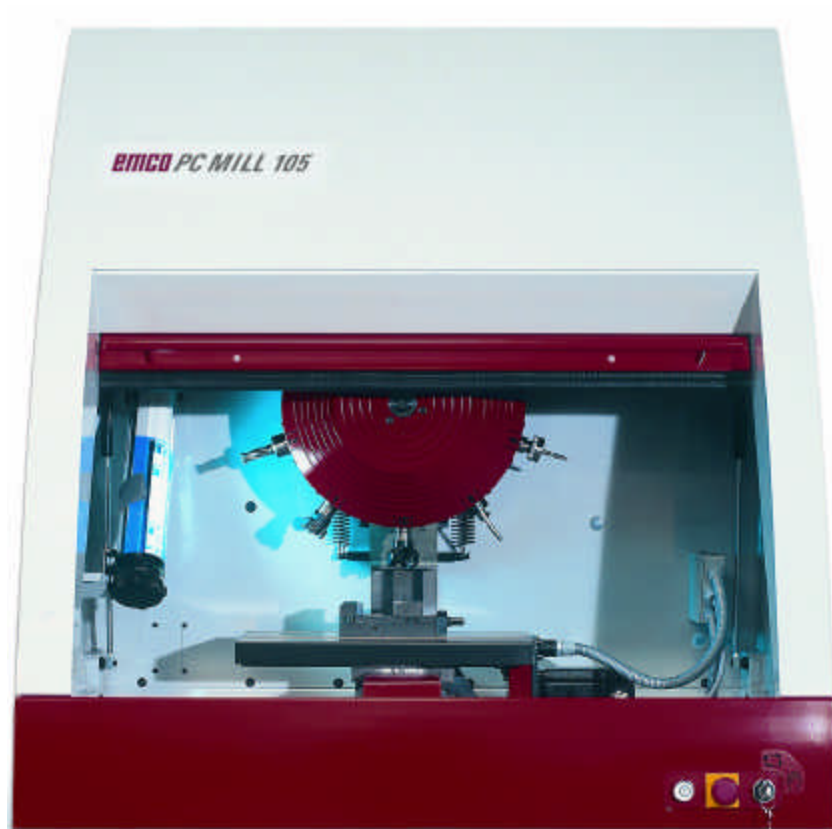




innovative machine tools



GE FANUC 21 50/55 MILL TRAINING GUIDE ON PC KEYBOARD

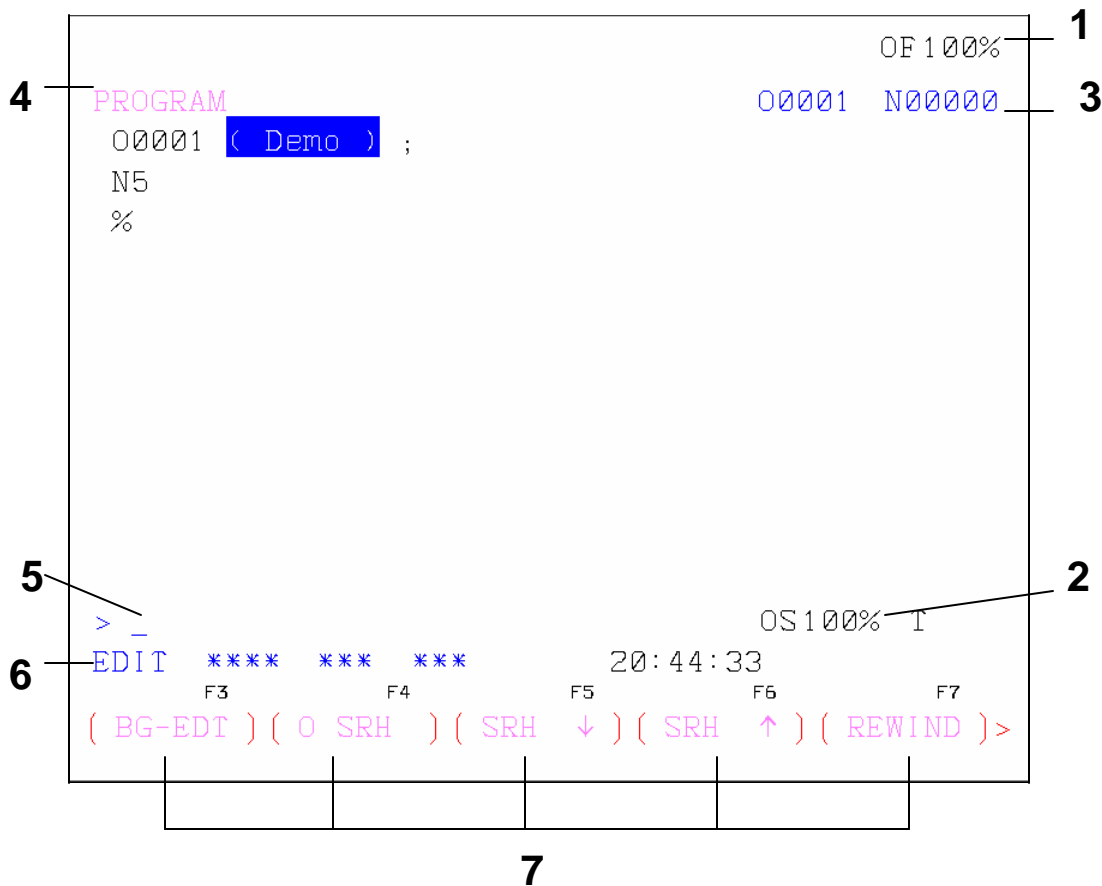
9/11/03 Version 6
Made by EMCO
Authored by Chad Hawk

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 Z	Pg 9
Work Shift for X	Pg 11
• Manually index the Tool Turret	
• Manually turning on Spindle	
Work Shift for Y	Pg 13
Tool Offsets (Description)	Pg 15
Setting Tool Offset	Pg 16

