



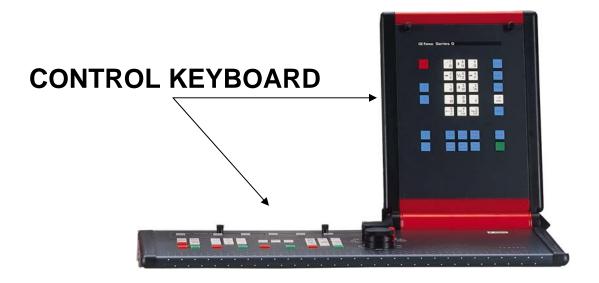
GE FANUC 21 CONCEPT 55 MILL 8 STATION CHANGER TRAINING GUIDE

9/11/04 Version 1 Made by EMCO Authored by Chad Hawk

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

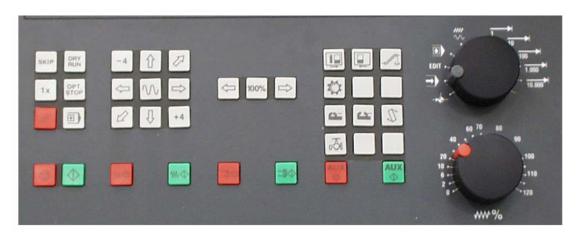
Program Training & Tool listing	Pg 19
Inserting a New Program	Pg 20
Calling a Existing Program up	
Insert a word	
Insert a End of Block	
Delete a Program	Pg 21
Delete all Programs	
Delete a word	
Delete a Block	
Cancel word	Pg 22
Alter a word	
Search for number Block	
Search for word	
G Codes	Pg 23
M 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



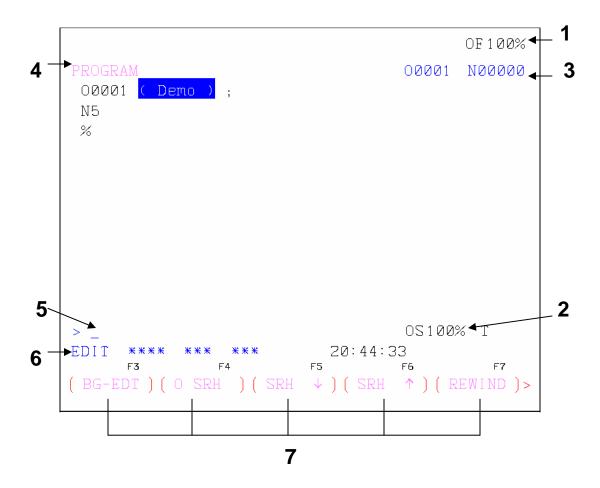
FANUC 21 CONTROL



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	RESET = cancels most alarms, resets program, interrupts programs
	CURSOR MOVEMENT KEYS
<u>†</u>	CURSOR UP = moves cursor up
•	CURSOR DOWN = moves cursor down, search function, program call up
\mathbb{H}	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 = alter word (replace word)
INSRT	INSRT = insert word, create new program
DELET	DELET = deletes word / block or programs
/,# EOB	EOB = end of block
CAN	CAN = deletes entries in the address one by one
INPUT	INPUT = input offsets / words or numbers

DATA INPUT KEYS



PROG

MESSAGE

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

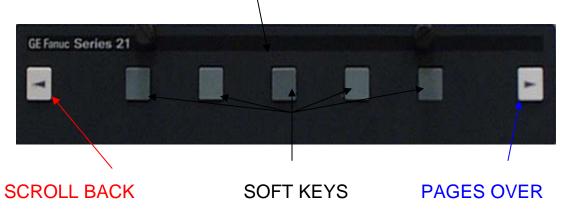
OFFSET/ SETTINGS = displays wear, geometry, work shifts pages

SYSTEM = displays parameters, diagnostic pages

MESSAGE = displays operator & alarm messages

GRAPH = displays 2-d graph simulation

SOFT KEYS



MACHINE KEYS

MACHINE FUNCTION KEYS



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



= Press for test run without spindle on (remove raw material from vise)



= (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



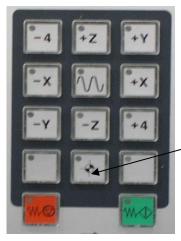
= (Single block) reads one block line at a time



