



innovative machine tools



# **GE FANUC 21 CONCEPT 55 MILL TRAINING GUIDE ON PC KEYBOARD**

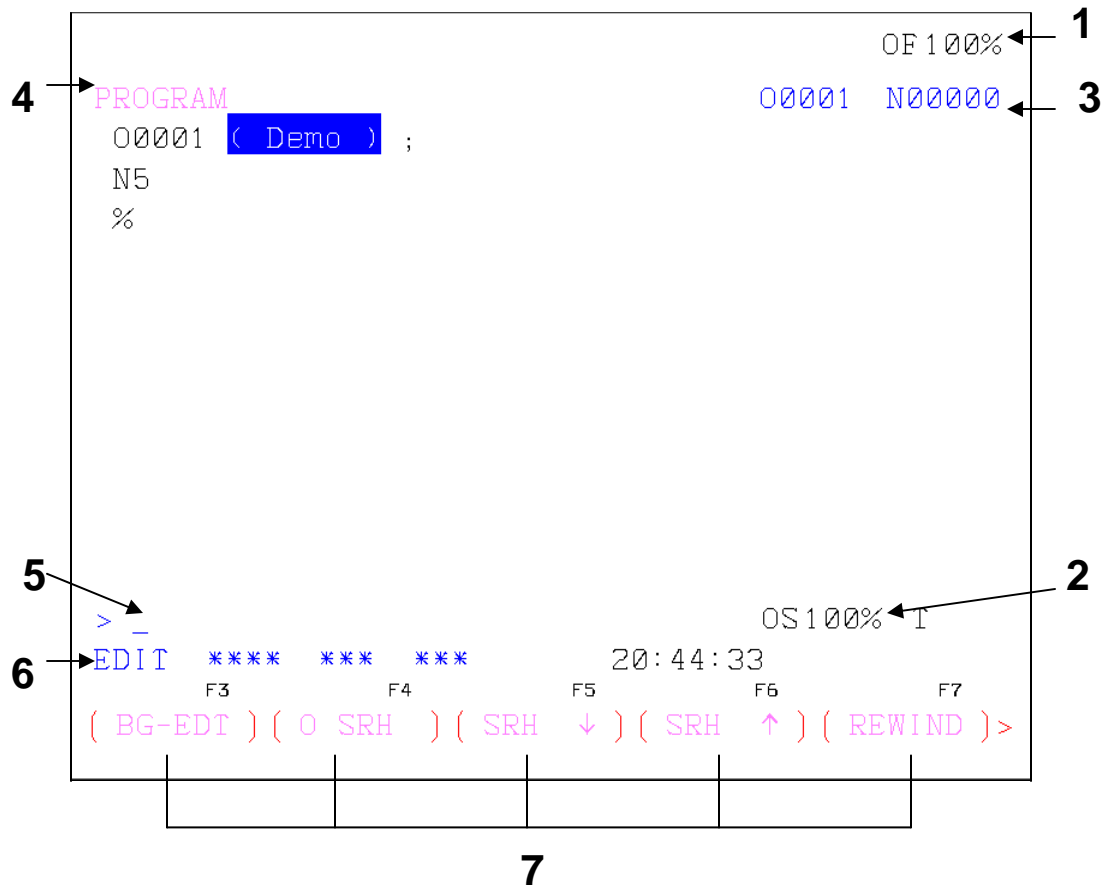
5/16/08 Version 1  
Made by EMCO  
Authored by Chad Hawk

# TRAINING INDEX

Fanuc 21 Screens .....	Pg 1
Fanuc 21 keys .....	Pg 2
• Cursor Movement keys	
• Change keys	
Function keys (Display keys) .....	Pg 3
• Machine Function keys	
Direction keys .....	Pg 4
• Spindle Overrides keys	
• Accessory Functions	
Mode Dial .....	Pg 5
• Feed Override Dial	
Pc Keyboard picture .....	Pg 6
Reference the machine .....	Pg 7
Work Shift (Description) .....	Pg 8
Work Shift for X .....	Pg 9
• Manually index the Tool Turret	
• Manually turning on Spindle	
Work Shift for Y .....	Pg 11
Tool Offsets (Description) .....	Pg 13
Setting Tool Offset .....	Pg 14

<b>Program Training .....</b>	<b>Pg 15</b>
<b>Insert a New Program .....</b>	<b>Pg 16</b>
• Call a Existing Program up	
• Insert a Code	
• Insert a End of Block	
<b>Delete a Program .....</b>	<b>Pg 17</b>
• Delete all Programs	
• Delete a Code	
• Delete a Block or Line number	
<b>Cancel Mistyped Codes .....</b>	<b>Pg 18</b>
• Alter a Code	
• Search for Code	
• Search for Letter	
<b>G Codes .....</b>	<b>Pg 19</b>
<b>M Codes .....</b>	<b>Pg 20</b>
• Used Addresses	
 <b>Program 1 .....</b>	 <b>Pg 22</b>
<b>2D simulation (setup) .....</b>	<b>Pg 23</b>
<b>Input and Output the Programs &amp; offsets thru the Fanuc software</b>	<b>Pg 25</b>
<b>Running program.....</b>	<b>Pg 26</b>
<b>Program 2 (C/R) .....</b>	<b>Pg 27</b>
<b>Program 3 (Deep hole Drilling) .....</b>	<b>Pg 28</b>
<b>Program 4 (R's) .....</b>	<b>Pg 29</b>
<b>Program 5 (I&amp;J) .....</b>	<b>Pg 30</b>
<b>I &amp; J Test .....</b>	<b>Pg 31</b>
<b>Program 6 (Sub Programming) .....</b>	<b>Pg 32</b>
<b>Program 7 (Pocket Milling) .....</b>	<b>Pg 33</b>
<b>Program 8 (Sub For program 7) .....</b>	<b>Pg 34</b>

# The Fanuc 21 Screen



1. **Displays of Feed**
2. **Spindle Speed override**
3. **Display of Program and Number block**
4. **Display of active Screen**
5. **Entry line**
6. **Display of active Mode**
7. **Display of Soft key Functions**

# **FANUC 21 KEYS ON PC KEYBOARD**

## **CURSOR MOVEMENT KEYS**

**Arrow key pointing up is CURSOR UP = moves cursor up**

**Arrow key pointing right moves cursor right**

**Arrow key pointing left moves cursor left**

**Arrow key pointing down is CURSOR DOWN = moves cursor down, search function, program call up**

**PAGE UP = moves one page up**

**PAGE DOWN = moves one page down**

## **CHANGE KEYS**

**Insert key is the ALTER = alter word (replace word)**

**Enter is the INSRT = insert word, create new program**

**DELETE = deletes word / block or a program**

**Enter pressed twice is EOB = End of a block or line**

**Back space is the CAN = deletes entries in the address**

## **FUNCTION KEYS (DISPLAY KEYS)**

### **F12 TOGGLES THE MENU FOR THE DISPLAY KEYS**

**F12- F3 is for POS = displays actual, relative & all positions**

**F12- F4 is for PRGRM = displays program, library page**

**F12- F5 is for OFFSET = displays offset & work pages**

**F12- F6 is for SYSTEM = displays parameters & diagnostic pages**

**F12- F7 is for ALARM = displays operator & alarm message**

**F12- F11- F3 is for GRAPH = displays 2-d graph simulation**

## **MACHINE FUNCTION KEYS**

**# keys is the same as Numeric keypad or 10 key**

**Press / on # keys = (SKIP) Press skip any block lines with ( / )  
(Slash) before block number will be skipped**

**Press Ctrl & / on # keys = (DRY RUN) Test run without spindle on  
(Remove raw material from chuck)**

**Press Ctrl & \* on # keys = (Optional stop) for programs with (m1)**

**Press 0 on the # keys = (Reset) cancels most alarms, resets  
program, interrupts programs**

**Press \* on # keys = (Single block) reads one block at a time**

**Press . on # keys = (Cycle stop) program hold, feed hold**

**Press Enter on # keys = (Cycle start) program start**

## **DIRECTION KEYS**

These keys control axis directional movements

1 on the # keys moves Y axis -  
2 on the # keys moves Z axis -  
4 on the # keys moves X axis -  
6 on the # keys moves X axis +  
8 on the # keys moves Z axis +  
9 on the # keys moves Y axis +

Ctrl & 4 = Feed stop

Ctrl & 5 = Feed start

Both works in all modes but EDIT & ZRN

## **SPINDLE OVERRIDE KEYS**

Ctrl & + on the # keys increase the spindle speed (50% to 120% highest)

Ctrl & - on the # keys decrease the spindle speed (120% to 50% lowest)

Ctrl & 6 = Spindle stop

Ctrl & 7 = Spindle start

Works in all modes except EDIT & ZRN

## **ACCESSORY FUNCTIONS**

Press Ctrl & + for Door open

press again Door closed

Press Ctrl & 3 for Rotary axis Indexing

Press Ctrl & 0 vise open

Press Ctrl & 9 vise closed

Press Ctrl & 2 coolant on

press again coolant off (only 100/125/155)

Press Ctrl & 2 puff blowing on

press again puff blowing off

Press Ctrl & 8 auxiliary drives on

Press Ctrl & - auxiliary drives off

Press Ctrl & 1 releases tool holder from spindle Note: Hold on to holder  
will drop