Program Training	Pg 17
Insert a New Program	Pg 18
• Call a Existing Program up	
• Insert a Code	
• Insert a End of Block	
Delete a Program	Pg 19
• Delete all Programs	
• Delete a Code	
• Delete a Block or Line number	
Cancel Mistyped Codes	Pg 20
• Alter a Code	
• Search for Code	
• Search for Letter	
G Codes	Pg 21
M Codes	Pg 22
• Used Addresses	
 Program O0001	 Pg 24
2D simulation (setup)	Pg 25
Input and Output the Programs & offsets thru the Fanuc software	Pg 27
Program O0001 (C/R)	Pg 28
Program O0002 (Deep hole Drilling)	Pg 29
Program O0002 (I/J)	Pg 30
Program O0003 (R's)	Pg 31
Program O0004 (Sub Programming)	Pg 32
Program O0005 (Pocket Milling)	Pg 33
Program O0006 (Sub For program 5)	Pg 34

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



**=(Agreement button) used for open/closing door or to jog axis
with the door open**



=(Mode Key) Automatic & Hand Mode

**Hand Mode is for moving machine around with door open and
works in conjunction with the (Agreement button)**

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

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

- The machine functions are active only with NUM LOCK on

Turning the Machine On/Entering Fanuc Software

Referencing the Machine

1. Make sure your feed rate is not on "0"
2. Make sure door is closed
3. Do step 4, 5, 6 or just do step 7
4. Press 8 on the # keys this references the Z axis.
5. Press 4 on the # keys this references the X axis
6. Press 1 on the # keys this references the Y axis

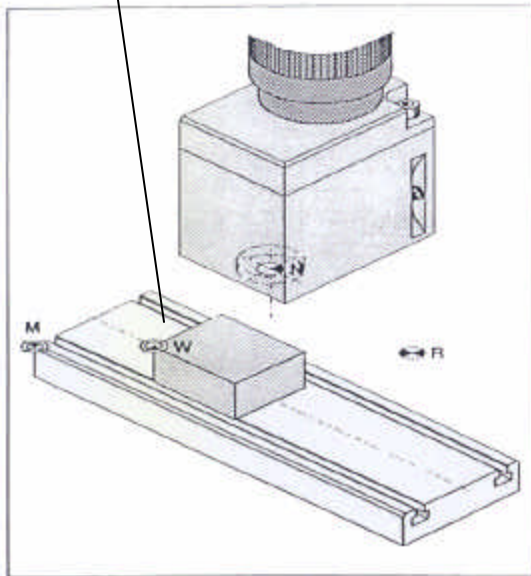
OR

7. 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".

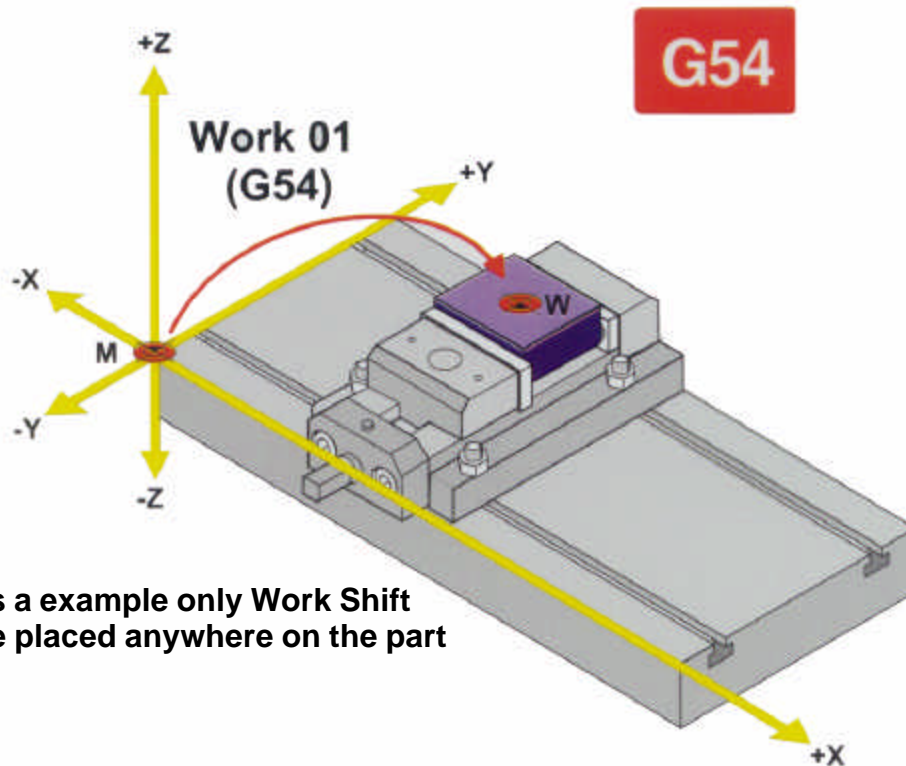
Required after every power failure.

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.



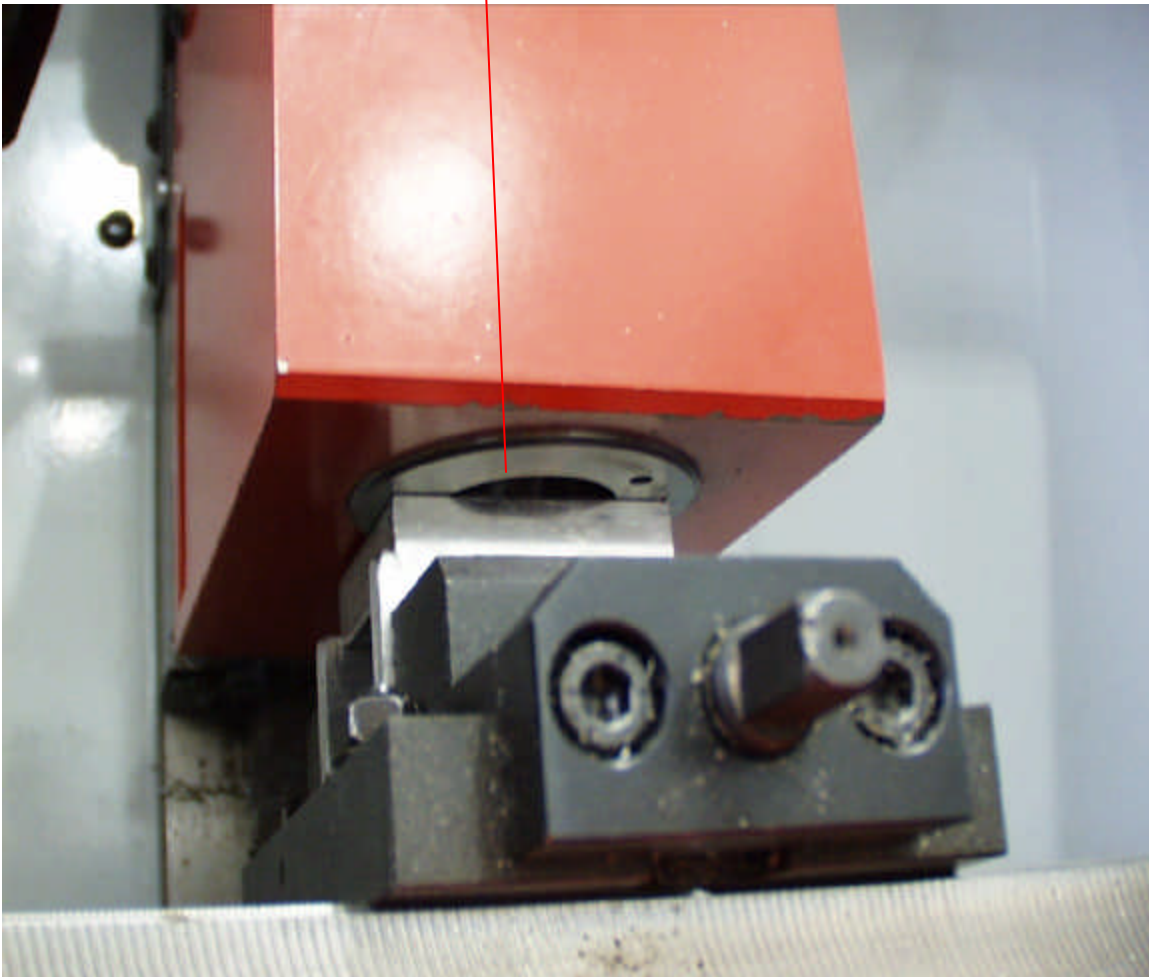
This is a example only Work Shift can be placed anywhere on the part

