



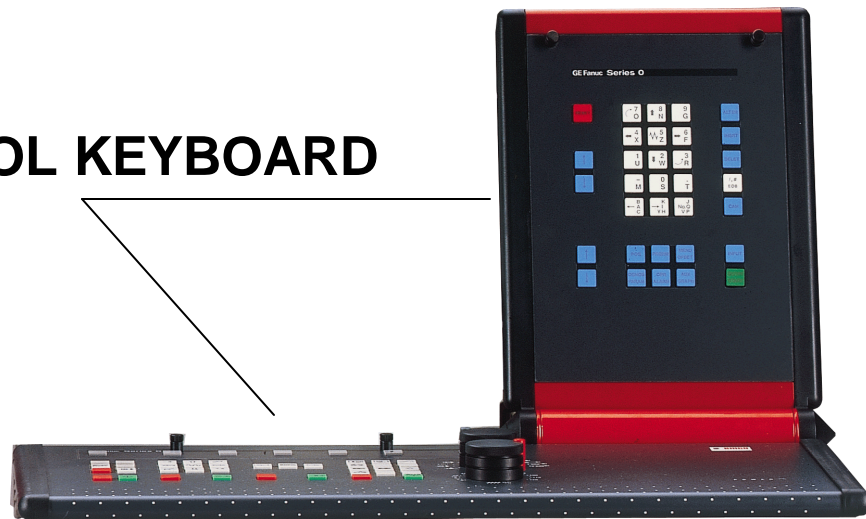
GE FANUC 21 105 MILL TRAINING GUIDE

Training Index

Control Keyboard.....	Pg 1
• Fanuc 21 Control	
• Machine Control	
Fanuc 21 Screen	Pg 2
Fanuc 21 Keys	Pg 3
• Cursor Movement Keys	
• Change Keys	
• Store Keys	
Data Input Keys	Pg 4
• Function Keys (Display Keys)	
• Soft Key Module	
Machine Keys	Pg 5
• Machine Function Keys	
Direction Keys	Pg 6
• Spindle Override Keys	
• Accessory Functions	
Mode Dial	Pg 7
• Feed Override Dial	
Pc Keyboard Keys	Pg 8
Referencing the Machine	Pg 9
Work Shift Description (Picture)	Pg 10
Work Shift (How to do Z Work Shift)	Pg 11
Work Shift (How to do X Work Shift)	Pg 13
• Manually starting the Spindle	
Work Shift (How to do Y Work Shift)	Pg 15
Tool Offset Description (Picture)	Pg 17
Tool Offset (How to do Tool Offsets)	Pg 18

Program Training & Tool listing.....	Pg 19
Inserting a New Program	Pg 20
• Calling a Existing Program up	
• Insert a word	
• Insert a End of Block	
Delete a Program	Pg 21
• Delete all Programs	
• Delete a word	
• Delete a Block	
Cancel word	Pg 22
• Alter a word	
• Search for number Block	
• Search for word	
G Codes	Pg 23
M Codes	Pg 24
• Used Addresses	
Program 1 (Contour Out Side)	Pg 26
2D simulation (Setup)	Pg 27
Input & Output the Programs & offsets thru the Fanuc Software	Pg 29
Program 1 (C & R)	Pg 30
Program 2 (Drilling)	Pg 31
Program 3 (I & J's)	Pg 32
Program 3 (R's)	Pg 33
Program 4 (Main Program) using M98 (sub programs)	Pg 34
Program 5 (Main for Pocket Milling)	Pg 35
Program 6 (Sub program for program 5)	Pg 36