## **MODE CONTROL**

### **F1 TOGGLES THE MENU FOR THE MODE CONTROL**

**F1 THEN F7 = REF for Reference or Home mode**

**F1 THEN F3 = MEM for Automatic mode for running a program**

**F1 THEN F4 = EDIT mode for program changes or entering a new program**

**F1 THEN F5 = MDI for Manual Data Input mode for manually programming  
and running the machine**

**F1 THEN F6 = JOG for Manual moving the axis in X; Y; or Z**

**F1 THEN F11 = STEPS Incremental feed movements**

**F1 THEN F11 THEN F3 = STEPS 1 OR .0001 or tenths**

**F1 THEN F11 THEN F4 = STEPS 10 OR .001 thousands**

**F1 THEN F11 THEN F5 = STEPS 100 OR .010 ten thousands**

**F1 THEN F11 THEN F6 = STEPS 1000 OR .100 hundred thousands**

## **FEED OVERRIDE CONTROL**

**+ on the # keys increase the feed rate speed (0% to 120%)**

**- on the # keys decrease the feed rate speed (120% to 0%)**

**These Control feed for jogging in the X-axis / Y-axis / Z axis**



Toggle Back					Over Toggle							
Esc	Mode	A	F3	F4	F5	F6	F7	F8	Output	Input	>	Display

Turret Air Rotary Jog Jog Spindle Spindle Vise Vise Door													
~	1	2	3	4	5	6	7	8	9	0	-	+	Backspace
.	!	@	#	\$	%	^	&	*	(	)	-	+	Cancel
Tab	Q	W	E	R	T	Y	U	I	O	P			
Caps Lock	A	S	D	F	G	H	J	K	L	EOB			Insert Input
Shift	Z	X	C	V	B	N	M						Shift
Ctrl		Alt	Space Bar							Alt			Ctrl

Alter		Page Up
Delete	End	Page Down

<	Λ	>
---	---	---

Num Lock	Dry Run	Op Stop	≡	-
	Skip	SBL	Λ/Λ	-
	Z+	Y+	≡	+
X-	REF ALL	X+	Λ/Λ	+
Y-	Z-		NC Start or	
Reset		NC Stop	(cycle start	

- Any key with Gray highlight Press Ctrl + the key for that function
- Some keys have two functions to them for 1st function just press the key
- 2nd function will be Grey press Ctrl + the key for the function
- Some automative keys when you press them 1 time this will close/turn off press them again will open/turn on
- F1 is a toggle key for the modes: Zero, Auto, Edit, MDI, Jog and F1 then F11 give Increment Step
- F12 is a toggle key for the Display screens: Position, Program, Offsets, Parameter, Alarm and F12 then F11 then F3 gives Graph
- F12 then F11 then F3 then F11 then F3 gives you 3D view
- Press enter 2 times this is the same as pressing EOB insert
- Alt + F4 will exit the software back to the desktop
- The Top right corner will allow the screen to be minimized, restored and close just like a standard windows screen

The machine functions are active only with NUM LOCK on

Keys are active they will move the axes if used as numbers. Use numbers on the keyboard.

# **Turning the Machine On/Entering Fanuc Software**

## **Referencing the Machine**

1. Make sure your feed rate above "0"
2. If vise is open then open door Ctrl & + and Ctrl & 9 to close vise
3. Make sure door is closed Ctrl & +
4. Do step 4, 5, 6 or just do step 7
5. Press 8 on the # keys this references the Z axis.
6. Press 4 on the # keys this references the X axis
7. Press 1 on the # keys this references the Y axis

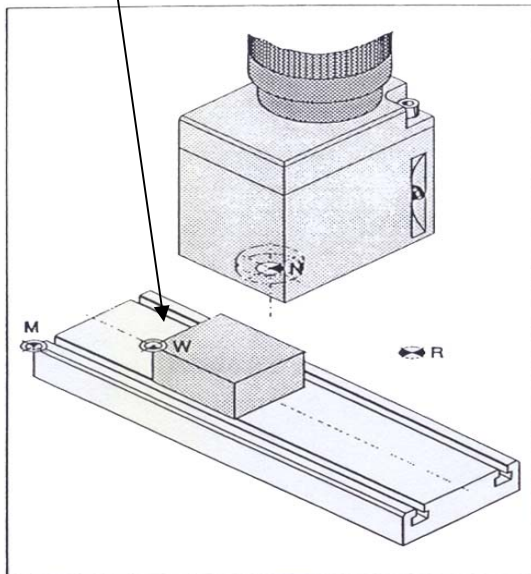
**OR**

8. Press 5 on the # keys this references all axis

**Note: Every time you enter Fanuc 21 Software or Turn the Machine On you must reference the axis**

# WORK SHIFT

Pages 8 – 16 is setting the Work shift & offsets to the lower left corner & the top of the part with the Spindle nose



Reference points in the working area

## Reference Points of the EMCO Milling Machines

### M = Machine zero point

An unchangeable reference point established by the machine manufacturer.

Proceeding from this point the entire machine is measured.

At the same time "M" is the origin of the coordinate system.

### R = Reference point

A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the "R".

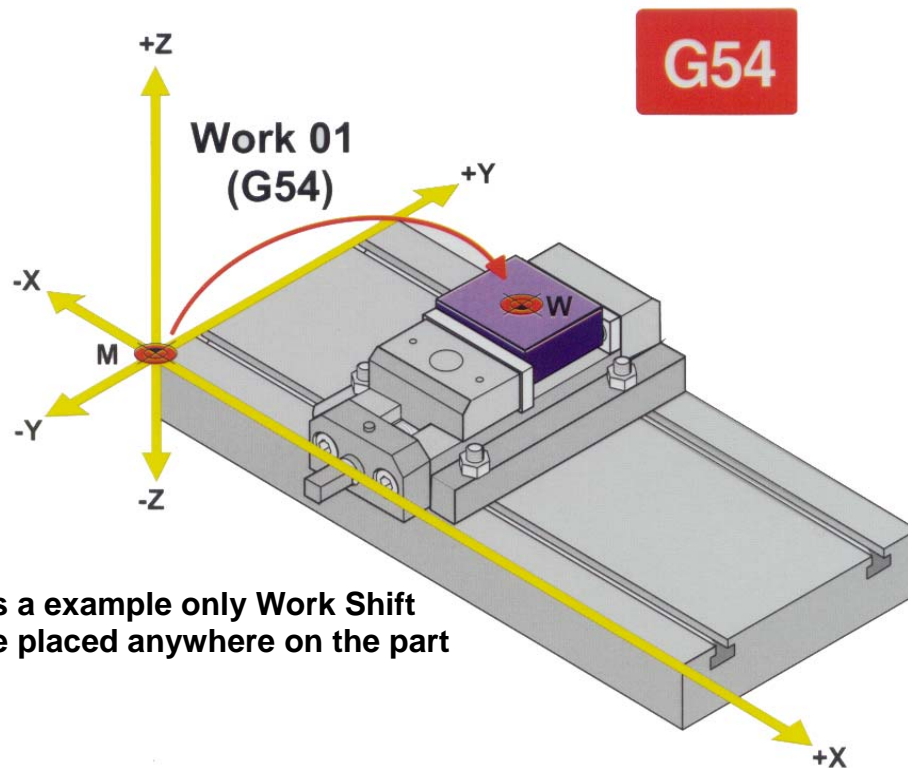
Required after every power failure.

### N = Tool mount reference point

Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

### W = Workpiece zero point

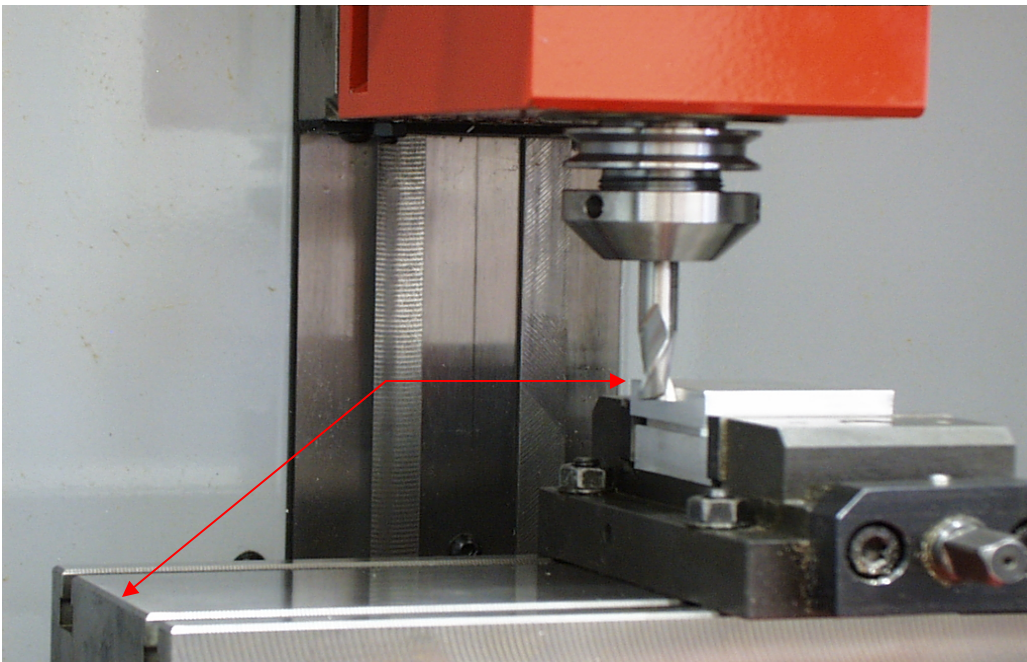
Starting point for the dimensions in the part program. Can be freely established by the programmer and moved as desired within the part program.