Work Shift:

1. Press F1 then F6 for JOG mode
2. Jog the Spindle nose to the top of the Work Piece & touch using the # keys 1,2,4,6,8,9.

(Use Feed -, + or Steps to approach at a slower feed rate)

(Use piece of paper between nose and Work Piece)



3. Press F12 then F5 for OFFSET
 - Press F5 for W.SHIFT (Example 2)
4. Make sure that X, Y, Z are all 0 if they have values then the Work Shift will be taken from those values not from the machine 0
5. Press F3 for OFFSET (Example 1)
 - Record the value in the Actual Position Relative Z
6. Press F5 for W.SHIFT (Example 2)
7. Move Cursor to 01 location
8. Recorded value type in Work Coordinates 01(Z) which is G54
Example type (Z 2.463) press Enter

This value is the distance from the top of the Machine bed to the top of the Work Piece.

Example 1

OF 100%

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 **** * F3 F4 F5 F6 F7 20:46:16

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

Example 2

OF 100%

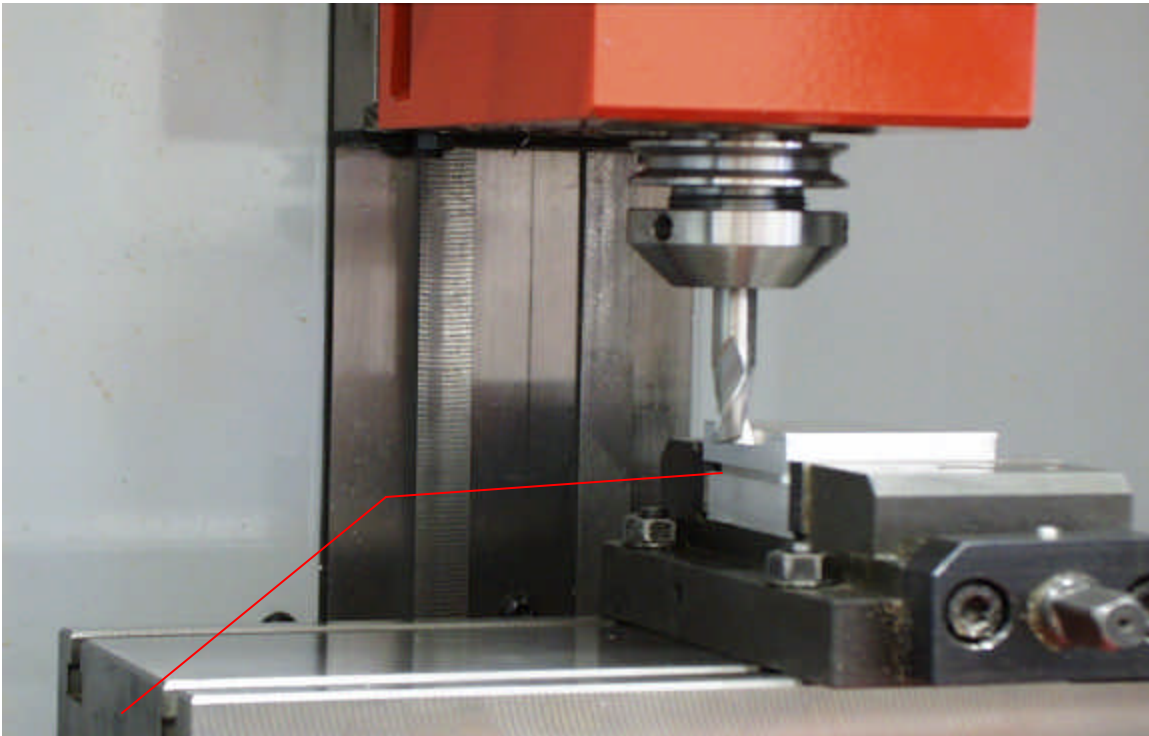
NO.	DATA	NO.	DATA
00 X	0.0000	02 X	0.0000
(EXT) Y	0.0000	(G55) Y	0.0000
Z	0.0000	Z	0.0000
01 X	0.0000	03 X	0.0000
(G54) Y	0.0000	(G56) Y	0.0000
Z	0.0000	Z	0.0000

OS100% T

> _ JOG **** * F3 F4 F5 F6 F7 20:49:22

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

9. Jog Spindle up away from WORK PIECE using 8 on # keys
10. Place a edge finder or a tool in the Spindle (Ex. use 3/8 endmill)
11. Either Follow step 12 or follow step 13 then go to step 14
12. 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)
13. For Scratching: Press F1 then F5 for MDI Mode
 - Press F12 then F4 for PROGRAM display press until top of the screen shows MDI (Program)
 - 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.