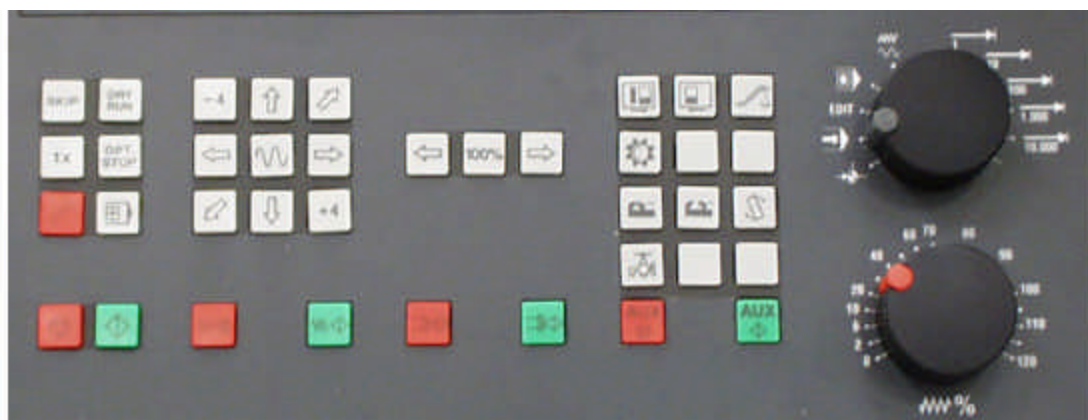
CONTROL KEYBOARD



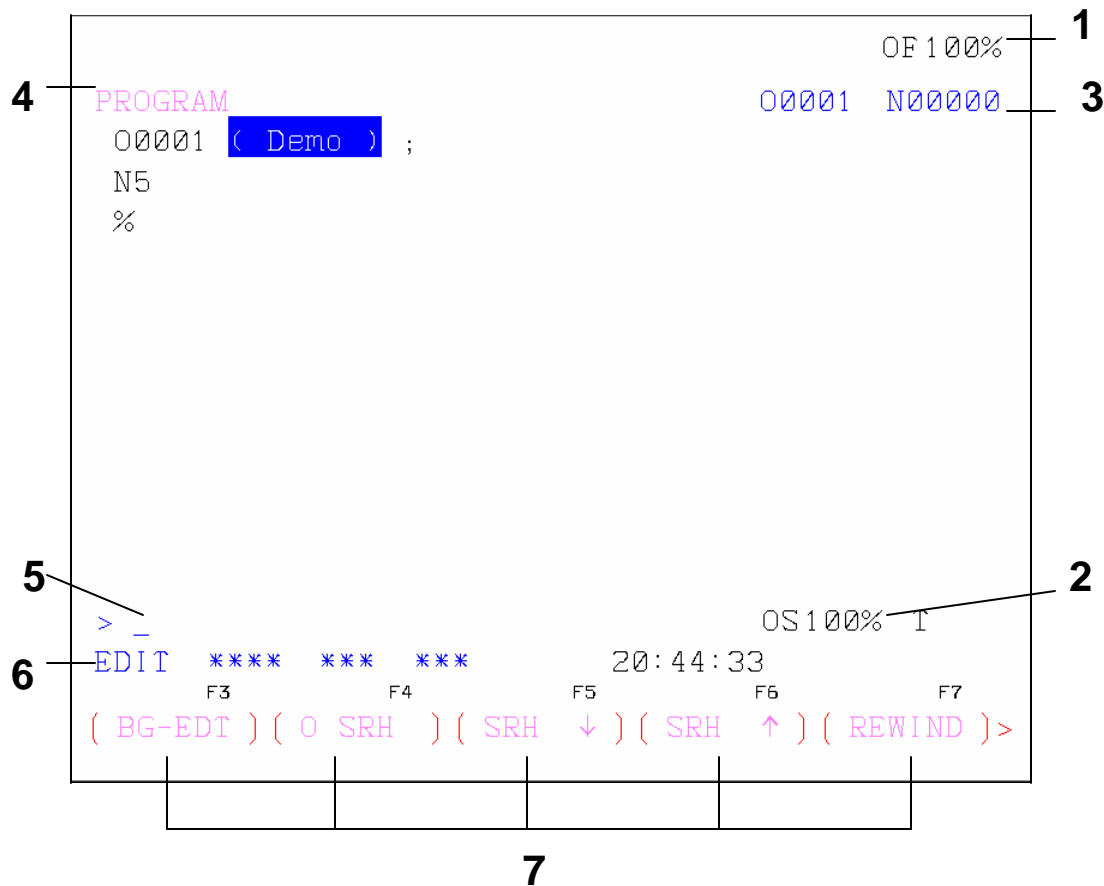
FANUC 21 CONTROL



MACHINE CONTROL



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



**RESET = cancels most alarms, resets program,
interrupts programs**

CURSOR MOVEMENT KEYS



CURSOR UP = moves cursor up



**CURSOR DOWN = moves cursor down, search
function, program call up**



PAGE UP = moves one page up



PAGE DOWN = moves one page down



CURSOR RIGHT = moves cursor right



CURSOR LEFT = moves cursor left

CHANGE KEYS



ALTER = alter word (replace word)



INSRT = insert word, create new program



DELET = deletes word / block or programs



EOB = end of block



CAN = deletes entries in the address one by one



INPUT = input offsets / words or numbers

DATA INPUT KEYS



Press a button for a letter / number needed. Use Shift for the second letter or symbol on that button.

FUNCTION KEYS (DISPLAY KEYS)



POS = displays actual, relative, machine positions



PROG = displays program, library page



OFFSET/ SETTINGS = displays wear, geometry, work shifts pages



SYSTEM = displays parameters, diagnostic pages

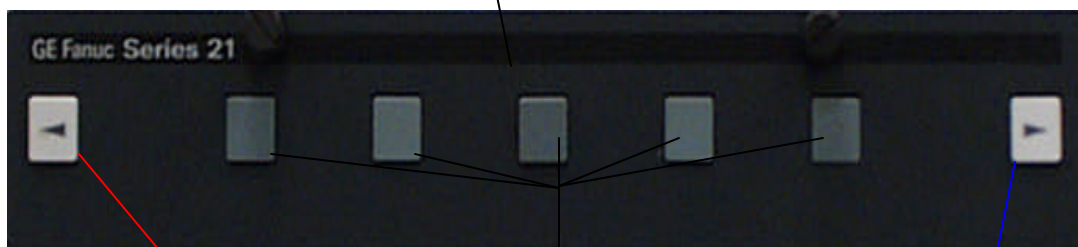


MESSAGE = displays operator & alarm messages



GRAPH = displays 2-d graph simulation

SOFT KEYS



SCROLL BACK

SOFT KEYS

PAGES OVER

MACHINE KEYS

MACHINE FUNCTION KEYS



= Press skip any block lines with (/ Slash) before block number will be skipped



= Test run without spindle on (remove raw material from chuck)



= (Single piece) for continuous mode active only on automatic material loading



= (Optional stop) for programs with (m1)



= (Reset) cancels most alarms, resets program, interrupts programs



= (Single block) reads one block line at a time



= (Cycle stop) program hold, feed hold



= (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

+4 & -4 = Additional axis

Feed stop (Red) / Feed start (Green) works all modes but EDIT & ZRN



SPINDLE OVERRIDE KEYS

Arrow key pointing right increase the Spindle speed (120% high)

Arrow key pointing left decrease the Spindle speed (50% low)

100% key jumps speed to 100%

Spindle stop (Red) / Spindle start (Green)

Works all modes except EDIT & ZRN (Reference)



ACCESSORY FUNCTIONS

Arrow right door open

Arrow left door closed

Press for Rotary axis Indexing

Press once vise closed

Press once vise open

Turret Index

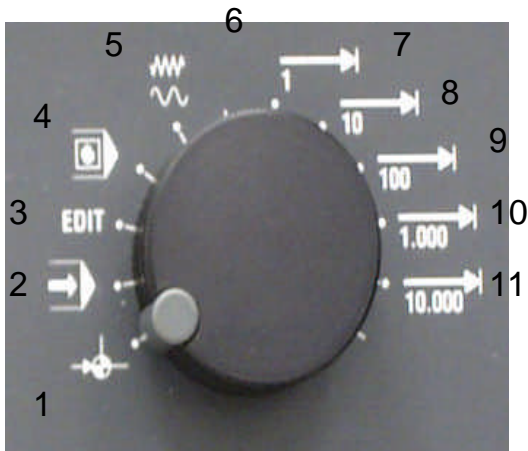
Press once coolant on

Press again coolant off

Press auxiliary drives on (Green)

Press auxiliary drives off (Red)

MODE DIAL



- (1) REF = Zero / Reference or Home mode
- (2) MEM = Automatic mode for running a program
- (3) EDIT = Edit mode for program changes or entering a new program
- (4) MDI = Manual Data Input mode for manually running the machine
- (5) JOG = Manual moving the axis in X, Y or Z
- (6) STEPS = Incremental feed movements
- (7) STEPS = .0001 or tenths
- (8) STEPS = .001 or thousands
- (9) STEPS = .010 or ten thousands
- (10) STEPS = .100 or hundred thousands
- (11) STEPS = .100 or hundred thousands

FEED OVERRIDE DIAL





Controls feed for jogging in the X, Y, Z Axis.
Overrides from 0% to 120% of the
programmed feed rate or the rapid rate

- The machine functions are active only with NUM LOCK on

Turning the Machine On/Entering Fanuc Software

Referencing the Machine

1. Press the **AUX** button  (This turns on the Auxiliary Drives)
2. Press the **Agreement** button 
Open door then **Shut** door (This Initialize the safety circuits on the Machine door)
3. Move the MODE dial to REF position also know as Reference make sure your feed rate is not on "0"



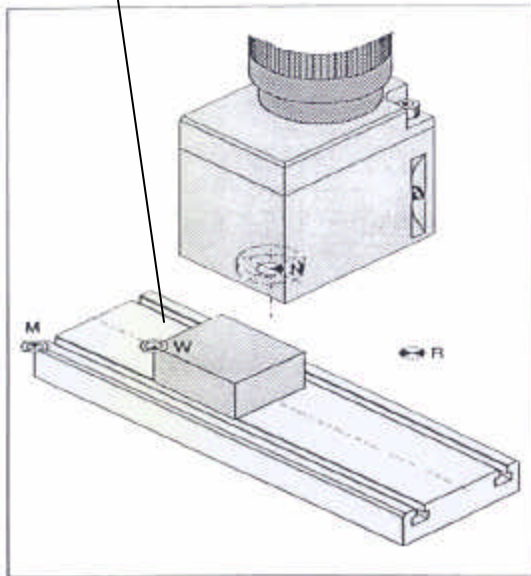
4. Make sure door is closed
5. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
6. Press the X- (arrow pointing left) this references the X axis
7. Press the Y- (arrow pointing left) this references the Y axis



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

WORK SHIFT

Pages 10 – 18 is setting the Work shift & offsets to the lower left corner & the top of the part with the Tool stump



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".