This is a example only Work Shift can be placed anywhere on the part

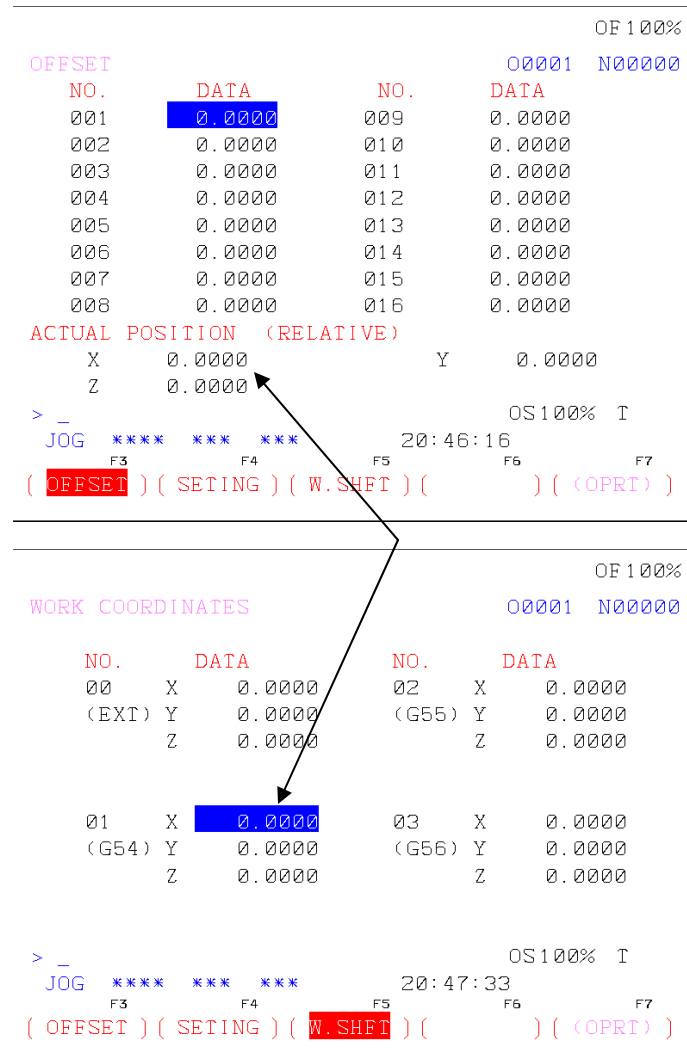
## Work Shift: Industry Way

1. Index to a edge finder or a tool (Ex. use 3/8 endmill)
  - Press F1 then F5 for MDI; press F12 then F4 for program
  - Type in T1 M6 if this position has the tool being used
2. Either Follow step 3 or follow step 4 then go to step 5
3. Press F1 then F6 for Jog and jog the Tool to the left side of the Work Piece & touch using 1,2,4,6,8,9 on the # keys.  
(Use Feed Dial or Steps to approach slower)
4. For Scratching:
  - Type S1000 for spindle speed then type M03 Spindle on Clockwise
  - Then press Enter on the # keys (Make sure door is closed)
  - Press F1 then F6 for Jog mode
  - Jog the Tool to the left side of the Work Piece & touch using 1,2,4,6,8,9 on # keys.



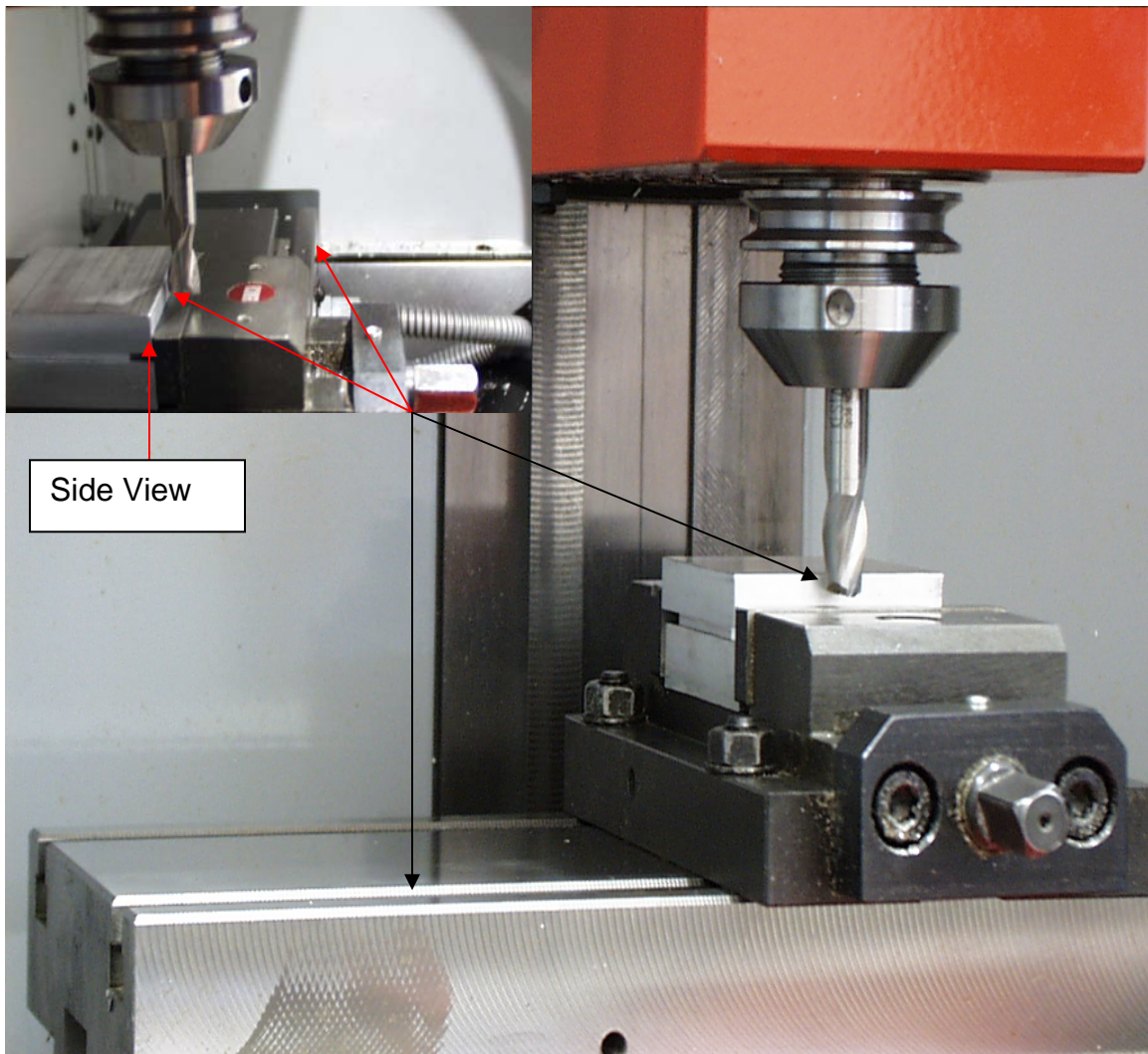
**Note:** Machine 0 in X is the center of the spindle to the left side of the Machine bed.

5. Press F12 then F5 for OFFSET
  - Look at the top left of the page to make sure the page is on offset if not press F3
  - Type in the value for Actual Position Relative X
6. Press the F2 for blue keys
7. Press F5 for W.SHIFT
8. Move Cursor to 01 location for X
9. Press enter then type in the radius of the tool being used to scratch ( example 3/8 Tool) then press F6 for +input





10. Jog Spindle up away from WORK PIECE using Z+
11. Jog the Tool to the Front of the Work Piece & touch using  
1,2,4,6,8,9 on # keys.  
(Use Feed Dial or Steps to approach slower)



**Note:** Machine 0 in Y is the center of the spindle to the Front of the Machine bed.

12. If the page is not on OFFSET or W.SHFT then press F12 then F5 then press F3 for OFFSET

- Look at the top left of the page to make sure the page is on offset if not press F3
- Type in the value for Actual Position Relative Y

13. Press the F2 for blue keys

14. Press F5 for W.SHIFT

15. Move Cursor to 01 location for Y

Press enter then type in the radius of the tool being used to scratch ( example 3/8 Tool) then press F6 for +input

OF 100%

OFFSET		00001 N00000	
NO.	DATA	NO.	DATA
001	0.0000	009	0.0000
002	0.0000	010	0.0000
003	0.0000	011	0.0000
004	0.0000	012	0.0000
005	0.0000	013	0.0000
006	0.0000	014	0.0000
007	0.0000	015	0.0000
008	0.0000	016	0.0000

ACTUAL POSITION (RELATIVE)

X	0.0000	Y	0.0000
Z	0.0000		

> \_ JOG \*\*\* \*\*

OS100% T 20:46:16

( OFFSET ) ( SETING ) ( W.SHFT ) ( ) ( OPRT )

---

OF 100%

WORK COORDINATES		00001 N00000	
NO.	DATA	NO.	DATA
00	X 0.0000	02	X 0.0000
(EXT)	Y 0.0000	(G55)	Y 0.0000
	Z 0.0000		Z 0.0000
01	X 0.0000	03	X 0.0000
(G54)	Y 0.0000	(G56)	Y 0.0000
	Z 0.0000		Z 0.0000