14. Press F12 then F5 for OFFSET

- Record the value in the Actual Position Relative X

15. Press F5 for W.SHIFT

16. Move Cursor to 01 location

17. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01(X)

Example: Type 8.463 press Enter

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 **** * F3 F4 F5 20:46:16 F6 F7

(OFFSET) (SETING) (W.SHET) () (OPRT)

OF 100%

WORK COORDINATES

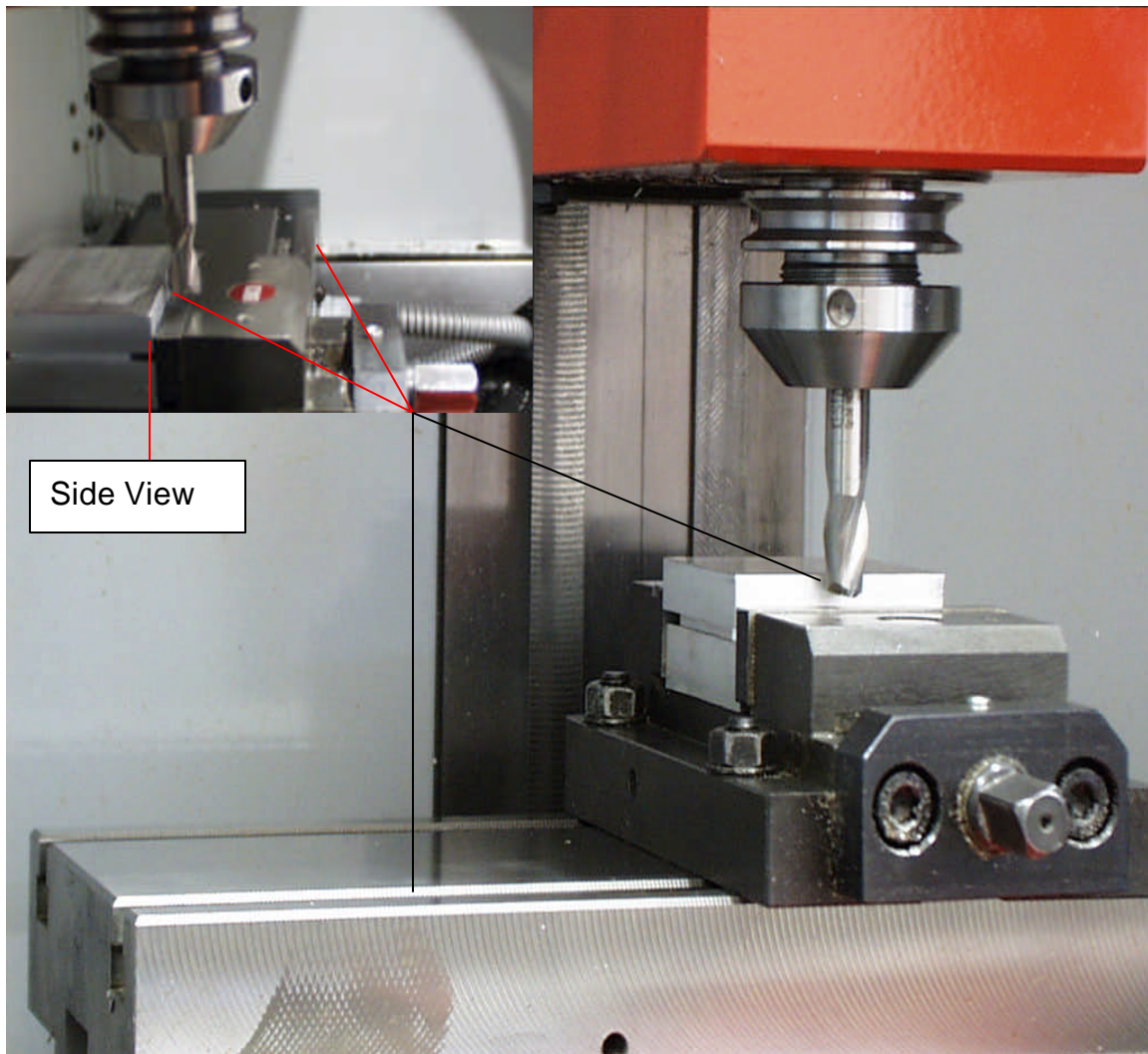
00001 N00000

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 **** * F3 F4 F5 20:47:33 F6 F7

(OFFSET) (SETING) (W.SHET) () (OPRT)