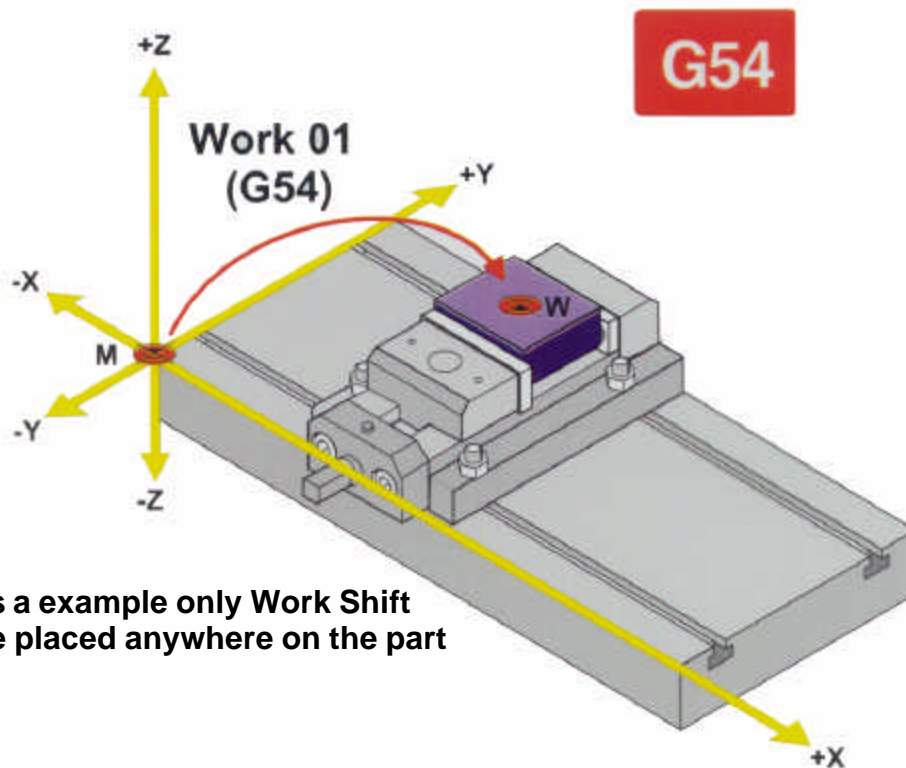
Required after every power failure.

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

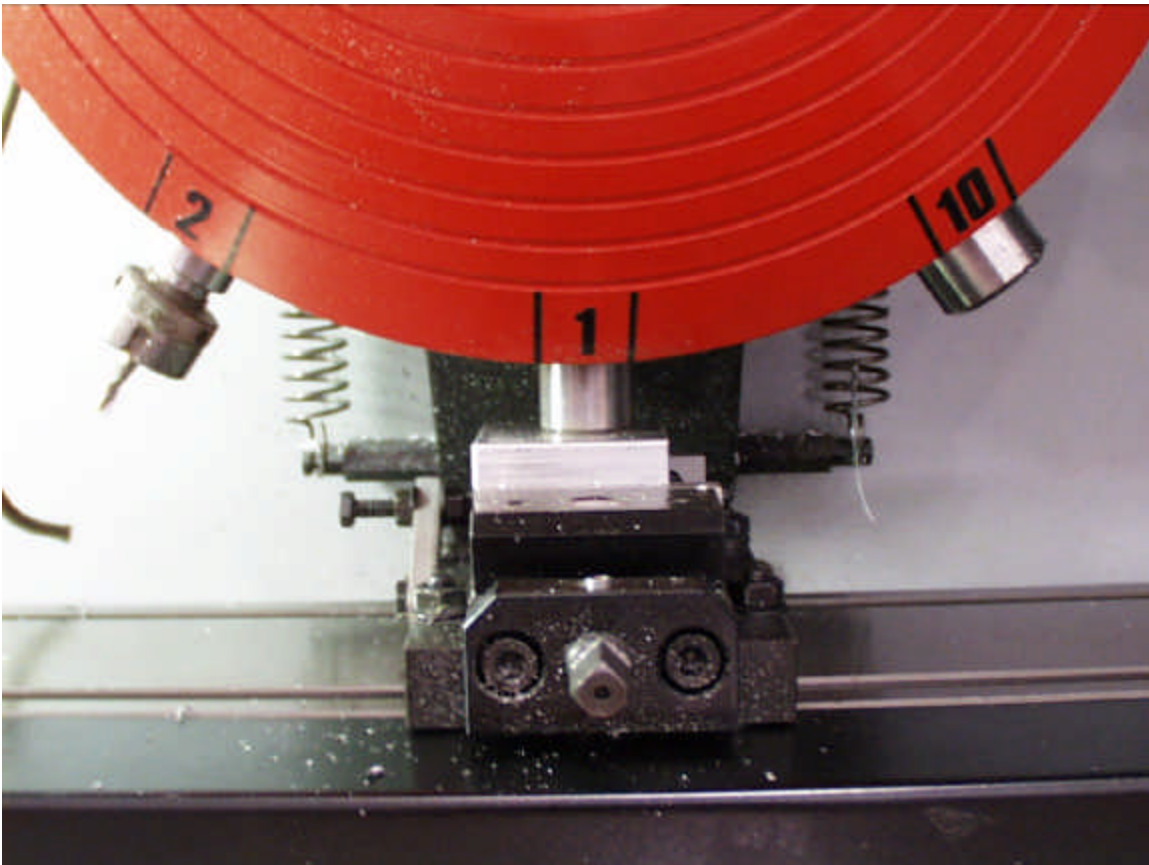
Work Shift:

1. Move the MODE dial to JOG position
2. Jog the Tool Stump to the top of the Work Piece & touch using the Direction keys.



(Use Feed Dial or Steps to approach at a slower feed)

(Use piece of paper between nose and Work Piece)



3. Press the OFFSET/SETT button

OFFSET
SETT.