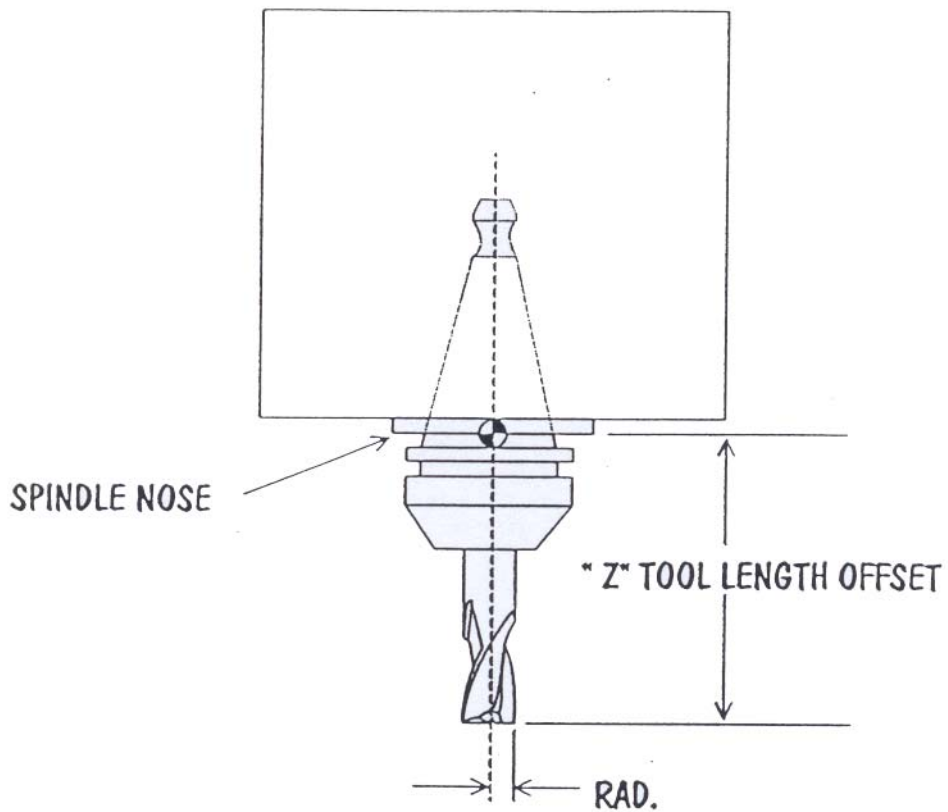
> \_ JOG \*\*\* \*\*

OS100% T 20:46:33

( OFFSET ) ( SETING ) ( W.SHFT ) ( ) ( OPRT )

21. Jog the Tool up above the Work Piece

# TOOL OFFSET



				OF 100%	
OFFSET				00000	N00000
NO.	DATA	NO.	DATA		
001	0.0000	009	0.0000		
002	0.0000	010	0.0000		
003	0.0000	011	0.0000		
004	0.0000	012	0.0000		
005	0.0000	013	0.0000		
006	0.0000	014	0.0000		
007	0.0000	015	0.0000		
008	0.0000	016	0.0000		
ACTUAL POSITION (RELATIVE)					
X	0.0000	Y	0.0000		
Z	0.0000				
> _				OS100% T	
JOG	****	***	***	11:05:38	
F3	F4	F5	F6	F7	
{ OFFSET }	{ SETING }	{ W.SHFT }	{	{ (OPRT) }	}



1. Jog Tool tip down & touch the Top of the Work Piece  
(Use Feed Dial or Steps to approach slower)
  - Use your Tools to do your Tool Offsets
2. Press F12 then F5 for OFFSET
3. The value in Actual Position (Relative) Z, type this value in Offset NO. 001(H1) If tool is going to be T1
4. Place the Radius in the corresponding Offset 011 (H11)
  - This is for the cutter compensation when using G41 or G42
5. To set more Tools Repeat Steps 1 thru 5
  - Drills & Taps do not need a Radius set for them

---

OF 100%

OFFSET		00000 N00000	
NO.	DATA	NO.	DATA
001	0.0000	009	0.0000
002	0.0000	010	0.0000
003	0.0000	011	0.0000
004	0.0000	012	0.0000
005	0.0000	013	0.0000
006	0.0000	014	0.0000
007	0.0000	015	0.0000
008	0.0000	016	0.0000

ACTUAL POSITION (RELATIVE)			
X	0.0000	Y	0.0000
Z	0.0000		

> \_
OS100% T

JOG \*\*\*\*\*
11:05:38

F3
F4
F5
F6
F7

{ OFFSET }
{ SETING }
{ W.SHFT }
{ }
{ (OPRT) }

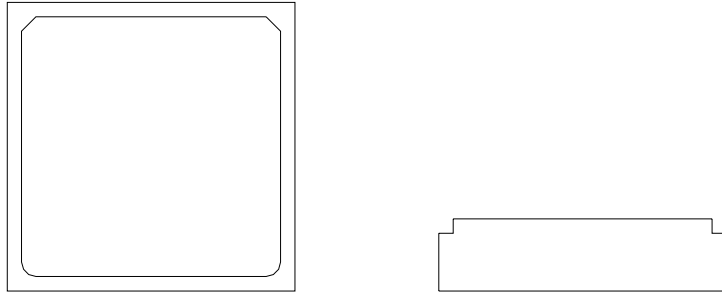
---

**NOTE:** When you use a T the H = Height

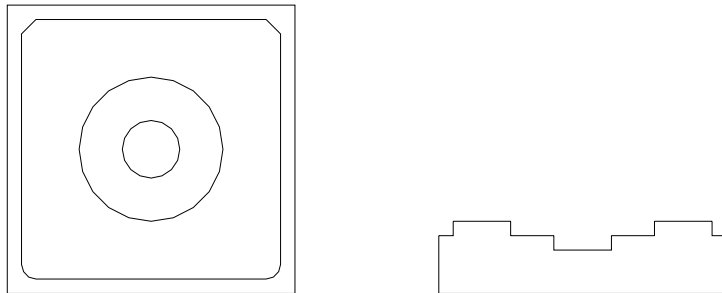
When you use a G41 or G42 the H = Radius

# Program Training

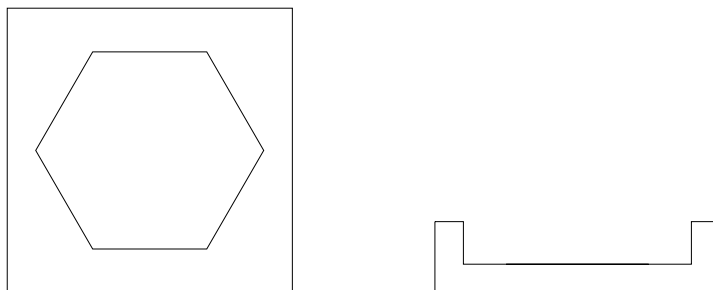
## Program O0001



## Program O0003



## Program O0005



Press F1 then F4 for Edit & Press F12 then F4 for Program to do functions below & on the next 2 Pages

- **INSERT A NEW PROGRAM**

1. Press letter O then program number
2. Press Enter

Example: O0001 OR O1

- **CALL A EXISTING PROGRAM UP**

1. Press letter O then program number
2. Press arrow pointing down

- **INSERT A WORD**

1. Press letter then number
2. Press Enter

**HINT:** When inserting a word to the left of the highlighted word the new word will be placed

Example: **N5** G01 X 0.25; G01 is the word being inserted

- **INSERT END OF BLOCK**

1. Press (;)
2. Press Enter
3. Or press enter 2 times

**HINT:** at the end of each number block needs an End Of Block looks like a Semicolon (;)

Example: N5 G01 X1.00 F.003;

- **DELETE A PROGRAM**

1. Press letter o then program number
2. Press Delete

**Example:** O0001 OR O1

- **DELETE ALL PROGRAMS**

1. Press letter o plus the – & 9999
2. Press Delete

**Example:** O – 9999

- **DELETE A WORD**

1. Highlight the word
2. Press Delete

- **DELETE A BLOCK OR LINE NUMBER**

1. Type the number line and highlight the number line
2. Press Delete

- **CANCEL MISTYPED WORD**

1. Press Backspace

**HINT:** In the ADRS. (Address) at the lower left of the screen is the word and numbers that's been typed in. Before pressing enter check if what was typed in is correct. If not press backspace and retype word

