CNC Programming Workbook

CODE	FUNCTION
G00	Rapid traverse motion; This is used for non-cutting rapid moves of the machine axis, or rapid retract moves after cuts have been completed. Maximum rapid motion (I.P.M.) of a CNC Machine will vary dependent on machine model.
G01	Linear interpolation motion; Used for cutting in a straight line under a controlled feedrate. Maximum feed rate (I.P.M.) of a CNC Machine will vary depending on the model of the machine.
G02	Circular Interpolation, Clockwise
G03	Circular Interpolation, Counterclockwise
G04	Dwell
G17	Circular Motion XY Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse) G91 is required for rapid move to the G28 reference point.
G40	Cutter Compensation CANCEL

Mill-Lesson-2

Introduction to CNC Codes

camInstructor

LESSON-2 - INTRODUCTION TO CNC CODES
AUTOMATIC TOOL CHANGER
STANDARD TOOL CAROUSEL

The CNC Machining Center used in this text is set-up with following tools.
All program examples and exercises in this workbook are using the tools and tool numbers listed below.

Carousel #	Tool Description
1	0.125" Diameter Flat End Mill
2	0.250" Diameter Flat End Mill
3	0.375" Diameter Flat End Mill
4	0.500" Diameter Flat End Mill
5	0.750" Diameter Flat End Mill
6	0.375" Diameter Spot Drill
7	0.250" Diameter Drill
8	0.201" Diameter Drill – Number 7 drill
9	0.25"-20 UNC Tap
10	#4 Center Drill

COMMONLY USED PREPARATORY **G** CODES

CODE	FUNCTION
G00	Rapid traverse motion; This is used for non-cutting rapid moves of the machine axis, or rapid retract moves after cuts have been completed. Maximum rapid motion (I.P.M.) of a CNC Machine will vary dependent on machine model.
G01	Linear interpolation motion; Used for cutting in a straight line under a controlled feedrate. Maximum feed rate (I.P.M.) of a CNC Machine will vary depending on the model of the machine.
G02	Circular Interpolation, Clockwise
G03	Circular Interpolation, Counterclockwise
G04	Dwell
G17	Circular Motion XY Plane Selection
G20	Verify Inch Coordinate Positions
G21	Verify Metric Coordinate Positions
G28	Machine Home (Rapid traverse) G91 is required for rapid move to the G28 reference point.
G40	Cutter Compensation CANCEL
G41	Cutter Compensation LEFT of the programmed path
G42	Cutter Compensation RIGHT of the programmed path
G43	Tool Length Compensation
G49	Tool Length Compensation CANCEL
G53	Positions the machine axis relative to Machine Home. It is non modal.
G54	Work Coordinate #1 (Part zero offset location)
G80	Canned Cycle CANCEL
G81	Drill Canned Cycle
G82	Spot Drill Canned Cycle
G83	Peck Drill Canned Cycle
G84	Tapping Canned Cycle
G90	Absolute Programming Positioning
G91	Incremental Programming Positioning
G98	Canned Cycle Initial Point Return
G99	Canned Cycle Rapid (R) Plane Return

COMMONLY USED MISCELLANEOUS **M** CODES

CODE	FUNCTION
M00	The M00 code is used for a Program Stop. The spindle stops and the coolant is turned off. Pressing CYCLE START again will continue the program.
M01	The M01 code is used for an Optional Program Stop command. Pressing the OPT STOP key on the control panel signals the machine to perform a stop command when the control reads an M01 command. It will then perform like an M00. Optional stops are useful when machining the first part to allow for inspection of the part as it is machined.
M03	Starts the spindle CLOCKWISE used for most machining. Must have a spindle speed defined. The M03 is used to turn the spindle on at the beginning of program or after a tool change.
M04	Starts the spindle COUNTERCLOCKWISE. Must have a spindle speed defined.
M05	STOPS the spindle. The M05 is used to turn the spindle off at the end of program or before a tool change. If the coolant is on, the M05 will turn it off.
M06	The tool change command along with a tool number will action a tool change. This command will automatically stop the spindle, Z-axis will move up to the machine zero position and the selected tool will be put in the spindle. The coolant pump will turn off right before executing the tool change.
M08	Coolant ON command.
M09	Coolant OFF command.
M30	Program End and Reset to the beginning of program.

Note: Only one "M" code can be used per line. And the M-codes will be the last command to be executed in a line, regardless of where it is located in that line.