- Press the W.SHIFT

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 the OFFSET Soft key (Gray Button)

- Record the value in the Actual Position Relative Z (Example 1)

6. Press the W.SHFT Soft key (Gray Button) (Example 2)

7. Move Cursor to 01 (G54) location so Z is highlighted

8. Recorded value type in Work Coordinates 01 (G54) Z

Example: Type 2.463 press Input button

INPUT

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

Example 1


		OF 100%	
		O0001 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 ****		20:46:16	
(OFFSET) (SETING) (W.SHFT) () (OPRT)			

Example 2

		OF 100%	
		O0001 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
> _		OS100% T	
JOG ****		20:49:22	
(OFFSET) (SETING) (W.SHFT) () (OPRT)			



9. Jog Spindle up away from WORK PIECE using Z+

10. Either follow step 11 or follow step 12 when finished go on to step 13

11. Index to a edge finder or tool (Example uses 3/8 end mill) Press 


- Jog the Tool to the left side of the Work Piece & touch using the Direction keys. (Use Feed Dial or Steps to approach at a slower feed)

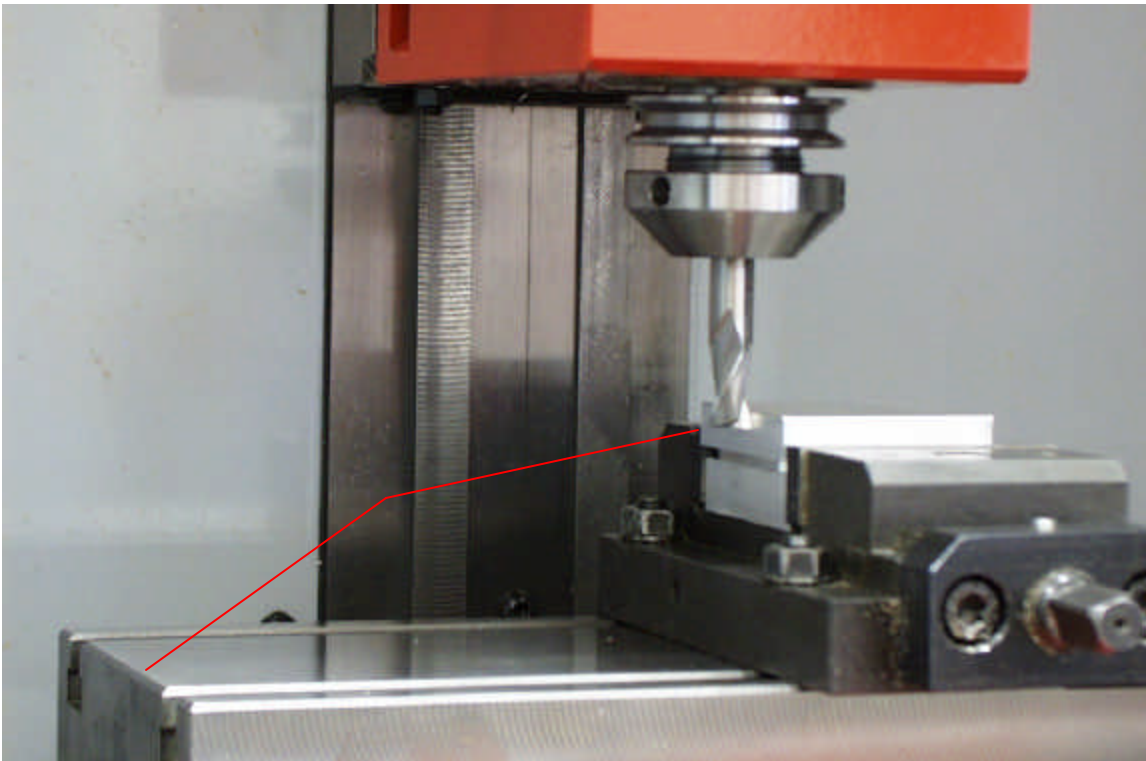
12. For Scratching move MODE Dial to MDI

- Press the PROGRAM display button  until top of the screen shows MDI (Program)
- Type S1000, M03, T1, M06 



S=RPM M03=Spindle on Clockwise T1=Tool Position M06=Index

- Then press CYCLE START  (**Door must be closed**)
- Move MODE Dial to Jog then Jog the Tool to the left side of the Work Piece & touch using the Direction keys.



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

13. Press the OFFSET/SETT button



- Example 1

- Record the value in the Actual Position Relative X

14. Press the W.SHFT Soft key (Gray Button) (Example 2)

15. Move Cursor to 01 (G54) location highlight X

16. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (G54) X

Example: Type 8.463 press Input button



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	

> _ JOG **** * F3 F4 F5 F6 F7

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

OS100% T

20:46:16

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

> _ JOG **** * F3 F4 F5 F6 F7

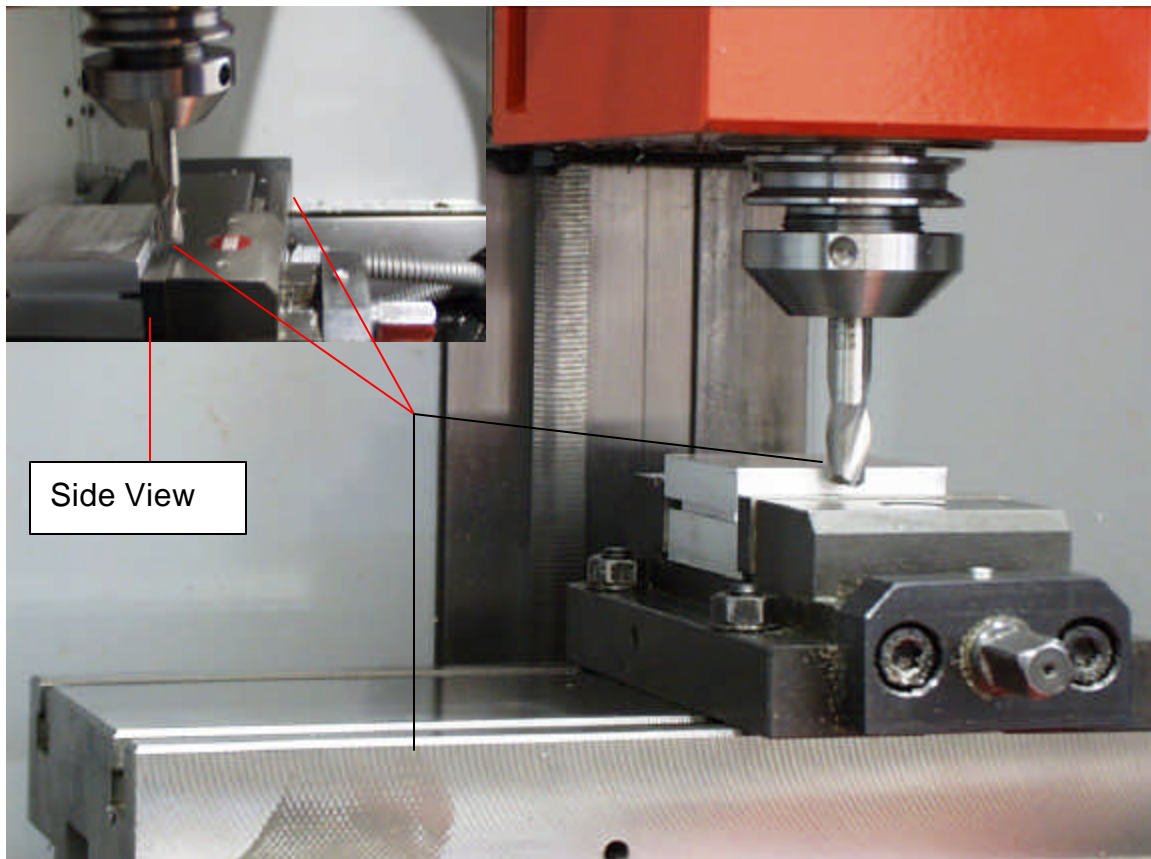
(OFFSET) (SETING) (W.SHFT) () (OPRT)