18. Jog Spindle up away from WORK PIECE using Z+
19. 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.

20. Press F12 then F5 for OFFSET

- Record the value in the Actual Position Relative Y

21. Press F5 for WORK

22. Move Cursor to 01 location

23. The Recorded value plus the radius of the tool being used to touch off (3/8) type in Work Coordinates 01 (Y)

Example: Type 8.463 press Enter

OF 100%

OFFSET

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 20:46:16

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

OF 100%

WORK COORDINATES

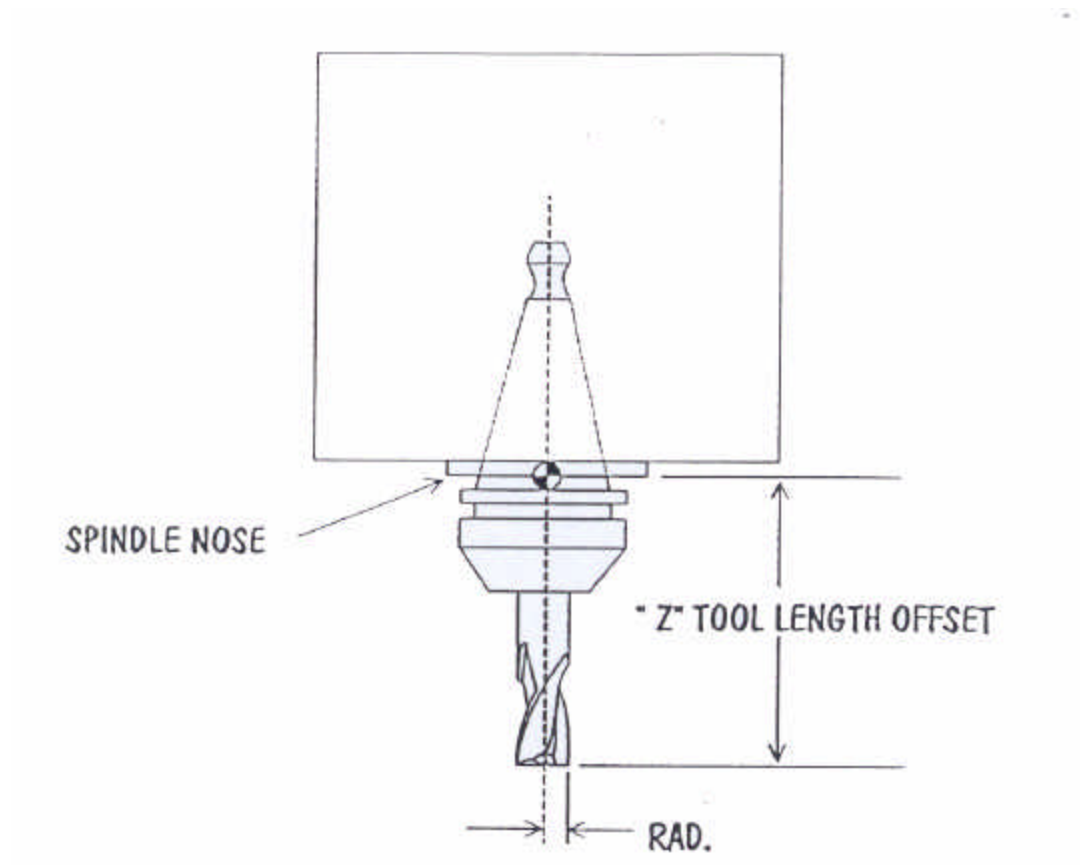
NO.	DATA	NO.	DATA
00 X	0.0000	02 X	0.0000
(EXT) Y	0.0000	(G55) Y	0.0000
Z	0.0000	Z	0.0000
01 X	0.0000	03 X	0.0000
(G54) Y	8.463	(G56) Y	0.0000
Z	0.0000	Z	0.0000

> JOG 20:48:33

(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

> _
11:05:38

JOG

F3
F4
F5
F6
F7

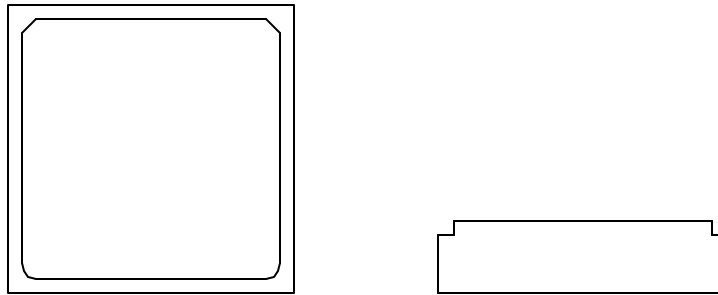
{ OFFSET }
{ SETING }
{ W.SHFT }
{ }
{ (OPRT) }

NOTE: When you use a T the H = Height

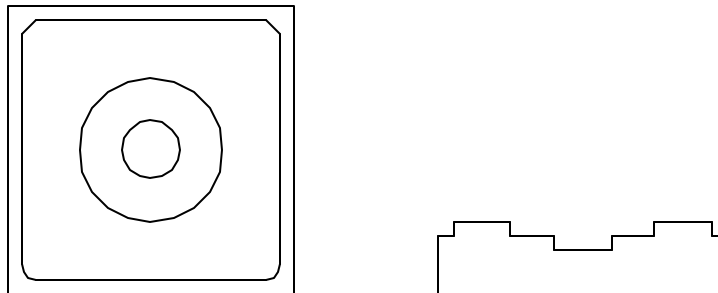
When you use a G41 or G42 the H = Radius

Program Training

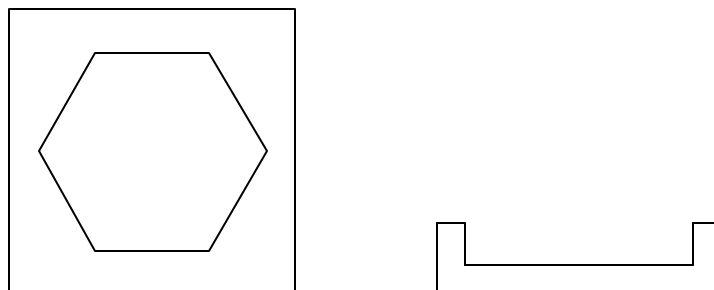
Program O0001



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

G00 Rapid motion

- G01 Linear interpolation in working feed
- G02 Circular interpolation, clockwise
- G03 Circular interpolation, counter-clockwise
- G04 Dwell time, active block by block
- G09 Exact hold

G17 Selection of plane X-Y

- G18 Selection of plane Z-X
- G19 Selection of plane Y-Z
- G20 Dimension in inch
- G21 Dimension in millimeter
- G28 Approach reference point

G40 Deselect miller radius compensation

- G41 Miller radius compensation left
- G42 Miller radius compensation right
- G43 Tool length compensation positive
- G44 Tool length compensation negative

G49 Deselect tool length compensation

- G53 Machine coordinate system

G54 Zero point shift 1

- G55 Zero point shift 2
- G56 Zero point shift 3
- G57 Zero point shift 4
- G58 Zero point shift 5
- G59 Zero point shift 6
- G73 Chip break cycle

G80 Delete drilling cycle (G83 to G85)

- G81 Drilling cycle
- G83 Excavation drilling cycle

G90 Absolute value programming

- G91 Incremental value programming

G94 Feed in inch/min

- G95 Speed with feed in inch/revolution

G97 Spindle speed per minute

G98 Retract to plane of start (drilling cycles)

Survey commands M-CODES: Mostly used


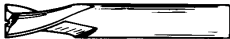
M00	Programmed stop, unconditional
M01	Programmed stop, conditional
M03	Spindle ON clockwise
M04	Spindle ON counter clockwise
M05	Spindle OFF
M25	Open clamping vice
M26	Close clamping vice
M30	Main program end with new start of program
M71	Blow-off ON
M72	Blow-off OFF
M98	Subroutine call-up
M99	Subroutine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

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

Used Addresses

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
M	Miscellaneous function
N	Block number 1 to 9999
O	Program number 1 to 9499
P	Dwell, subroutine
Q	Cutting depth or shift value
R	Radius, retraction height
S	Spindle speed
T	Tool called out
X, Y, Z	Position data
;	Block end