EXAMPLE OF PROGRAM **START-UP BLOCKS**

%	Programs must begin and end with “%” depending on the type of control.	
O00023	Letter “O” and up to a five digit program number. Blocks are always terminated by the “;” symbol: End of Block (EOB)	
N10 G20	Nnn - Sequence Number G20 - Verify Inch	Startup Block (Machine Default Setting)
N20 G00 G17 G40 G49 G80 G90	G00 - Rapid Traverse G17 - X, Y Circular Plane Selection G40 - Cutter Compensation Cancel G49 - Tool Length Compensation Cancel G80 - Canned Cycle Cancel G90 - Absolute Programming	
N30 T8 M06	T8 - Tool number #8 to be loaded into the spindle. M06 - Tool Change	
N40 G00 G90 G54 X1.0 Y1.0 S4000 M03	G00 - Rapid Traverse G90 - Activates control to be in ABSOLUTE. G54 - Selects work coordinate offset system No. 1 X__ - Axis move to initial X position. Y__ - Axis move to initial Y position. S4000 - Spindle speed 4000 RPM for this tool. M03 - Turns the spindle on in a clockwise direction	
N50 G43 H8 Z2. 0	G43 - Tool Length Compensation: Recognizes the tool length offset value stored in the Hnn code offset display register in the offset length display. H8 - Defines to the control the offset register the tool offset value is stored in. **** Tool Length offset # = Tool #*** Z2.0 - Informs the control to move from full spindle retract to this Z value and apply the tool length offset.	

EXAMPLE OF PROGRAM **END BLOCKS**

N200 G00 Z2.0	G00 - Rapid Traverse Z2.0 – Retracts tool to 2.0 above part zero	
N210 M05	M05 – Turn off spindle	
N220 G28 G91 Z0 **** N220 G53 Z0***	G91 - Incremental Programming G28 - Machine Zero Return Z0 - Z axis in the up direction to machine zero	***Send to machine zero Z-axis first to avoid any crash.*** ****G53 is another way to return to machine zero***
N230 G28 X0 Y0 **** N230 G53 X0 Y0***	G28 - Machine Zero Return X0 - X axis to machine zero Y0 - Y axis to machine zero	
N240 M30	M30 – End of Program and Reset	

EXAMPLE OF PROGRAM **TOOL CHANGE LINES**

N100 G00 Z2.0	Rapid Traverse and Retracts tool to 2.0 above part zero	
N110 M05	M05 – Turn off spindle	
N120 G28 G91 Z0 ; / ****N120 G53 Z0*	Machine Zero Return - Z axis	***Send to machine zero Z-axis first to avoid any crash.***
N130 G28 X0 Y0 / ****N130 G53 X0 Y0*	Machine Zero Return - X, Y axis	
N140 M01	Optional Program Stop	
N150 T9 M06	Tool Change - Tool # 9	
N160 G00 G90 G54 X1.0 Y1.0 S4000 M03	Turn on the spindle and Rapid traverse to X1. Y1.	
N170 G43 H9 Z2.0	Tool Length compensation for Tool #9 (H9)	

*G53 - Positions the machine axis relative to Machine Home. It is non modal.

RAPID **G00** AND LINEAR **G01** INTERPOLATION

G00 RAPID TRAVERSE

This code is used for rapid motion of the cutter in air to traverse from one position to another as fast as possible. This code will work for all axis motion up to three axes at once.

This G00 code is modal and causes all the following blocks to be in rapid motion until another Group 01 code is specified. The actual rapid federate is dependent on the machine.

Generally, rapid motions "will not" be in a straight line. All the axes specified are moved at the maximum speed and will not necessarily complete each axis move at the same time. It activates each axis drive motor independently of each other and, as a result, the axis with the shortest move will reach its destination first. So ***you need to be careful of any obstructions to avoid with this type of rapid move.***

- G00 is used when you are positioning the cutter in 'fresh air'.
- Retracting from a hole you have drilled.
- Rapid traverse is not used when cutting the part.
- Used incorrectly, rapid traverse will break a cutter very easily.

G01 LINEAR INTERPOLATION

This G code provides for straight line (linear) motion with programmed feedrate for all axis motions from point to point. Motion can occur up to three axes at once.