OS100% T

20:47:33

17. Jog Spindle up away from WORK PIECE using Z+
18. Jog the Tool to the Front of the Work Piece & touch using the Direction keys.

(Use Feed Dial or Steps to approach at a slower feed)



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


19. Press the OFFSET/SETT button  (Example 1)

- Record the value in the Actual Position Relative Y

20. Press the W.SHFT Soft key (Gray Button) (Example 2)

21. Move Cursor to 01 (G54) location highlight Y

22. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (G54) Y

Example: Type 8.463 press Input button 

This value is the distance from the left side of the Work Piece to the left side of the Machine Bed.

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	

> _ JOG
*** **
*** **
*** **
20:46:16
OS100% T

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

Example 2

OF 100%

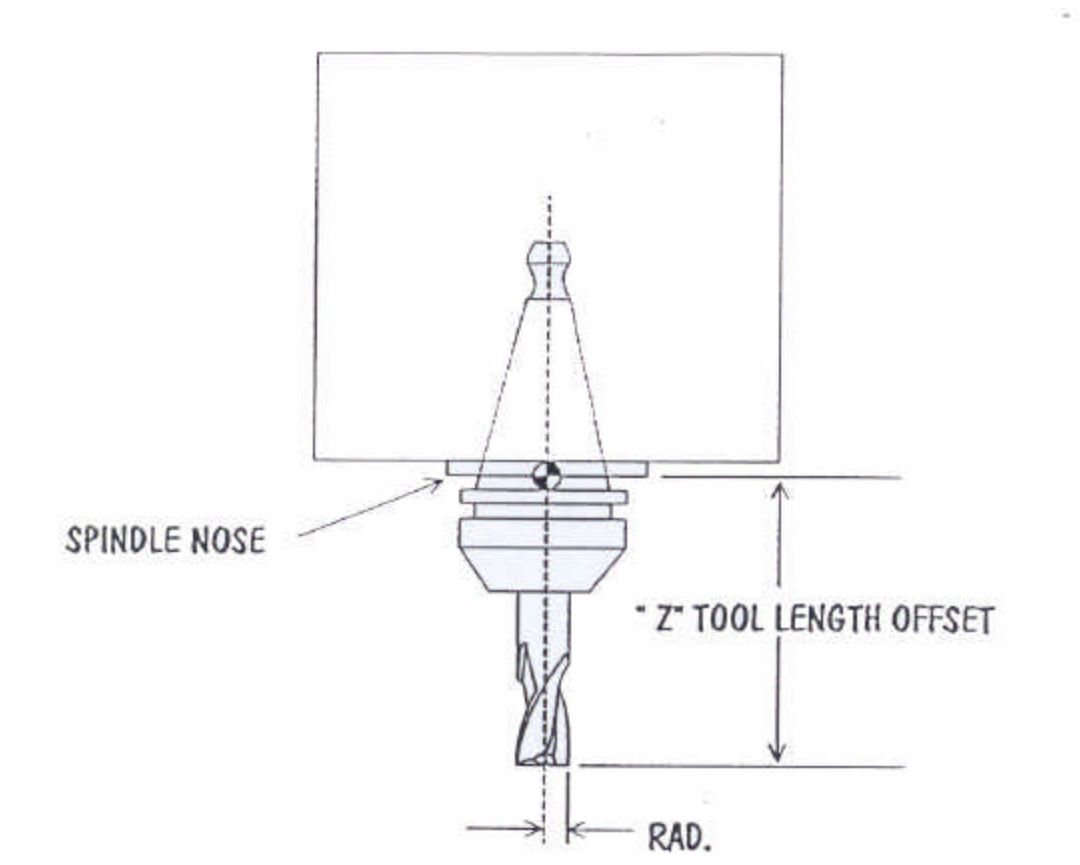
NO.	DATA	NO.	DATA
00 X	0.0000	02 X	0.0000
(EXT) Y	0.0000	(G55) Y	0.0000
Z	0.0000	Z	0.0000
01 X	0.0000	03 X	0.0000
(G54) Y	0.0000	(G56) Y	0.0000
Z	0.0000	Z	0.0000

> _ JOG
*** **
*** **
*** **
20:48:33
OS100% T


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

23. 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 at a slower feed)
2. Press the OFFSET/SETT button 
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 don't need a Radius set for them

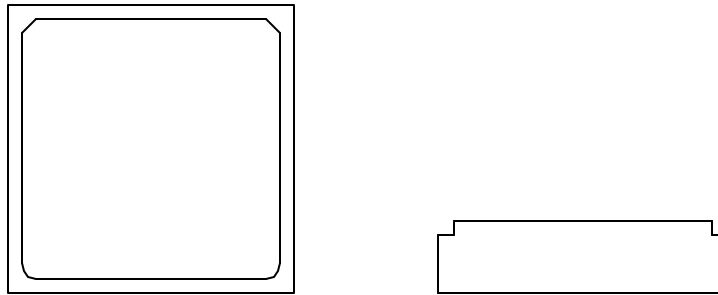
				OF 100%	
OFFSET		00000 N00000			
NO.	DATA	NO.	DATA		
001	0.0000	009	0.0000		
002	0.0000	010	0.0000		
003	0.0000	011	0.0000		
004	0.0000	012	0.0000		
005	0.0000	013	0.0000		
006	0.0000	014	0.0000		
007	0.0000	015	0.0000		
008	0.0000	016	0.0000		
ACTUAL POSITION (RELATIVE)					
X	0.0000	Y	0.0000		
Z	0.0000				
> _				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

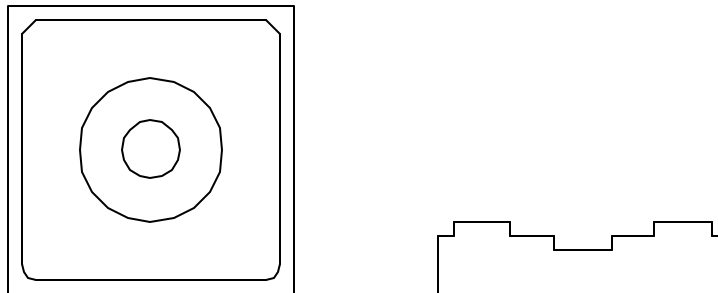
When you use a G41 or G42 the H = Radius