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

F1Z 010	<u>Collet holder</u>	For ESX-25 collets	
225 100	(9.0-10.0mm) \pm 3/8"	ESX 25 COLLETS	
764 308	Acc. to DIN 327, shape B cutting- \varnothing 10 mm / shank- \varnothing 10mm	<u>Slot end mill, HSS</u>	

Program screen & Edit mode

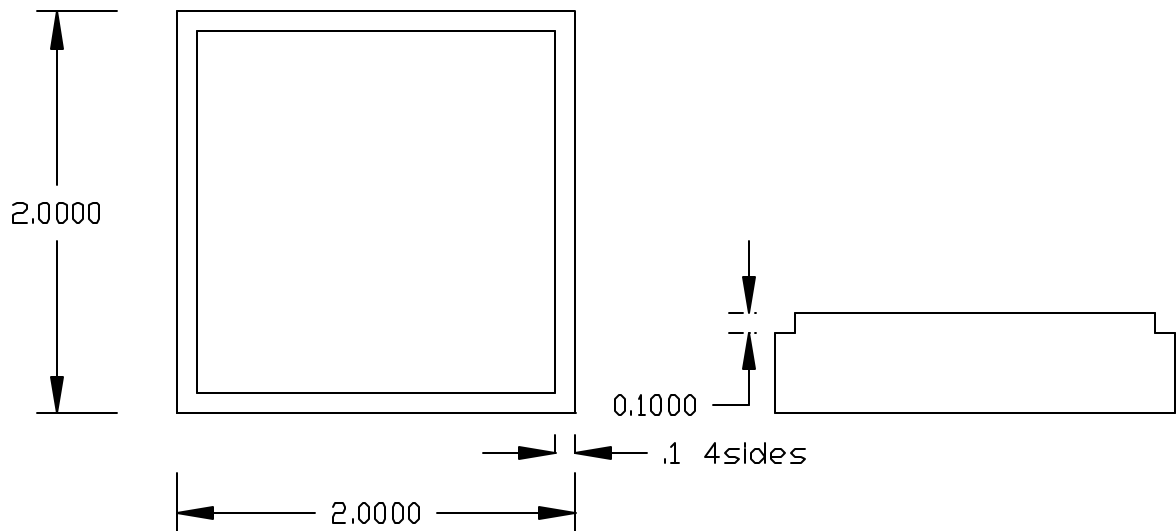
- To edit / change a program / insert new programs & input or output excising programs & offsets

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 1) (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 Z1.5

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 G28 Z1.5

N85 G28 X2.5 Y2.5

N90 M30

2D Simulation PC Keyboard

1. Press F12 then F11 then F3 for the Graph screen to appear

OF 100%

GRAPHIC PATH (PARAMETER)
00000 N00000

AXIS
P=
0

(XY=0, XZ=1, YZ=2)

ANGLE

ROTATION
A=
0

TILTING
A=
0

SCALE
K=
0

MAXIMUM/MINIMUM

X=
0.0000
Y=
0.0000
Z=
0.0000

I=
0.0000
J=
0.0000
K=
0.0000

START SEQ. NO. N=
0

END SEQ. NO. N=
0

> _

OS 100% T

JOG

21:41:41

F3
F4
F5
F6
F7

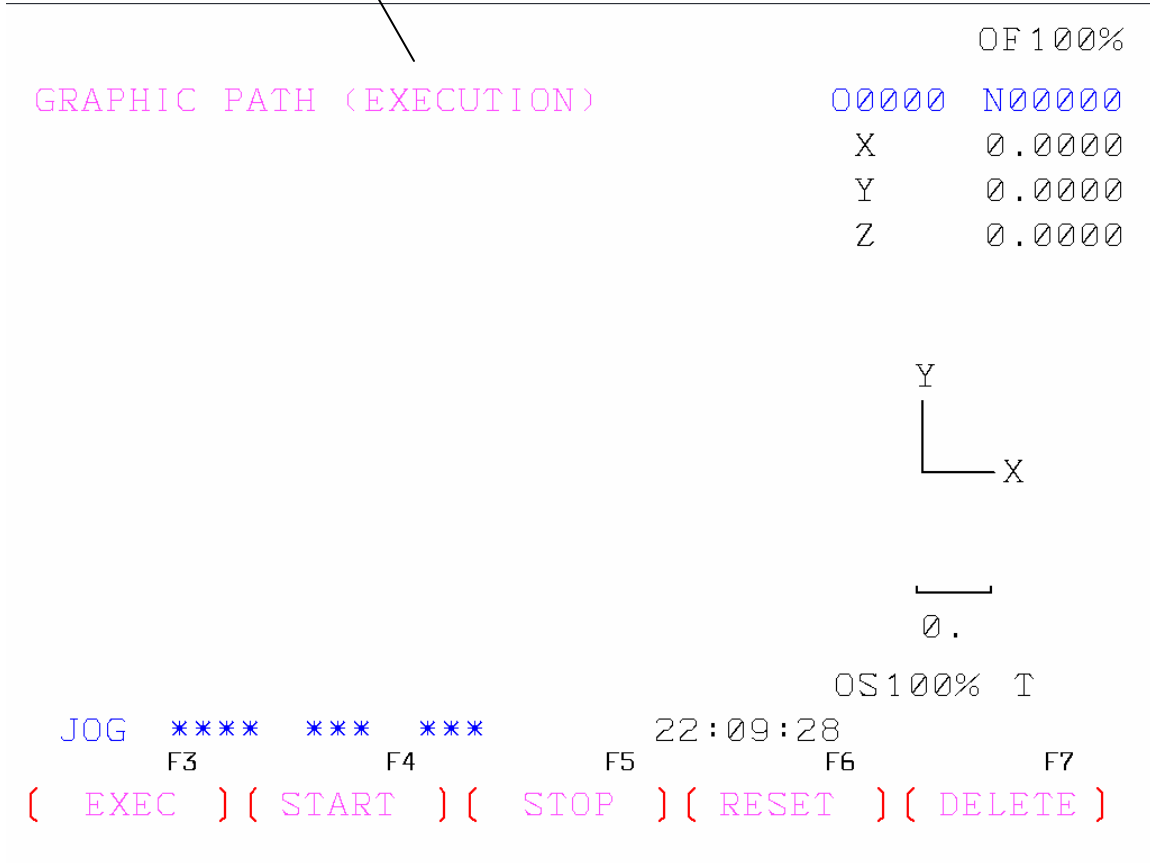
{
PARAM
}
{
EXEC
}
{
SCALE
}
{
POS
}
{

}
>

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 F3 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.