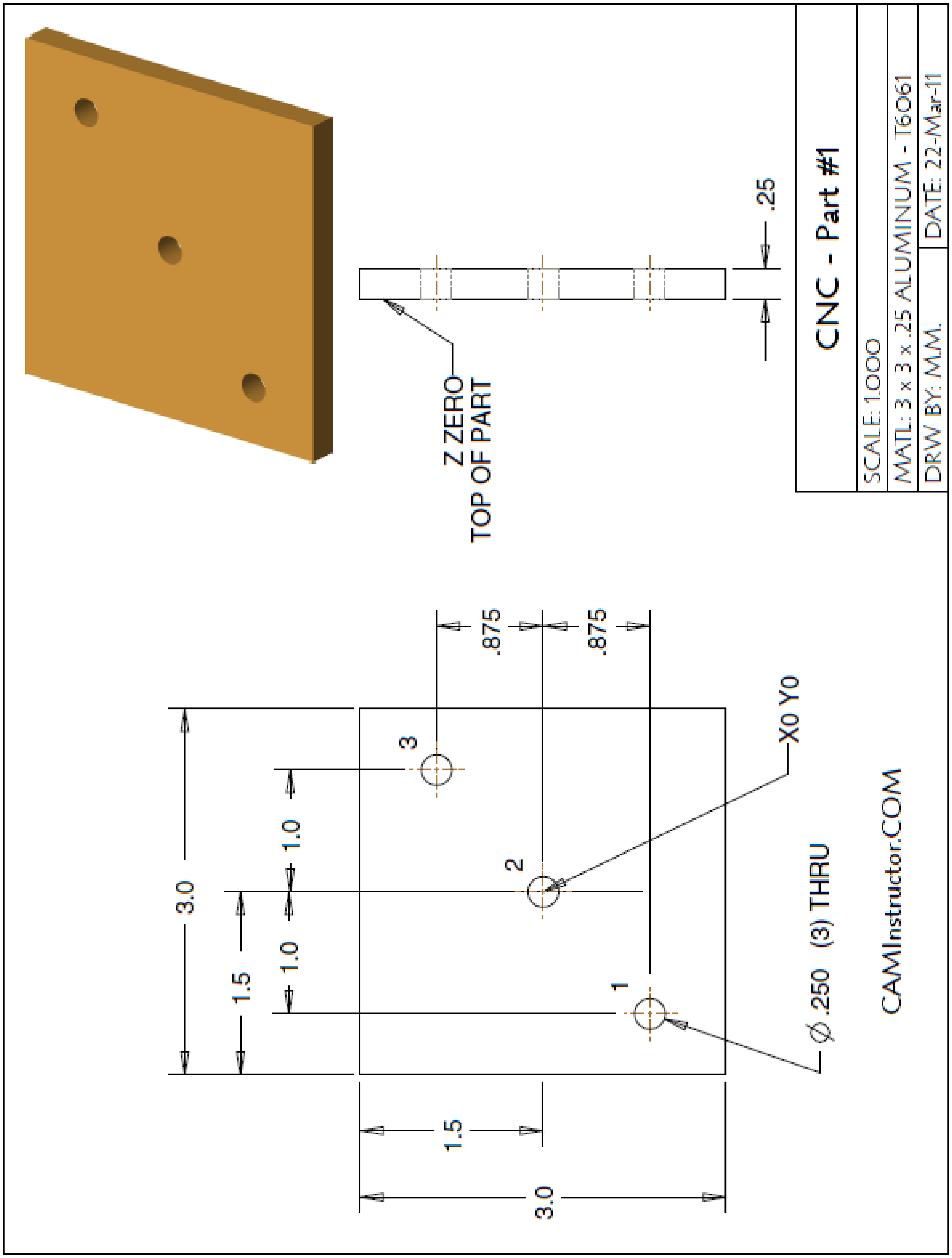
All axes specified will start at the same time and proceed to their destination and arrive simultaneously at the specified feedrate.

To program a feedrate, the F command is used. The F command is modal and may be specified in a previous block.