= (Cycle stop) program hold, feed hold



= (Cycle start) program start



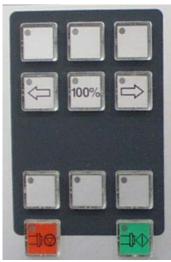
DIRECTION KEYS

These keys control axis directional movements

+4 & -4 = Additional axis

Reference all axis

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



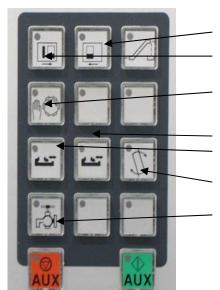
SPINDLE OVERRIDE KEYS

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 closed Arrow left door open

Press once rotary rotate

Press vise open
Press vise closed

Press for Tool turret Index

Press once coolant on Press again coolant off

Press auxiliary drives on (Green)
Press auxiliary drives off (Red)

MODE DIAL



(2) MEM = Automatic mode for running a program



(4) MDI = Manual Data Input mode for manually running the machine



(6) STEPS = Incremental feed movements

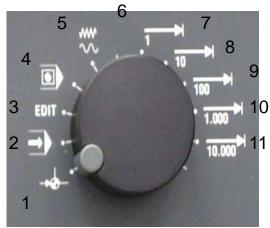
(7) STEPS = .0001 or tenths

(8) STEPS = .001 or thousands

(9) STEPS = .010 or ten thousands

(10) STEPS = .100 or hundred thousands

(11) STEPS = .100 or hundred thousands

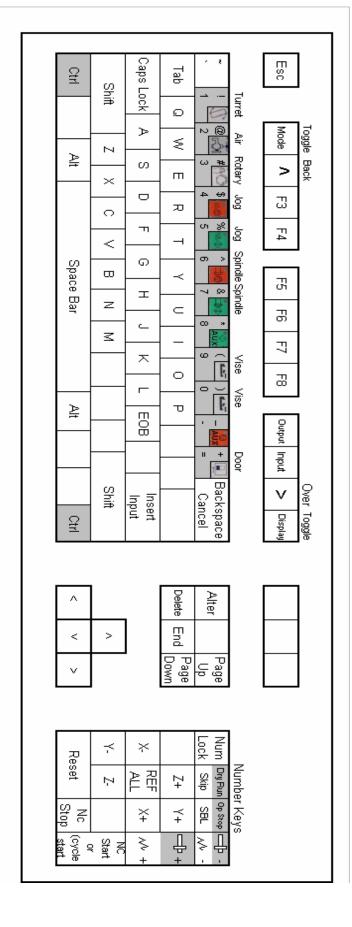


FEED OVERRIDE DIAL



Controls feed for jogging in the X,Y, Z Axis.

Overrides from 0% to 120% of the programmed feed rate or the rapid rate



- 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

ωΝ

- 4. Some automative keys when you press them 1 time this will close/turn off press them again will open/turn on
- 5. F1 is a toggle key for the modes: Zero, Auto, Edit, MDI, Jog and F1 then F11 give Increment Step
- 6. F12 is a toggle key for the Display screens: Position, Program, Offsets, Parameter, Alarm and F12 then F11 then F3 gives Graph
- 7. 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

 \odot

- 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

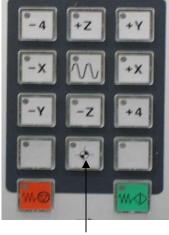
Referencing the Machine

1. Move the MODE dial to REF position also know as Reference make sure your feed rate is not on "**0**"



- 2. Make sure the Door is Closed
- 3. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
- 4. Press the X- (arrow pointing left) this references the X axis

5. Press the Y- (arrow pointing left) this references the Y axis

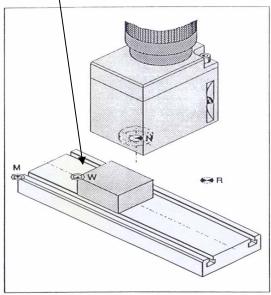


Reference all axis

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 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".