Program Training

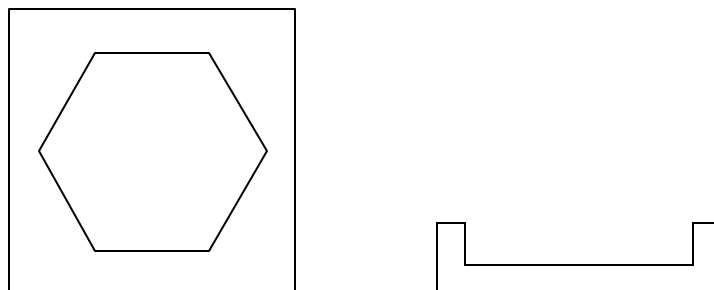
Program O0001



Program O0003




Program O0005




Change the Mode Dial to Edit & Press the  to do functions below
& on the next 2 Pages

- **INSERT A NEW PROGRAM**



1. Press letter o then program number
2. Press insert button 

Example: Q0001 OR Q1

- **CALL A EXISTING PROGRAM UP**

1. Press letter o then program number
2. Press cursor down button 




- **INSERT A WORD**

1. Press letter then number
2. Press insert button  or input 

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 the (EOB) button 
2. Press insert button  or input 

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

Example: N5 G01 X1.00 F.003;

NOTE: IN EDIT & IN PROGRAM USE INSERT OR INPUT.
USE INPUT FOR ALL OTHER APPLICATIONS.

- **DELETE A PROGRAM**


1. Press letter o then program number

2. Press delete button 

Example: O0001 OR O1

- **DELETE ALL PROGRAMS**


1. Press letter o plus the – & 9999

2. Press delete button 

Example: O – 9999

- **DELETE A WORD**

1. Highlight the Word

2. Press delete button 

- **DELETE A BLOCK OR LINE NUMBER**

1. Type the number line and highlight the number line

2. Press delete button 

- **CANCEL MISTYPED WORD (Backspace)**


1. Press cancel button 

HINT: In the ADRS. (Address) at the lower left of the screen is the word & numbers that has been typed in. Before pressing insert or input check if what was typed in is correct. If not press cancel until error is erased and retype


- **ALTER A WORD**

1. Highlight the word needed altered type the change
2. Press alter button 


- **SEARCH FOR NUMBER BLOCK**

1. Press letter n and the number of the block
2. Press cursor down button 

- **SEARCH FOR WORD**

1. Type in word & number
2. Press cursor down button 

- **SEARCH FOR LETTER**

1. Press letter
2. Press cursor down button 

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	Cancel cutter compensation
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation positive
G44	Tool length compensation negative
G49	Cancel tool length compensation
G53	Machine coordinate system (00)
G54	Zero point shift 1 (01)
G55	Zero point shift 2 (02)
G56	Zero point shift 3 (03)
G57	Zero point shift 4 (04)
G58	Zero point shift 5 (05)
G59	Zero point shift 6 (06)
G73	Chip break cycle
G80	Cancel drilling cycle (ALL Drilling Cycles)
G81	Spot or chamfer drilling cycle
G83	Deep hole 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 functions:


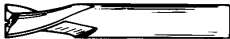
M00	Programmed stop, unconditional
M01	Programmed stop, conditional
M02	Main programmed end
M03	Spindle ON clockwise
M04	Spindle ON counter clockwise
M05	Spindle OFF
M06	Tool change
M08	Coolant ON
M09	Coolant OFF
M19	Orientated spindle stop
M25	Release clamping device
M26	Close clamping device
M30	Main 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 910	<u>Collet holder</u>	For ESX-25 collets	
152 800	(9.0-10.0mm) ± 3/8"	ESX 25 COLLETS	
764 308	Acc. to DIN 327, shape B cutting-ø10 mm / shank-ø10mm	<u>Slot end mill, HSS</u>	

Program screen & Edit mode

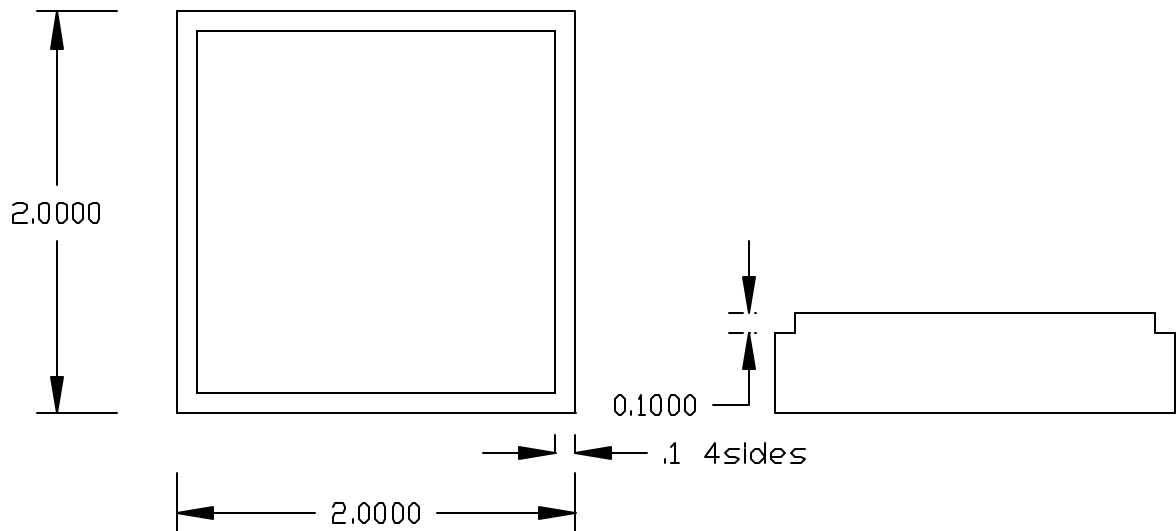
- To edit / change a program / insert new programs & input or output excising programs & offsets