- **ALTER A WORD**

1. Highlight the word needed altered type the change
2. Press Insert

- **SEARCH FOR NUMBER BLOCK**

1. Press letter n and the number of the block
2. Press arrow pointing down

- **SEARCH FOR WORD**

1. Type in word & number
2. Press arrow pointing down

- **SEARCH FOR LETTER**

1. Press letter
2. Press arrow pointing down

**HINT:** This goes to the first (G). Follow steps 1 & 2 cursor goes to the next (G)

## Survey commands G-CODES : Mostly used only

<b>Model</b>	<b>G00</b>	<b>Rapid motion</b>
<b>Model</b>	G01	Linear interpolation in working feed
<b>Model</b>	G02	Circular interpolation, clockwise
<b>Model</b>	G03	Circular interpolation, counter-clockwise
<b>Non Model</b>	G04	Dwell time, active block by block only
<b>Non Model</b>	G09	Exact holds, active block by block only
<b>Model</b>	<b>G17</b>	<b>Selection of plane X-Y</b>
<b>Model</b>	G18	Selection of plane Z-X
<b>Model</b>	G19	Selection of plane Y-Z
<b>Model</b>	G20	Dimension in inch
<b>Model</b>	G21	Dimension in millimeter
<b>Non Model</b>	G28	Approach reference point, active block by block only
<b>Model</b>	<b>G40</b>	<b>Cancel cutter compensation</b>
<b>Model</b>	G41	Cutter compensation left
<b>Model</b>	G42	Cutter compensation right
<b>Model</b>	G43	Tool length compensation positive
<b>Model</b>	G44	Tool length compensation negative
<b>Model</b>	<b>G49</b>	<b>Cancel tool length compensation</b>
<b>Model</b>	G53	Machine coordinate system (00)
<b>Model</b>	<b>G54</b>	<b>Zero point shift 1 (01)</b>
<b>Model</b>	G55	Zero point shift 2 (02)
<b>Model</b>	G56	Zero point shift 3 (03)
<b>Model</b>	G57	Zero point shift 4 (04)
<b>Model</b>	G58	Zero point shift 5 (05)
<b>Model</b>	G59	Zero point shift 6 (06)
<b>Model</b>	G73	Chip break cycle
<b>Model</b>	<b>G80</b>	<b>Cancel drilling cycle (ALL Drilling Cycles)</b>
<b>Model</b>	G81	Spot or chamfer drilling cycle
<b>Model</b>	G83	Deep hole drilling cycle
<b>Model</b>	<b>G90</b>	<b>Absolute value programming</b>
<b>Model</b>	G91	Incremental value programming
<b>Model</b>	<b>G94</b>	<b>Feed in inch/min</b>
<b>Model</b>	G95	Speed with feed in inch/revolution
<b>Model</b>	<b>G97</b>	<b>Spindle speed per minute</b>
<b>Model</b>	<b>G98</b>	<b>Retract to plane of start (drilling cycles)</b>

## Survey commands M-CODES: Mostly used

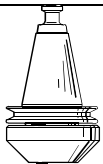

M00	Programmed stop, unconditional
M01	Programmed stop, conditional
M03	Spindle ON clockwise
M04	Spindle ON counter clockwise
<b>M05</b>	<b>Spindle OFF</b>
M06	Tool Index
M25	Open clamping vice
M26	Close clamping vice
M30	Main program end with new start of program
M71	Blow-off ON
<b>M72</b>	<b>Blow-off OFF</b>
M98	Subroutine call-up
M99	Subroutine end

**A maximum of three M commands allowed for each program block!**

## Used Addresses

A	Angle
C	Chamfer
F	Feed rate, thread pitch
G	Path function
H	Tool height, tool radius
I, J, K	Circle parameter, scale factor, K number of repetition, I if statements
M	Miscellaneous function
N	Block number 1 to 9999
O	Program number 1 to 9499 only (label) not in program as a # 0
P	Dwell, subroutine
Q	Cutting depth or shift value
R	Radius, retraction height
S	Spindle speed, limit
T	Tool called out
X, Y, Z	Position data

### Tools needed for Programs 1, 2, 3, 4, 5, 6

F1Z 010	<b><u>Collet holder</u></b>	For ESX-25 collets	
<b>225 100</b>	<b>(9.0-10.0mm)Ø 3/8"</b>	<b>ESX 25 COLLETS</b>	
<b>764 308</b>	Acc. to DIN 327, shape B cutting-ø10 mm / shank-ø10mm	<b><u>Slot end mill, HSS</u></b>	

#### Program screen & Edit mode

- To edit / change a program / insert new programs & input or output excising programs & offsets

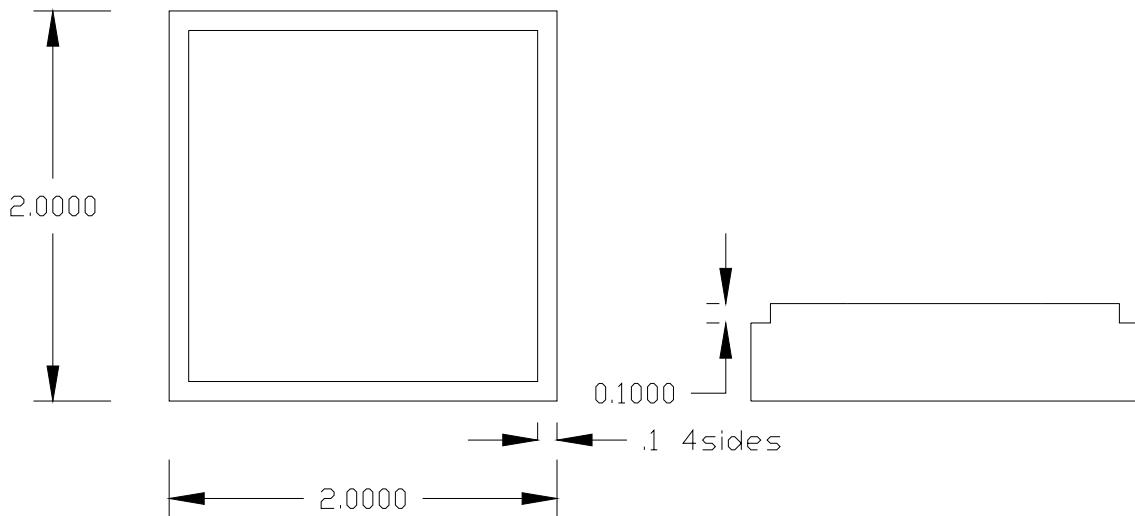
#### Program screen & MDI mode

- To manually program the spindle speed / move the axis (X,Y,Z) to a specified location and or Index to a certain tool

**Note: Material is 2024-T4 Alum, All feeds & speeds are programmed for this type of Aluminum**



## Program Q0001



O0001 (Demo) (2 X 2 X .5 Alum.)