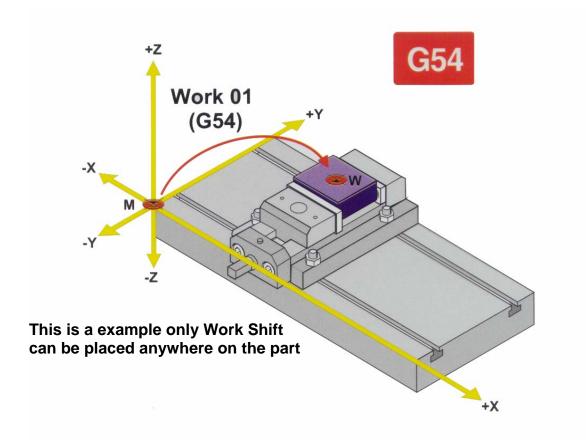
Required after every power failure.

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.



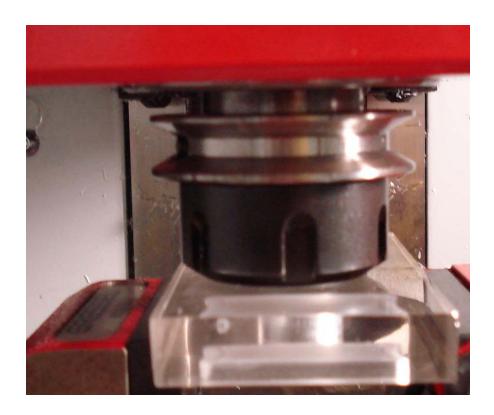
Work Shift:

- 1. Move the MODE dial to JOG position
- Jog a collet tool holder without tool in the collet to the top of the Work Piece & touch using the Direction keys.

Note: Use Feed Dial or Steps to approach at a slower feed rate.

Use piece of paper between nose and Work Piece

EDIT .



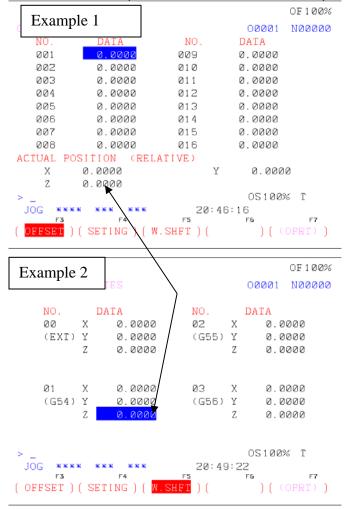
Note: Machine 0 for Z is the spindle nose touching the top of the machine bed.

3. Press the OFFSET/SETT button

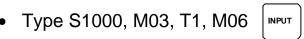


- Press the W.SHIFT
- 4. Make sure that X, Y, Z are all 0 if they have values then the Work Shift will be taken from those values not from the machine 0
- 5. Press the OFFSET Soft key (Gray Button)
 - Record the value in the Actual Position Relative Z (Example 1)
- 6. Press the W.SHFT Soft key (Gray 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

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



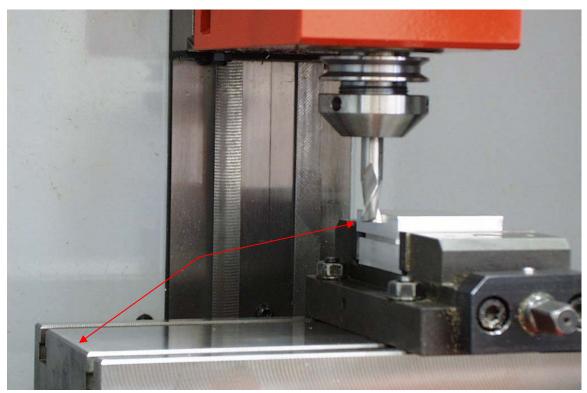
- 9. Jog Spindle up away from WORK PIECE using Z+
- 10. Either follow step 11 or follow step 12 when finished go on to step 13
- 11. Index to a edge finder or tool (Example uses 3/8 end mill) Press
 - Jog the Tool to the left side of the Work Piece & touch using the Direction keys. (Use Feed Dial or 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)





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.

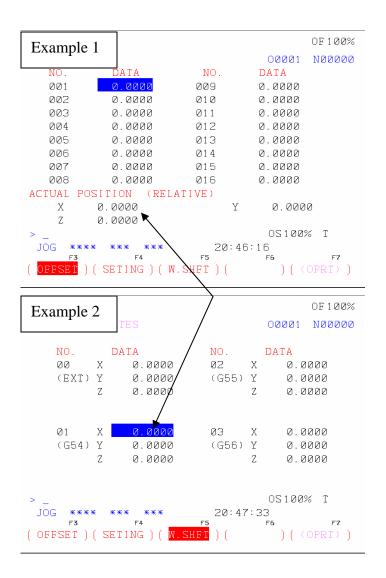


Note: Machine 0 in X is the center of the spindle to the left side of the Machine bed.

