation.

Z-.125 tool feeds to .125 below the work surface.

F10 feed rate set at 10 in./min.

N105 X1.250 Y-.750

angular slot cut to top right corner.

N110 G00 Z.100

tool rapids to .100 above work surface.

Machining the Circular Groove

N115 X.750 Y-1.000 (location C)

tool rapids to start of circular groove.

N120 G01 Z-.125 F10

tool feeds to .125 below the work surface.

N125 G03 X1.000 Y-1.250 R.250

G03 circular interpolation counterclockwise

X & Y location of end of circular groove.

R.250 radius of arc is .250.

N130 G00 Z.100

tool rapids to .100 above work surface.

N135 X-1.000 Y1.000

tool rapids back to tool change position.

N140 M05

M05 spindle turned off.

N145 M30

M30 end of program



Hole Operation



- Automatic Tool Changer (ATC) is a multipurpose device designed to:
 - Minimize tool inventory by interfacing different size cutting tools with the same spindle
 - Be easily manipulated by the tool changing mechanism
 - Ensure repeatability of a tooling position with respect to the work every time the tool is used
 - Provide fast and easy off-line tool assembly
- Tool holding systems can be divided to two most popular systems: V-flange (CAT) and the BT-flange.

Fixed / Canned Cycle



- Simplest of operation as it (the program) only needs to specify the coordinates of the hole and type of machine motions to be performed
- Comprise of a G, X & Y coordinate, Z depth, R clearance and F codes
- The following sequence of operations automatically occur:
 - Rapid move to the X and Y coordinate of the hole center
 - Rapid move to the Z-axis clearance plane, R
 - Feed to Z-axis final depth
 - Rapid back to either the Z-axis initial position or the Z-axis clearance plane, R
- The program only needs the next hole coordinates to perform the same operation again

Hole Operation Command



- Drill, Center Drill or Ream Cycle:
 - G81, Xn Yn, Zn, Rn, Fn
- Counterbore Cycle or Spotface Cycle:
 - G82, Xn Yn, Zn, Rn, Fn, Pn
- Deep Drill or Peck Drill Cycle:
 - G83, Xn Yn, Zn, Qn, Rn, Fn
- Tap Cycle:
 - G84, Xn Yn, Zn, Rn, Fn
- Bore Cycle:
 - G85, Xn Yn, Zn, Rn, Fn

Writing a Hole Operation Program



1. Program number/headings
2. Job setup data listing (part origin location, tooling, etc)
3. Tool 1 in spindle; machine start up
4. Hole Operation 1
5. Machine stop
6. Change to tool 2; machine start
7. Hole Operation 2
8. Repeat step 5, 6 & 7 as many time as needed
9. Machine stop; program end



End Chapter 5