N5 **G00 G17 G40 G80**

N10 **G90 G94 G98**

N15 **G54**

N20 G43 T1 H1 M6 (3/8 or 10mm Endmill) Tool call out line

N25 S1800 M3

N30 G0 Z1

N35 X-1 Y1

N40 Z-.1

N45 G1 G41 H11 X.1 F7

N50 Y1.9

N55 X1.9

N60 Y.1

N65 X.1

N70 Y1

N75 G0 G40 X-1

N80 G91

N85 G28 Z0

N90 G90

N95 M30

**Default G code  
not needed**

Work shift Call out

Tool call out line

Spindle on CW

Safe move above part

Positioning X, Y for cutting

Positioning Z at depth

CRC on and moving to X

Position Top left corner

Position Top right corner

Position Bottom right corner

Position Bottom left corner

Back to start point Y

CRC off and start point X

incremental mode

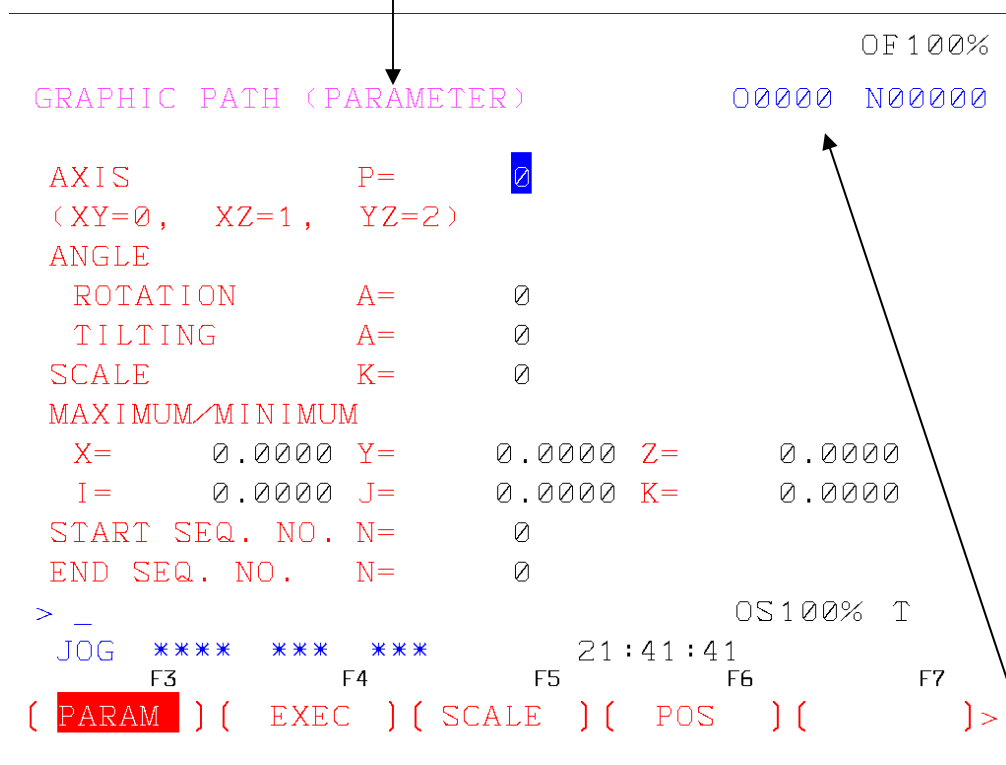
from position to Z home

absolute mode

End of Program

## 2D Simulation

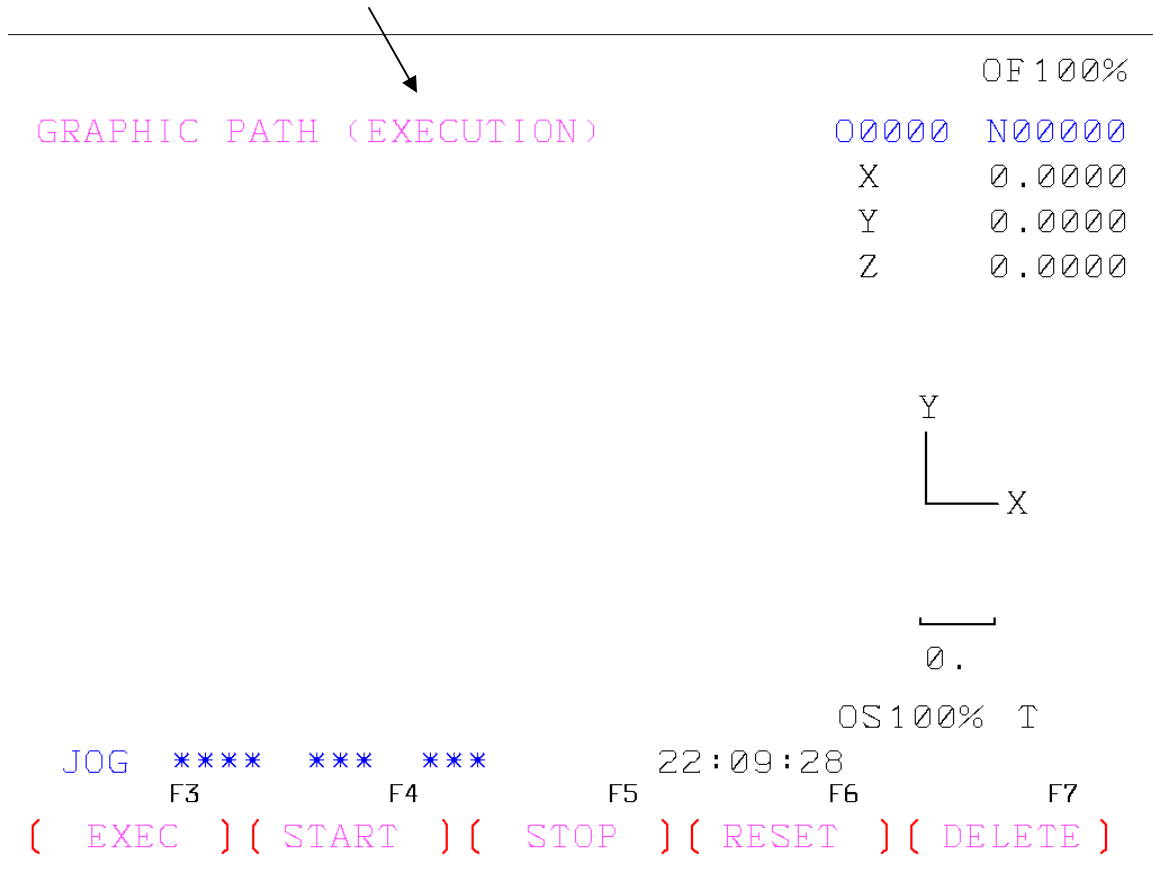
1. Press F12 F11 F3 for **Graph** screen to appear



**Note:** There are only 7 values you can change on this page the rest of them change by the values you will enter. This graph only works with an active program and runs only the current program selected

2. Axis P = 0 means G17 1 means G18 2 means G19
3. Maximum/Minimum X = Overall Length of the stock in X direction this is a positive value
4. Maximum/Minimum Y = Overall Width of the stock in Y direction this is a positive value
5. Maximum/Minimum Z = Overall Height of the stock in Z direction this is a positive value
6. Maximum/Minimum I = This value is normally a negative number and this is the viewable area passed X0 going negative
7. Maximum/Minimum J = This value is normally a negative number and this is the viewable area passed Y0 going negative
8. Maximum/Minimum K = This value is normally a negative number and this is the viewable area passed Z0 going negative

9. Press F4 for Execution screen



Note: If you press the EXEC on this screen this will auto scale for you. You will need to press the arrow left on the soft keys to go back and enter your values that you originally had.

10. Now press F4 for **start** and you will see the tool movements of the program

- **Changing I/O to floppy drive** (Only need to do this once stays default)

1. Press F1 then F4 for **EDIT**
2. Press F12 F6 for **System**
3. Page down until you see Parameter (Manual)
4. Cursor down to the I/O
5. Type A (for the Floppy Drive) press Input key

Other Drives useable: **B (Drive), C (Drive), P (Printer), 1, 2 (Com Ports)**

**Note: If you want to use USB use C and then follow instruction in the Appendix**

- **Output Program from Fanuc software to Drive unit**

1. Press F12 then F4 for **Program**
2. Type program number to be send out  
Example: letter Q and program number  
(Q0002) or (Q2)
3. Press F11
4. Press F5 for Punch then press F7 for Exec

- **Output Offsets from Fanuc software to Drive unit**

1. Press F12 F5 for **Offset**
2. Press F7 for (OPRT)
3. Press F11
4. Press F5 for Punch then press F7 for Exec

- **Input Program into Fanuc Software from Drive unit**

1. Press F12 then F4 for **Program**
2. Type program number to be send out  
Example: letter Q and program number  
(Q0002) or (Q2)
3. Press F11
4. Press F4 for Read then press F7 for Exec

- **Input Offsets into Fanuc Software from Drive unit**

1. Press F12 F5 for **Offset**
2. Press F7 for (OPRT)
3. Press F11
4. Press F4 for Read then press F7 for Exec

## Running a Program

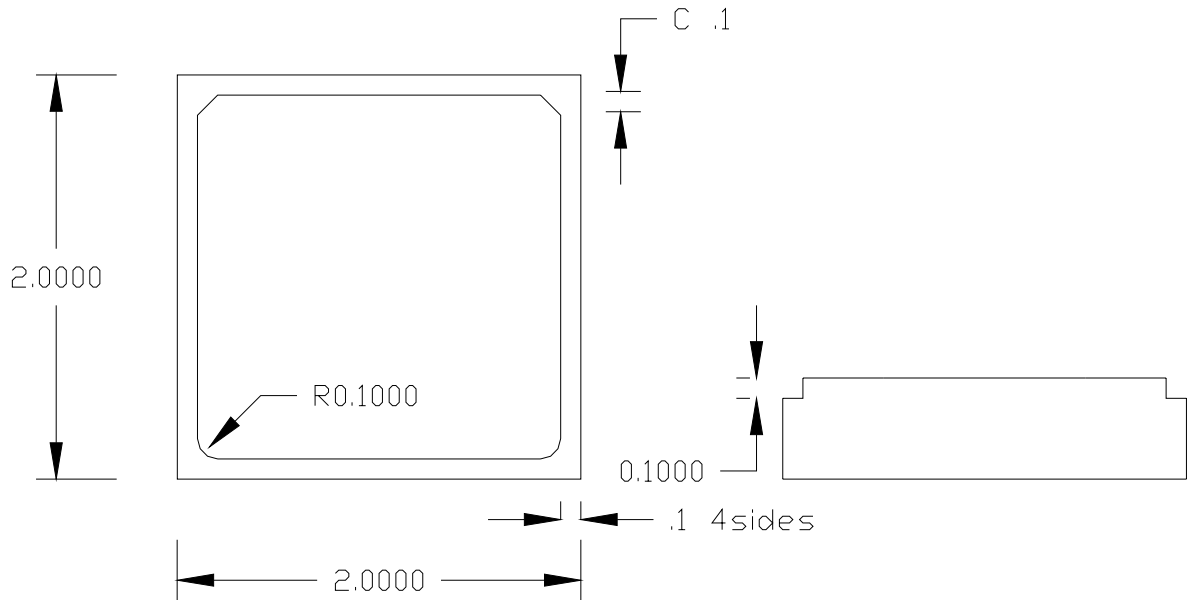
Note: If the correct program # is at the top right corner of the screen then skip step 3 only and press reset for step 3

1. F1 then F4 for Edit
2. Press F12 then F4 for program
3. Call up Program to be run / cut  
(Example O1 for program 1)
4. F1 F3 for MEM
5. If screen is not in **PROGRAM CHECK** then press F12 and F4 until this is at the top left of the screen
6. Press the \* on the # keys for Single Block for the program to run one line at a time.

Note: Use one hand on the feed override + / - on the # keys slowly increasing it and the other pressing enter on the # keys for cycle start and close to the reset button

7. Press enter on the # Keys for Cycle Start and continue  
(Once the program have moved in the safe called out locations for X, Y, Z and looks right; you can take single block off and run the program)
8. Press enter on the # Keys for Cycle Start one more time  
(If there are more than one tool before the next tool use single block to check the offsets locations for the Z only then continue at step 8 again)

## Program Q0002 (C & R)



O0002 (C/R) (2 X 2 X .5 Alum.)