- 13. Press the OFFSET/SETT button
- OFFSET SETT.

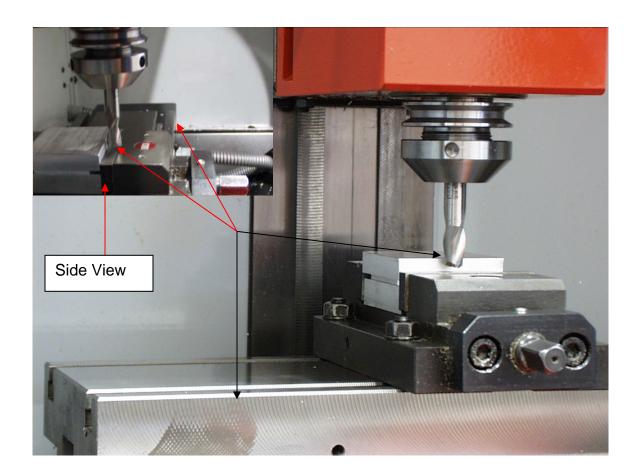
- Example 1
- Record the value in the Actual Position Relative X
- 14. Press the W.SHFT Soft key (Gray Button) (Example 2)
- 15. Move Cursor to 01 (G54) location highlight X
- 16. The Recorded value PLUS the radius of the tool being used to scratch (3/8 Tool) type in Work Coordinates 01 (G54) X

Example: Type 8.463 press Input button



- 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)

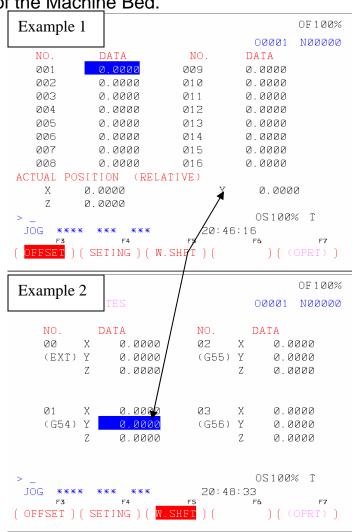


Note: Machine 0 in Y is the center of the spindle to the Front of the Machine bed.

- 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

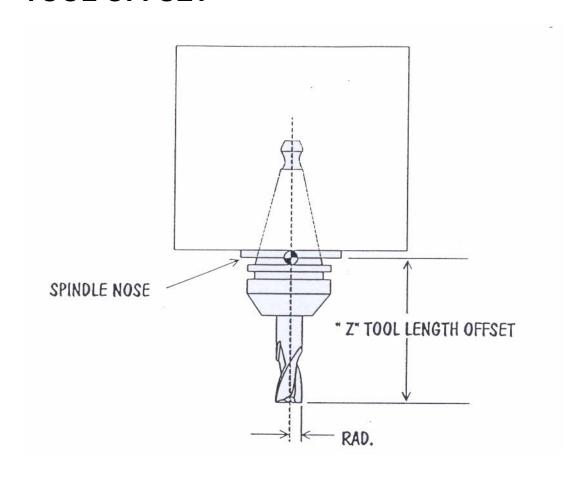
Example: Type 8.463 press Input button

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



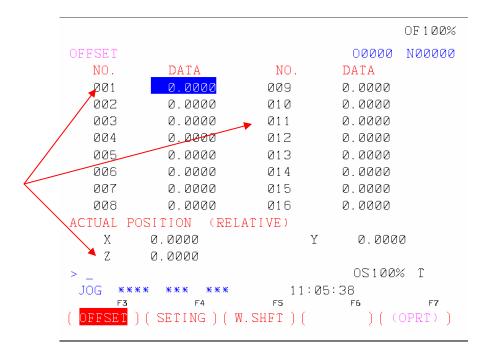
24. Jog the Tool up above the Work Piece

TOOL OFFSET



			OF 1 00%
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 (REL	ATIVE)	
X	0.0000	Y	0.0000
Z	0.0000		
> _			OS100% T
JOG *	*** *** ***	11:0	0 5:38
F:		F5	F6 F7
(OFFSE1	r)(seting)(w	.SHFT) () ((OPRI))