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

1. Press **Graph** button on the Display Keys 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

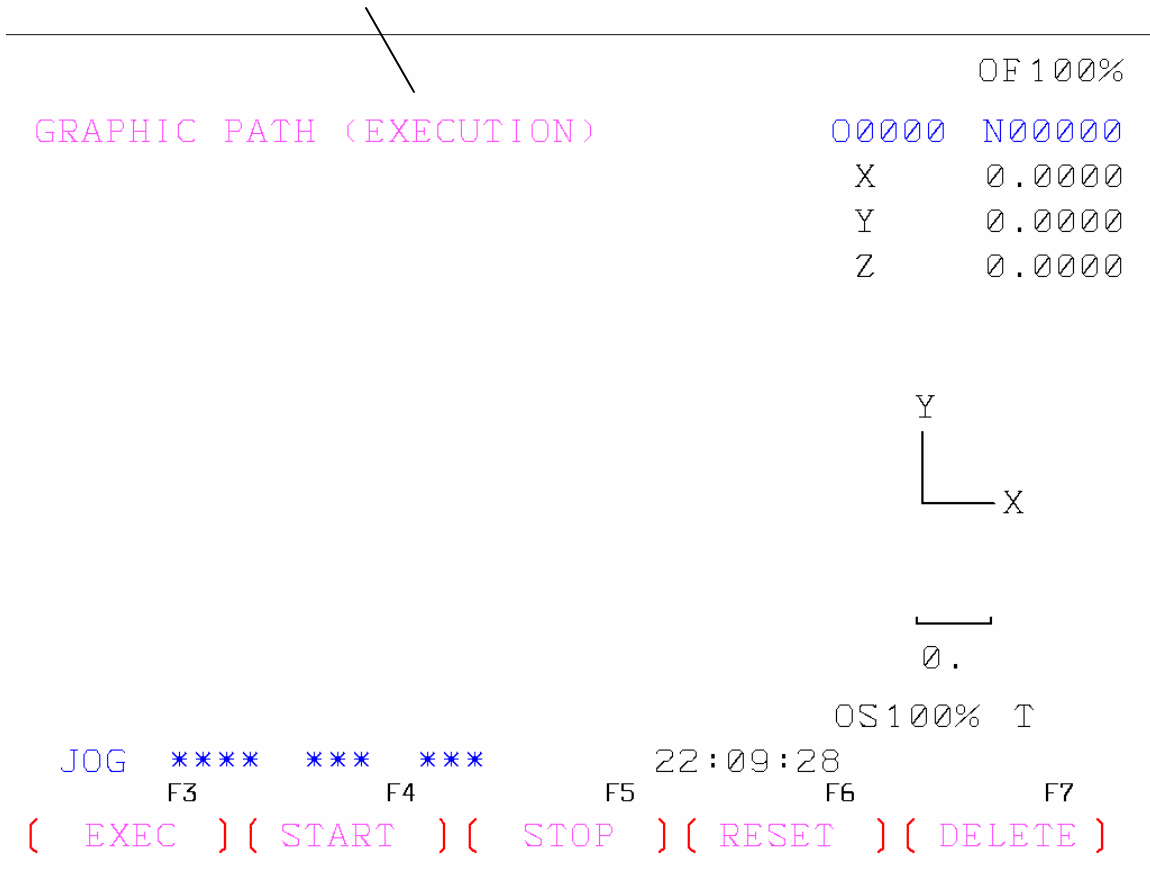
JOG **** * F3 F4 F5 21:41:41 F6 F7

[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 the Soft key **EXEC** 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 **Cycle start** or Soft Key **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. Move the Mode Dial to **EDIT**
2. Press **System** on the display keys
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)**

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

1. Press the **Program** on the display key
2. Type program number to be send out
Example: letter O and program number
(O0002) or (O2)
3. Press the right Arrow key on the Soft keys
4. Press Punch then press Exec

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

1. Press the **Offset/Sett** display key
2. Press (OPRT)
3. Press the right Arrow key on the Soft keys
4. Press Punch then press Exec

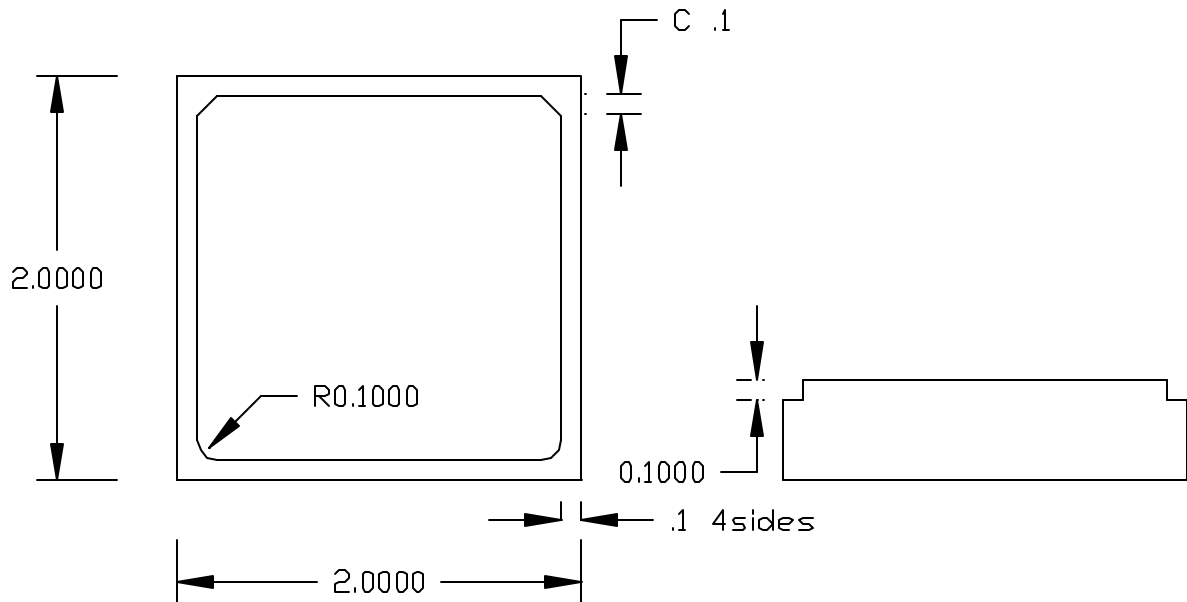
- **Input Program into Fanuc Software from Drive unit**

1. Press the **Program** display key
2. Type program number to be read
Example: letter O and program number
(O0002) or (O2)
3. Press the right Arrow key on the Soft keys
4. Press Read then press Exec

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

1. Press the **Offset/Sett** display key
2. Press (OPRT)
3. Press the right Arrow key on the Soft keys
4. Press Read then press 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