N5 **G00 G17 G40 G80**

N10 **G90 G94 G98**

N15 **G54**

N20 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N25 S1800 M3

N30 G0 Z1

N35 X-1 Y1

N40 Z-.1

N45 G1 G41 H11 X.1 F7

N50 Y1.9 **C.1**

N55 X1.9 **C.1**

N60 Y.1 **R.1**

N65 X.1 **R.1**

N70 Y1

N75 G0 G40 X-1

N80 G91 G28 Z0

N85 G28 X0 Y0

N90 G90

N95 M30

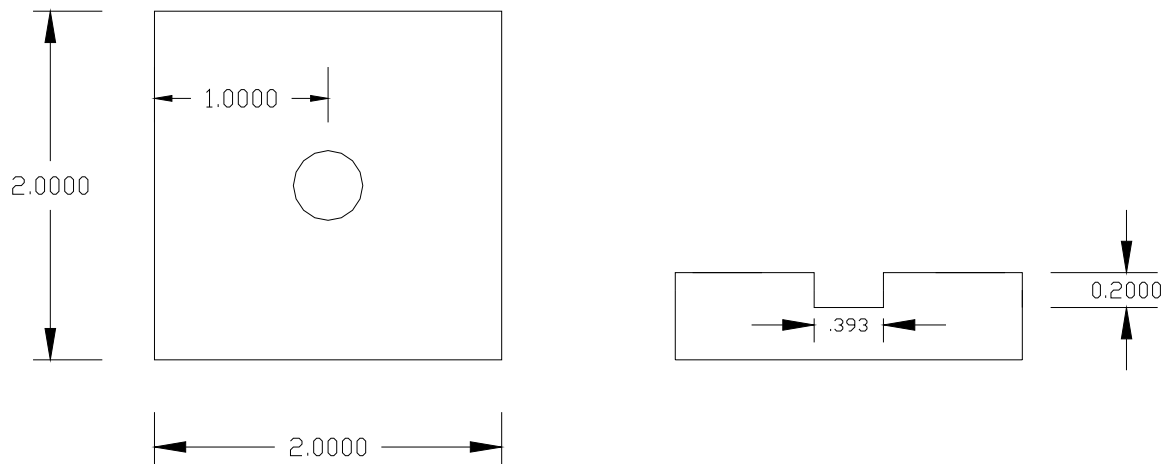
Automatic Chamfer 45 degrees

Automatic Chamfer 45 degrees

Automatic Radius 90 degrees

Automatic Radius 90 degrees

## Program Q0003 (Deep Hole Drilling)



G83 X = Location of hole Y = location of hole

**Z = Overall Depth of hole** P = Dwell at bottom of hole

R = Retract after Cycle **Q = incremental peck depth per pass**

K = Incremental repeats only used with G91 **F = Feed rate**

O0003 (3 Drill) (2 X 2 X .5 Alum)

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

**N25 X1 Y1**

**N30 Z.05**

**N35 G83 Z-.2 R.1 Q.05 F3**

**N40 G80**

N45 G91 G28 Z0

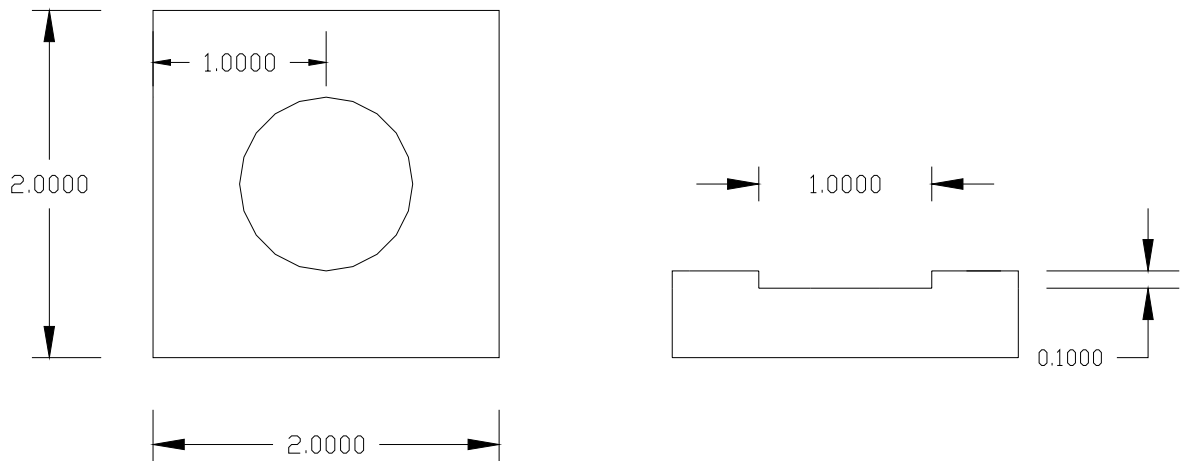
N50 G28 X0 Y0

N55 G90

N60 M30

**Note: G0, G1, G2, G3 all cancel the drilling cycle**

## Program Q0004 (R)



O0004 (R) (2 X 2 X .5 Alum)

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.1

N35 G1 Z-.1 F3

N40 S1800

N45 G1 G42 H11 X.5 F5

**N50 G2 X1.5 Y1 R.5 (180 Degrees)**

**N55 G2 X.5 Y1 R.5 (180 Degrees)**

N60 G0 G40 X1

N65 G91 G28 Z0

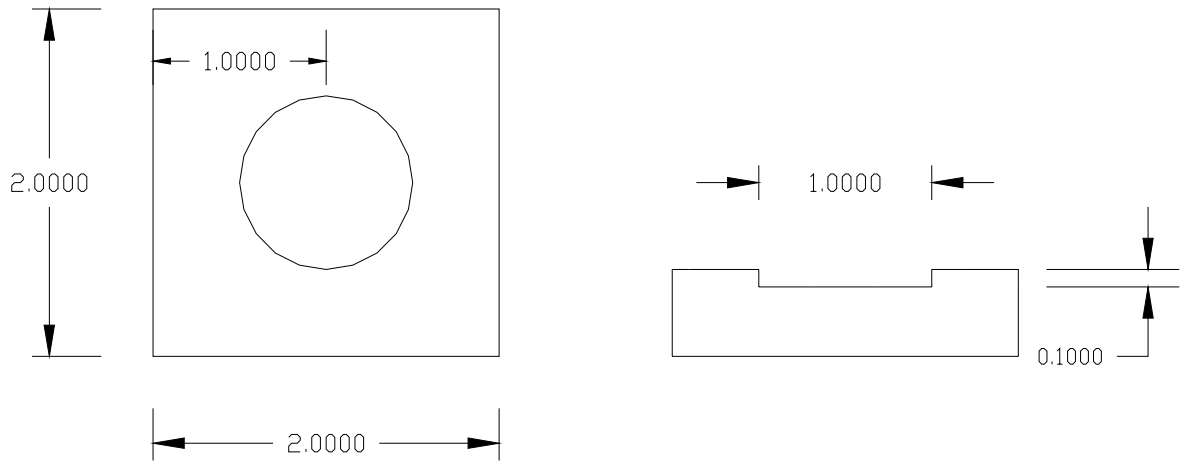
N70 G28 X0 Y0

N75 G90

N80 M30



## Program Q0005 (I & J)



O0005 (I/J) (2 X 2 X .5 Alum)