- 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 (SETT.)
- 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

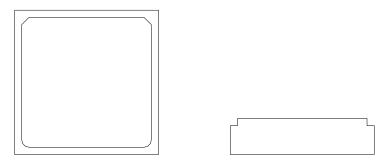


NOTE: When you use a T the H = Height

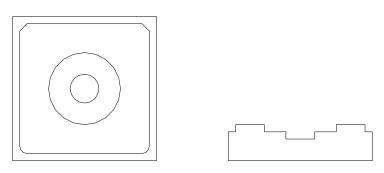
When you use a G41 or G42 the H = Radius

Program Training

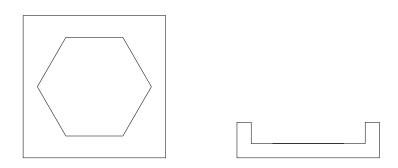
Program O0001



Program O0003



Program O0005



to do functions below

INSERT A NEW PROGRAM

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

Example: <u>0</u>0001 OR <u>0</u>1

• 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 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 INSRT or input 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: <u>0</u>0001 OR <u>0</u>1

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 DELET

DELETE A BLOCK OR LINE NUMBER

- 1. Type the number line and highlight the number line
- 2. Press delete button DELET

• CANCEL MISTYPED WORD (Backspace)

1. Press cancel button

CAN

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 ALTER

• SEARCH FOR NUMBER BLOCK

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



- 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-CODES: Mostly used

M00 Programmed stop, unconditional M01 Programmed stop, conditional M03

Spindle ON clockwise

M04 Spindle ON counter clockwise

M05 **Spindle OFF** M06 Tool Index

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

С Chamfer

F Feed rate, thread pitch

G Path function

Н Tool height, tool radius

I, J, K Circle parameter, scale factor, K number of repetition

Μ Miscellaneous function Ν Block number 1 to 9999 0 Program number 1 to 9499

Р Dwell, subroutine

Q Cutting depth or shift value R Radius, retraction height

S Spindle speed Т Tool called out X, Y, Z Position data Block end

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

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

Program screen & Edit mode

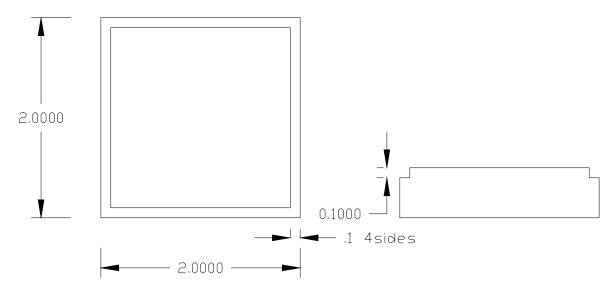
• To edit / change a program / insert new programs & input or output excising programs & offsets

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 <u>O</u>0001



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 10mm Endmill)

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

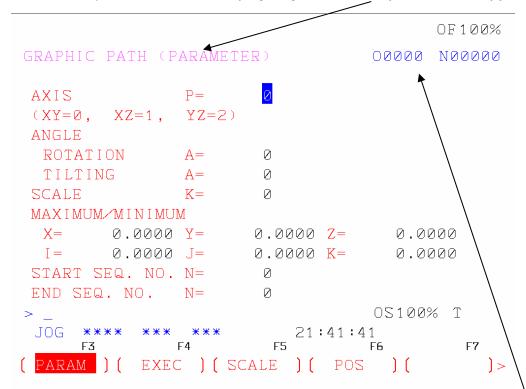
N80 G28 Z3

N85 G28 X2.5 Y2.5

N90 M30

2D Simulation

1. Press Graph button on the Display Keys for the 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 the Soft key **EXEC** for Execution screen

```
OF 100%
GRAPHIC PATH (EXECUTION)
                                     00000 N00000
                                      Χ
                                            0.0000
                                      Y
                                            0.0000
                                      Z
                                            0.0000
                                     OS100% T
 JOG
                            22:09:28
                F4
                          F5
                                               F7
      F3
                                   F6
  EXEC ) ( START ) (
                      STOP ) ( RESET ) ( DELETE )
```

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 <u>O</u> and program number (<u>O</u>0002) or (<u>O</u>2)
- 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