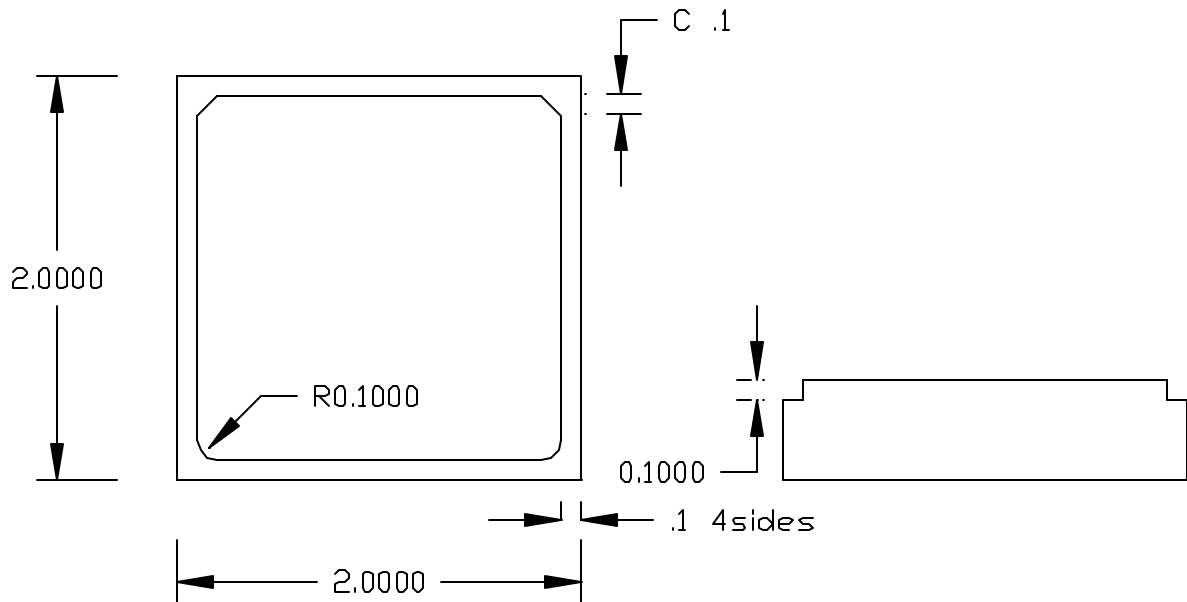
10. Now press Enter on the # Keys for Cycle start or F4 for Start and you will see the tool movements of the program

- **Changing I/O to floppy drive** (Only need to do this once stays default)
 1. Press F1 then F4 for **EDIT**
 2. Press F12 then F6 for **SYSTEM**
 3. Page down until you see Parameter (Setting 1)
 4. Cursor down to (I/O)
 5. Type A (for the floppy drive) then press Enter key

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

- **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 O and program number (O0002) or (O2)
 3. Press the F11
 4. Press F5 Punch then press F7 Exec
- **Output Offsets from Fanuc software to Drive unit**
 1. Press F12 then F5 **Offset**
 2. Press F7 (OPRT)
 3. Press the F11
 4. Press F5 Punch then press F7 Exec
- **Input Program into Fanuc Software from Drive unit**
 1. Press F12 then F4 for **Program**
 2. Type program number to be read
Example: letter O and program number (O0002) or (O2)
 3. Press the F11
 4. Press F4 Read then press F7 Exec
- **Input Offsets into Fanuc Software from Drive unit**
 1. Press F12 then F5 **Offset**
 2. Press F7 (OPRT)
 3. Press the F11
 4. Press F4 Read then press F7 Exec

Program O0001 (C & R)