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

**N30 Z.1**

**N35 G1 Z-.1 F3**

**N40 S1800**

**N45 G1 G42 H11 X.5 F5**

**N50 G2 X.5 Y1 I.5 J0 (360 degrees)**

**N55 G0 G40 X1**

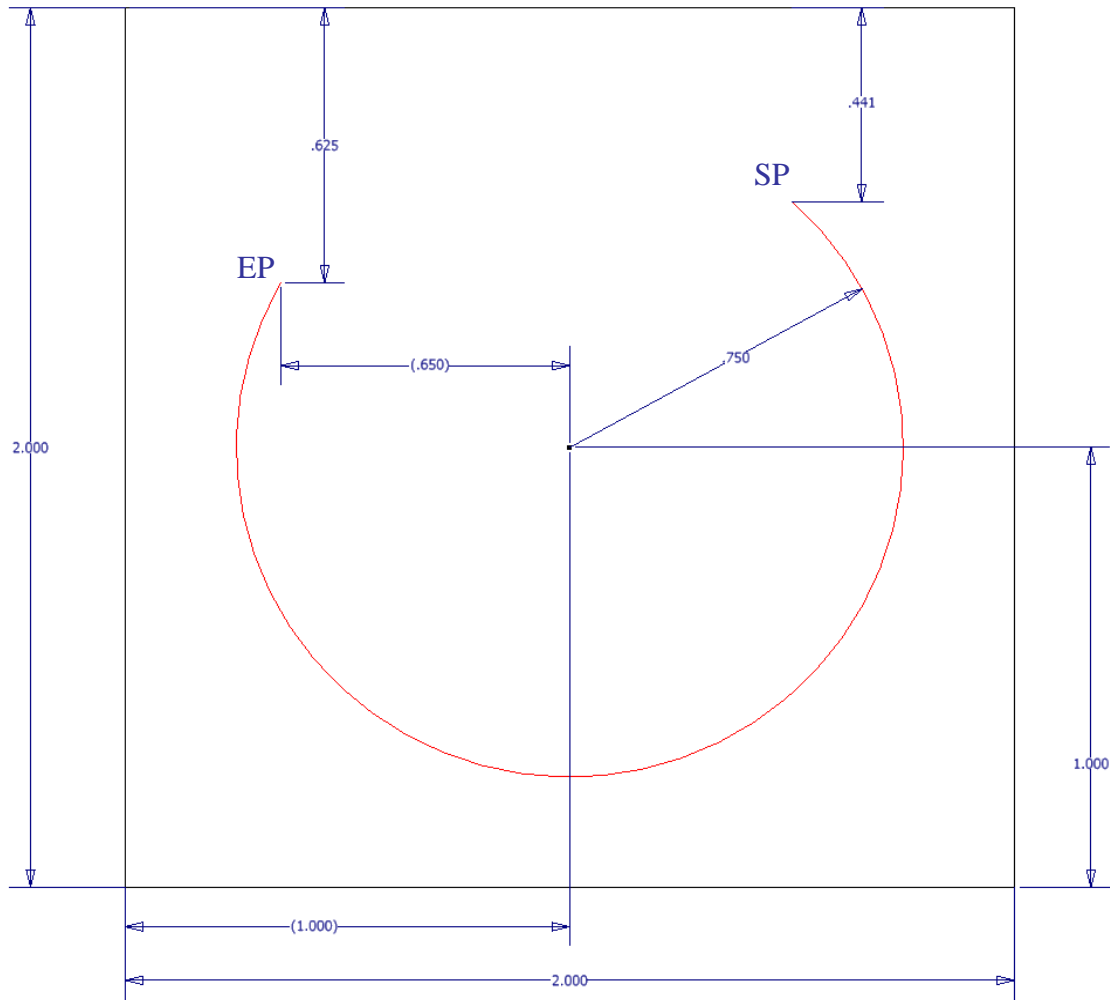
N60 G91 G28 Z0

N65 G28 X0 Y0

N70 G90

N75 M30

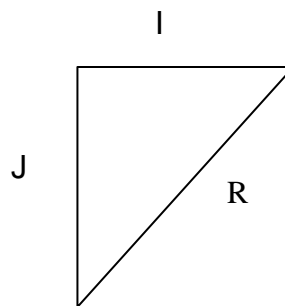
## I and J Test



Using T1 (3/8 End Mill) and cutting on the inside of the red arc starting at the SP

```

N10 G1 Z-.1
N15 G1 G H X (SP)
N20 G1 Z-.1
N25 G X Y I J (EP)
N30 G1 Z-.1
N35 G0 G
    
```



$$A^2 \text{ (K leg)} + B^2 \text{ (I leg)} = C^2 \text{ (H radius)}$$

S  $\frac{O}{H}$   
 C  $\frac{A}{H}$   
 T  $\frac{O}{A}$

Degrees

Sally Can Tell Oscar Has A Hat On Always

SINE COSINE TANGENT

1. To make all programs tie together O0002, O0003, O0004 to run together. Use M98 this calls out Sub programs or Sub routines.  
Example: M98 P010001
2. After M98 P is identified with 6 digits.
  - The First 2 digits is the number of times program is to be repeated
  - The next 4 digits is the program number without the letter O
3. Programs that are being used as a Sub Programs must end with M99 instead of M30.
4. All programs can be used as Sub Programs or Main Programs  
M99 means program is Sub, M30 means program is a Main
5. A main Program can also use M99 at the end.
  - Program is being used to repeat without cutting multiple parts.
  - This is mainly used for Demo's for just seeing Tool movements.
6. To link all 3 programs together follow Program O0006
  - Program O0002(C/R), O0003, O0004(I,J) must all have M99 at the end to link together

## **TEST FOR SUB PROGRAMS**

O0006 (Tie Programs)

N5

N10 (Demo 2 C/R)

N15 (Demo 3 Drilling)

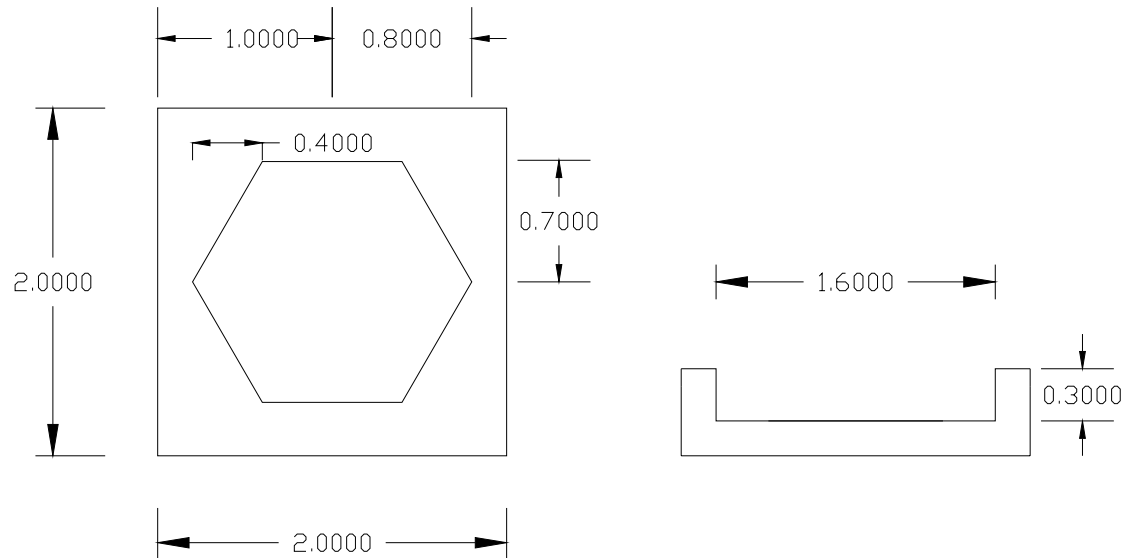
N20 (Demo 5 I & J)

N25 M30

Changing Item

Note: Change the end of O0002, O0003, and O0005 to M99 for running them as SUB PROGRAMS

## Program O0007 (Pocket Milling) (Making a Cycle)



O0007 (Pocket) (2 X 2 X .5 Alum)

N5 G54

N10 G43 T1 H1 M6 (3/8 or 10 mm end mill)

N15 S1500 M3

N20 G0 Z1

N25 X1 Y1

N30 Z.1

N35 G1 Z0 F3

N40 M98 P030008

N45 G0 G91 G28 Z0

N50 G28 X0 Y0

N55 M30

## **Program O0008 (Sub for program 7)**

O0008 (Sub for Prog 7)

N5 G91

N10 G1 Z-.1 F2

N15 G90

N20 S1800

N25 G41 H11 X.4 Y1.35 F7

N30 X.2 Y1

N35 X.6 Y.3

N40 X1.4

N45 X1.8 Y1

N50 X1.4 Y1.7

N55 X.6

N60 X.2 Y1

N65 X.4

N70 X.8 Y.5

N75 X1.2

N80 X1.6 Y1

N85 X1.2 Y1.5

N90 X.8

N95 X.4 Y1

N100 G0 G40 X1

N105 M99

Shorter Program Test:

Make program O0008 shorter by using the information given during the training!

N5 G91

N10 G1 Z-.1 F2

N15 G90

N20 S1800

N25 G41 H11 X.4 Y1.35 F7

N30 X.2 Y1

N35 X.6 Y.3

N40 X1.4

N45 X1.8 Y1

N50 X1.4 Y1.7

N55 X.6

N60

N65

N70

N75

N80

N60

N65

N70

N75

N80

N85

# Appendix

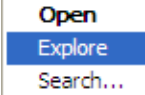
## Changing Drive to USB Port

### 1. Close out the SW (software)

- Press ALT and F4 to exit the Software

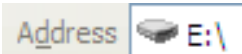

### 2. Make sure USB is plug into port

### 3. Open Explorer

- Right Click on Either My Computer, My Documents or any Folder on the Desktop
- Move mouse to  (Explorer)
- Left Click
- If you right clicked on My computer skip to step 4 if not then Left Click on My Computer

### 4. Copy Drive directory

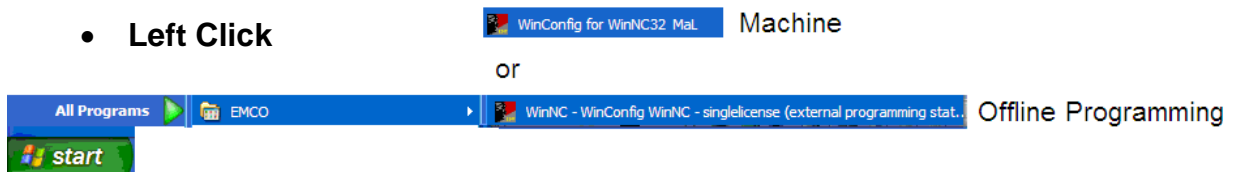


- Click on you USB drive
- At the top of the active screen or page in the Address copy or remember drive info 
- Close the active screen or page using either Alt and F4 or  at top of the active screen





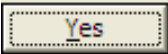
## 5. Setting up WinConfig

- Left Click on Green Start button on Desktop
- Move mouse to All Program or Programs
- Move mouse to EMCO
- Move mouse to WinNC-WinConfig WinNC or WinNC32 – Singlelicense or MultipleLicense or Mal (Machine)


- Left Click



## 6. In Winconfig

- Left Click on  (INI) button
- Double Left Click on **Directories** (Directories)
- Left click on white box  (Import / Export directory)
- Either Press Ctrl and V (this will paste in the info) or type in USB directory
- Left Click on  (OK)
- Left Click on  (Close)
- Left Click on  (Yes) to save the changes

## 7. Restart SW (software)

- Left Click on Green Start button on Desktop
- Move mouse to All Program or Programs
- Move mouse to EMCO
- Move mouse to WinNC with this  icon on it
- Left Click