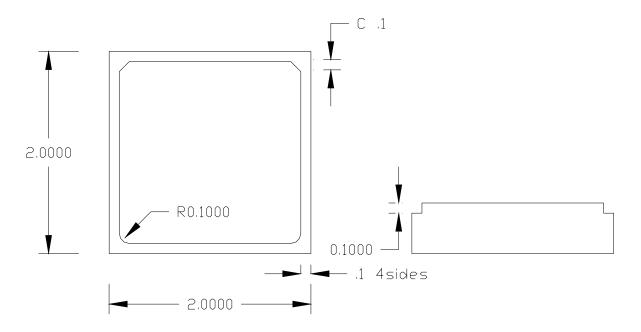
Input Program into Fanuc Software from Drive unit

- 1. Press the **Program** display key
- 2. Type program number to be read Example: letter <u>O</u> and program number (<u>O</u>0002) or (<u>O</u>2)
- 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 <u>O</u>0001 (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

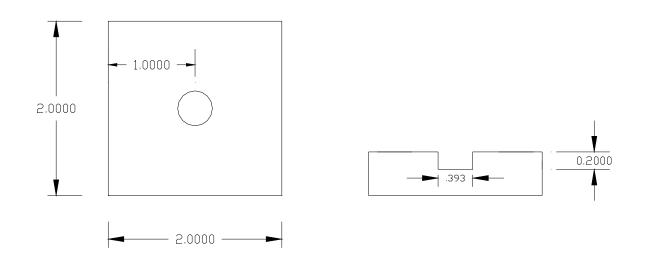
N75 G0 G40 X-1

N80 G28 Z3

N85 G28 X2.5 Y2.5

N90 M30

Program O0002 (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

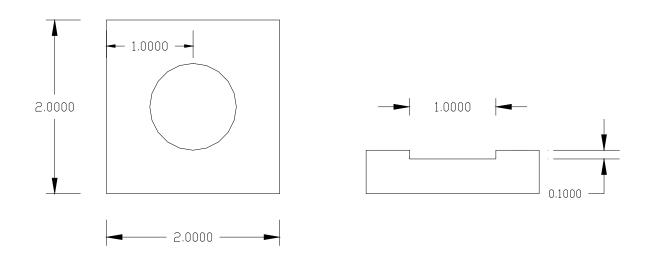
N40 G80

N45 G28 Z3

N50 G28 X2.5 Y2.5

N55 M30

Program <u>O</u>0003 (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)

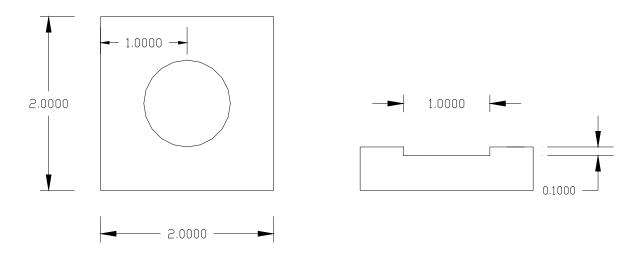
N55 G0 G40 X1

N60 G28 Z3

N65 G28 X2.5 Y2.5

N70 M30

Program <u>O</u>0003 (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 Z3

N65 G28 X2.5 Y2.5

N70 M30

1. To make all programs tie together or all programs <u>O</u>0001 thru <u>O</u>0003 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 <u>O</u>0004 (Main Program)

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

N5 G54

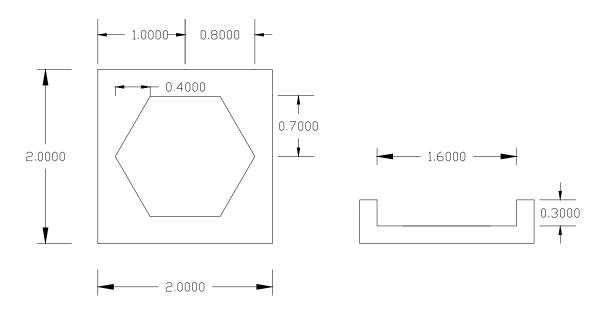
N10 M98 P010001

N15 M98 P010002

N20 M98 P010003

N25 M30

Program <u>O</u>0005 (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 Z3

N50 G28 X2.5 Y2.5

N55 M30

Program <u>O</u>0006 (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