O0001 (Demo 1) (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

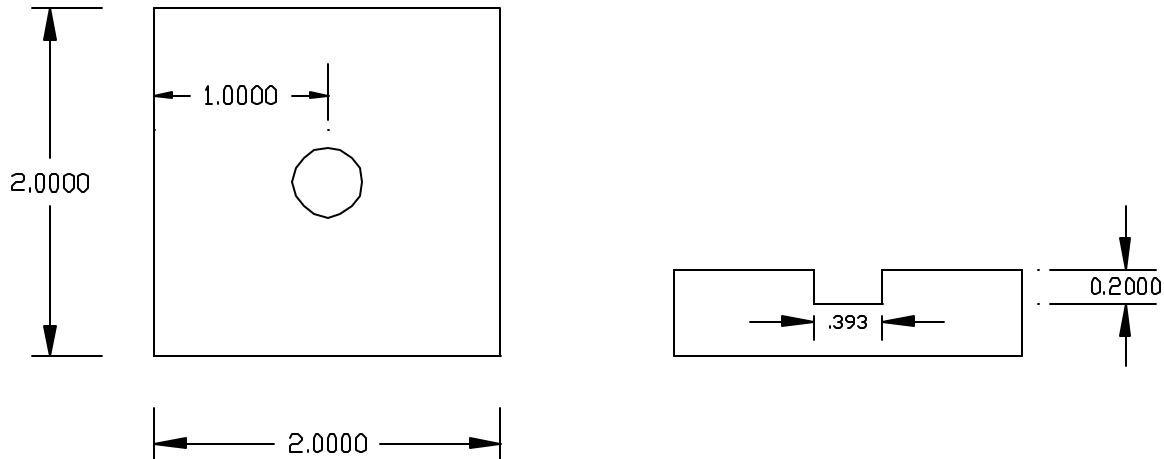
N75 G0 G40 X-1

N80 G28 Z1.5

N85 G28 X2.5 Y2.5

N90 M30

Program Q0002 (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**

O0002 (Demo 2) (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

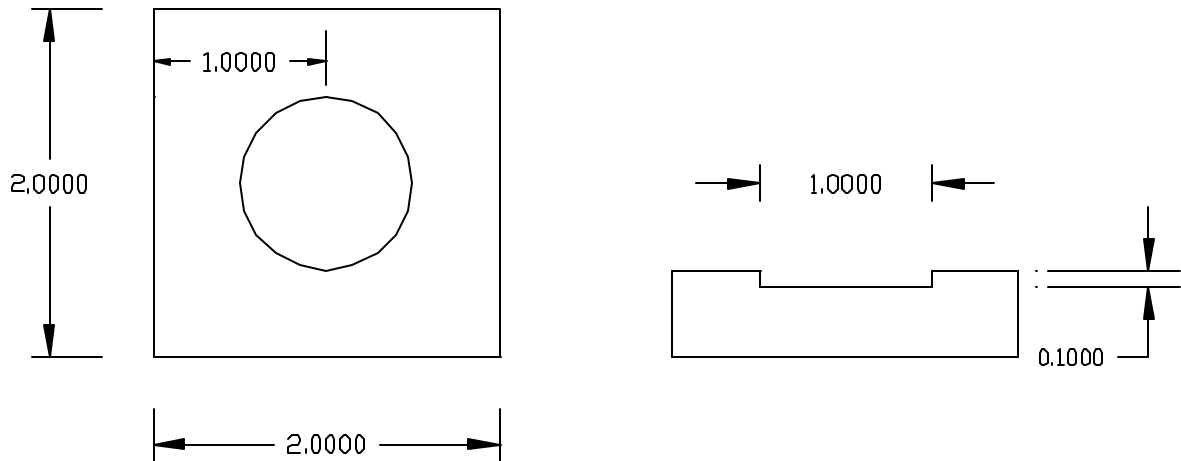
N40 G80

N45 G28 Z1.5

N50 G28 X2.5 Y2.5

N55 M30

Program O0003 (I & J)



O0003 (Demo 3) (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)

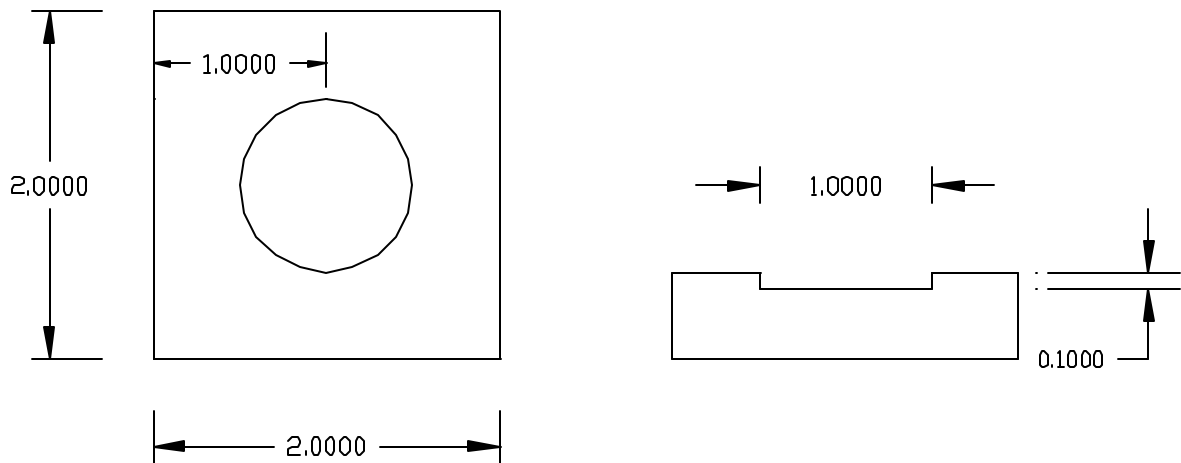
N55 G0 G40 X1

N60 G28 Z1.5

N65 G28 X2.5 Y2.5

N70 M30

Program O0003 (R)



O0003 (Demo 3) (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 G28 Z1.5

N65 G28 X2.5 Y2.5

N70 M30

1. To make all programs tie together or all programs O0001 thru O0003 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 O0004
 - Program O0001, O0002(R), O0003 must all have M99 at the end to link together

Program O0004 (Main Program)

O0004 (Tie Prog. 1,2,3 together)

N5 G54

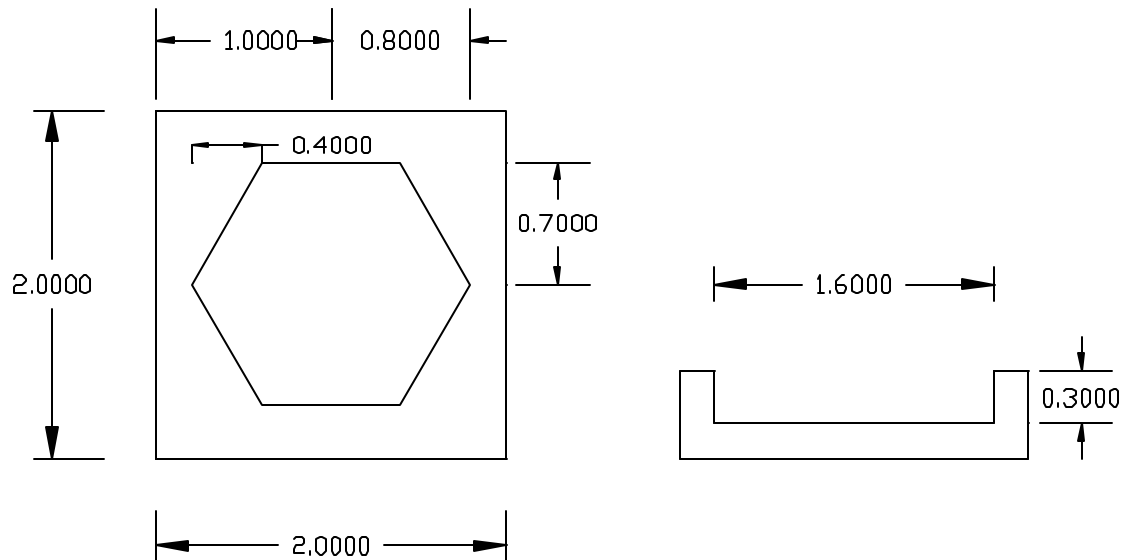
N10 M98 P010001

N15 M98 P010002

N20 M98 P010003

N25 M30

Program O0005 (Pocket Milling) (Making a Cycle)



O0005 (Demo 5) (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 P030006

N45 G0 G28 Z1.5

N50 G28 X2.5 Y2.5

N55 M30

Program O0006 (Sub for program 5)

O0006 (Sub Prog. for Prog. 5)

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