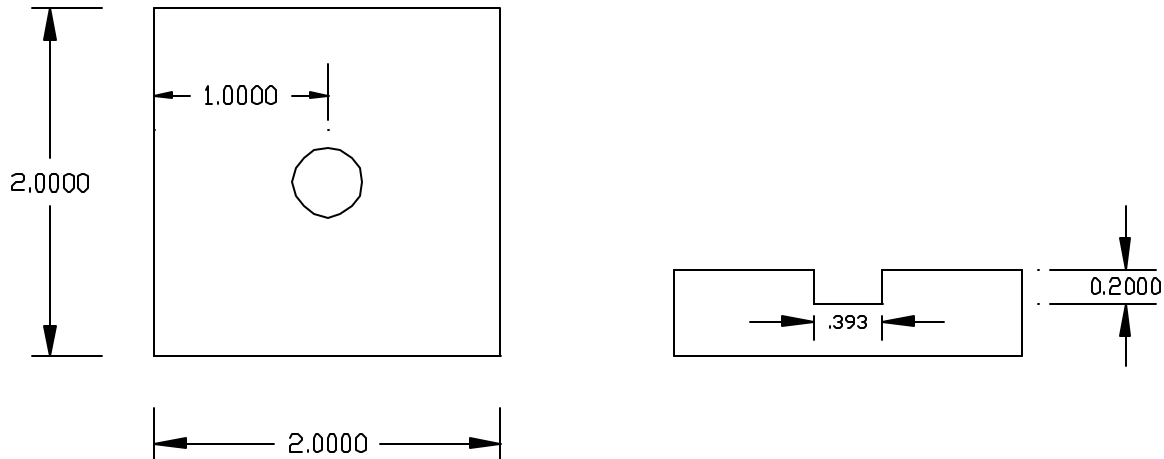
N75 G0 G40 X-1

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

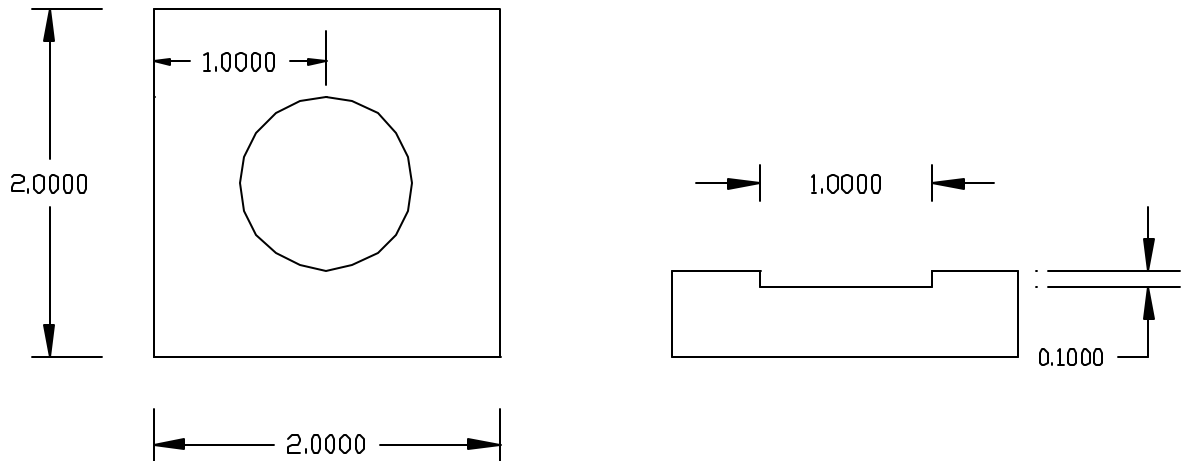
N40 G80

N45 G28 Z1.5

N50 G28 X2.5 Y2.5

N55 M30

Program Q0003 (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)

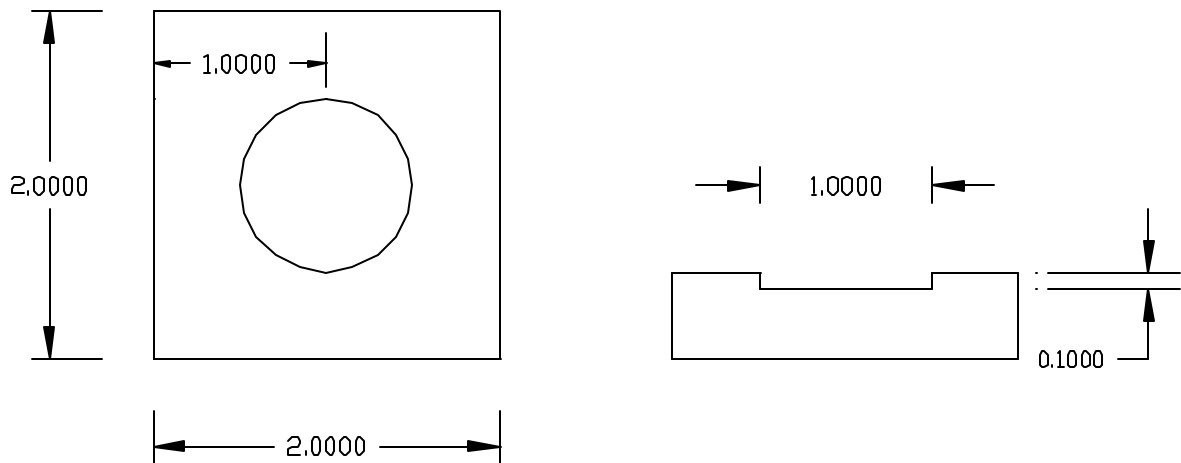
N55 G0 G40 X1

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

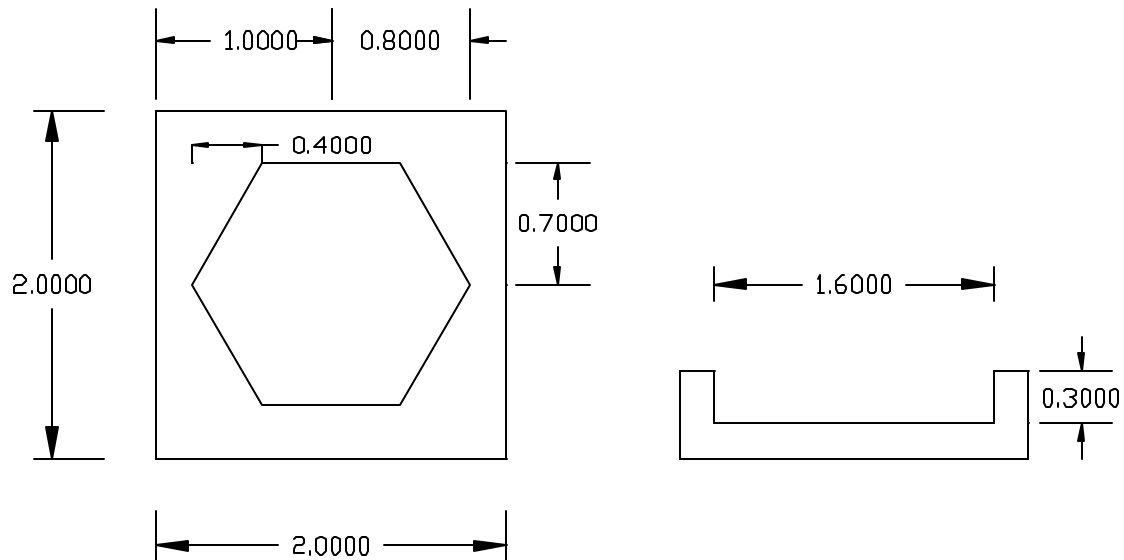
N10 M98 P010001

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