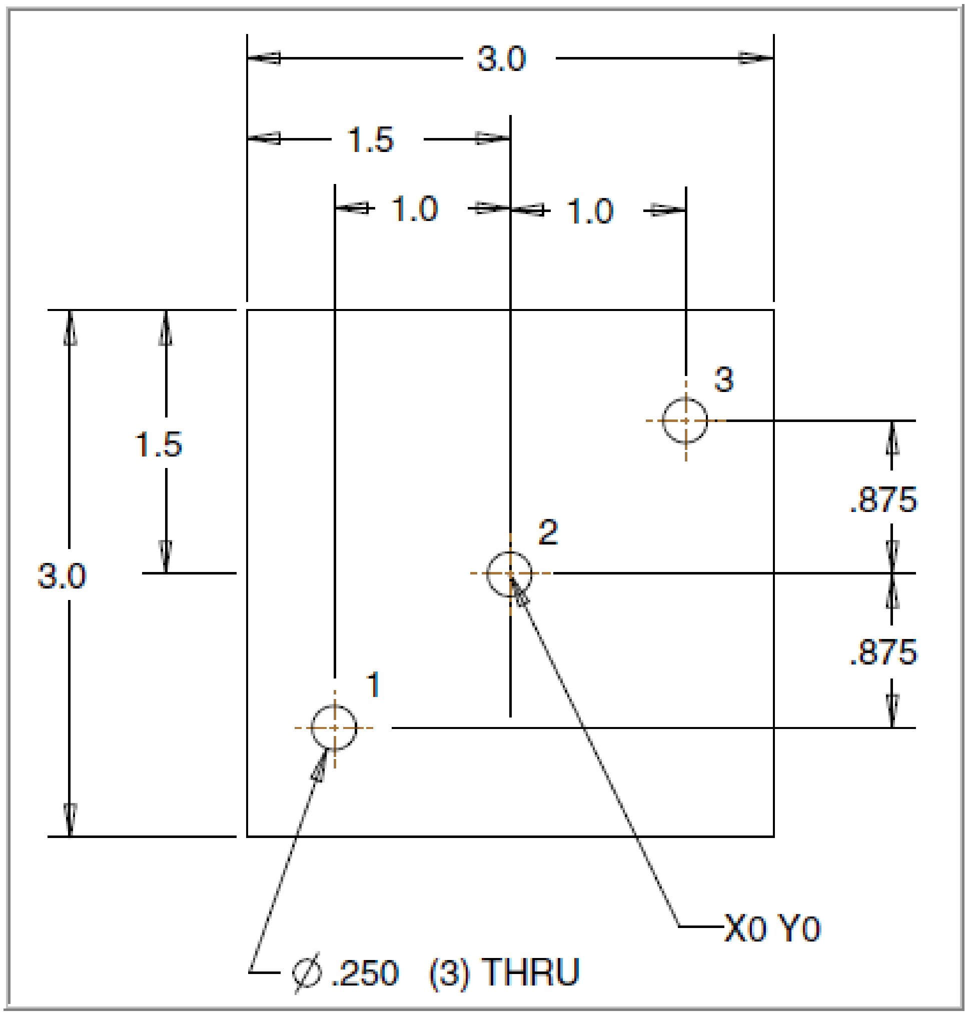
G01 is used for

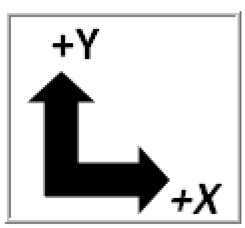
- Drilling a hole
- Machining a slot
- Machining a profile

LESSON-2 - CNC - PART #1



- ***WORK OUT THE X AND Y COORDIANTES FOR HOLES 1,2 AND 3***
  - X0Y0 is at the centre of the part

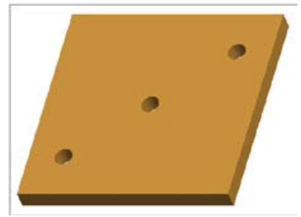




G90	X	Y
1		
2		
3		

LESSON-2 - CNC - PART #1

- ***PROGRAM TO <u>SPOT DRILL THE THREE HOLES ONLY</u> USING A COMBINATION OF G00 AND G01 (CANNED CYCLE DRILL WILL BE USED LATER)***
  - Below is the program to spot drill the three holes with an explanation of each block
  - Use a 0.375” diameter Spot Drill Tool # 6
  - Spindle Speed = 2750 Feed rate = 11 IPM
  - Spot Drill Depth = Z-0.150”
  - X0Y0 is at the centre of the part
  - Z=0 is the top of the part.
  - Information inside the parenthesis ( ) is a comment.
  - The CNC control will ignore all text between the parenthesis



%	(Program must begin and end with a %)
O1	(Program #1 - CNC-PART-1-SPOT DRILLING ONLY)
N10 G20	(Inch programming)
N20 G00 G17 G40 G49 G80 G90 (MACHINE DEFAULT SETTING)	
N30 T06 M06	(T6-Select tool number 6 to be loaded M06-Activates the tool changer)
N40 G00 G90 G54 X-1.0 Y-0.875 S2750 M03 – (Rapid to the X and Y position and turn on the spindle at 2750 RPM)	
N50 G43 H06 Z0.1	(G43 - Activate the tool offset value stored in H06 and rapid to Z0.1)
N60 G01 Z-0.15 F11.0	(Hole #1 - Feed down to Z depth at 11 inches per minute)
N70 G00 Z0.1	(G00- Retract out of hole #1 at rapid to 0.1 above the top of the work piece)
N80 X0 Y0	(G00 is modal - Move at rapid in the X and Y axis to hole #2)
N90 G01 Z-0.15	(Hole #2 - Feed down to Z depth at 11 inches per minute, Feed rate is modal)
N100 G00 Z0.1	(G00- Retract out of hole #2 at rapid to 0.1 above the top of the work piece)
N110 X1.0 Y0.875	(G00 is modal - Move at rapid in the X and Y axis to hole #3)
N120 G01 Z-0.15	(Hole #3 - Feed down to Z depth at 11 inches per minute, Feed rate is modal)
N130 G53 G00 Z0 M05	(G53 – Machine Zero positioning, non modal. Rapid to machine zero in Z, switch spindle off)
N140 G53 X-15.0 Y0	(G53 – Rapid in relation to machine zero X-15.0 and Y0)
N150 M30	(Program end rewind program to the beginning)
%	(Program must begin and end with a %)