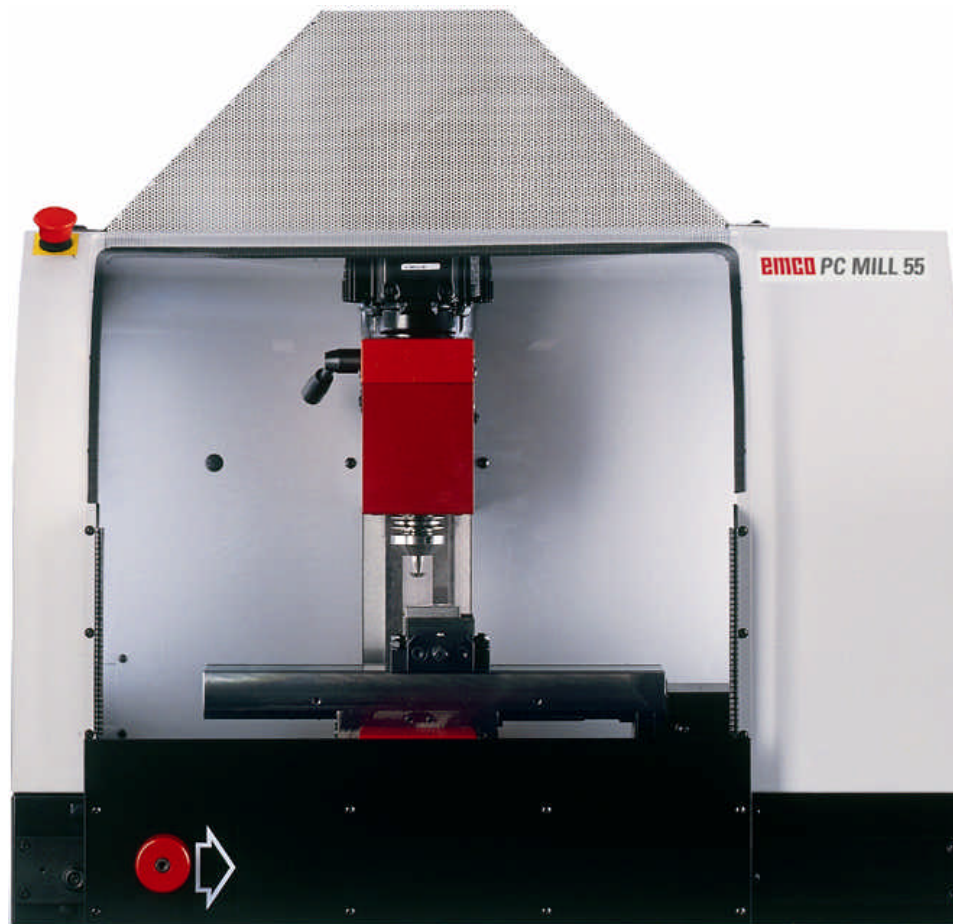




innovative machine tools



# GE FANUC 21 50/55 MILL TRAINING GUIDE

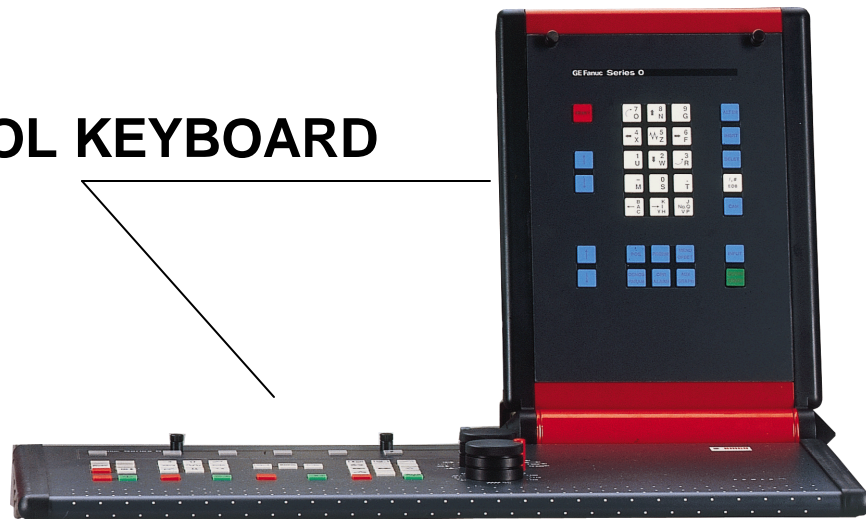
9/11/03 Version 6  
Made by EMCO  
Authored by Chad Hawk

# Training Index

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

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

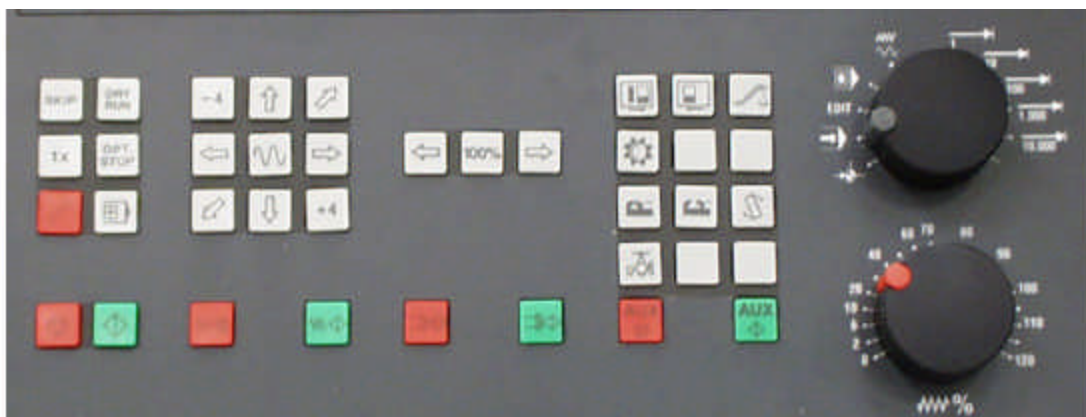
# CONTROL KEYBOARD



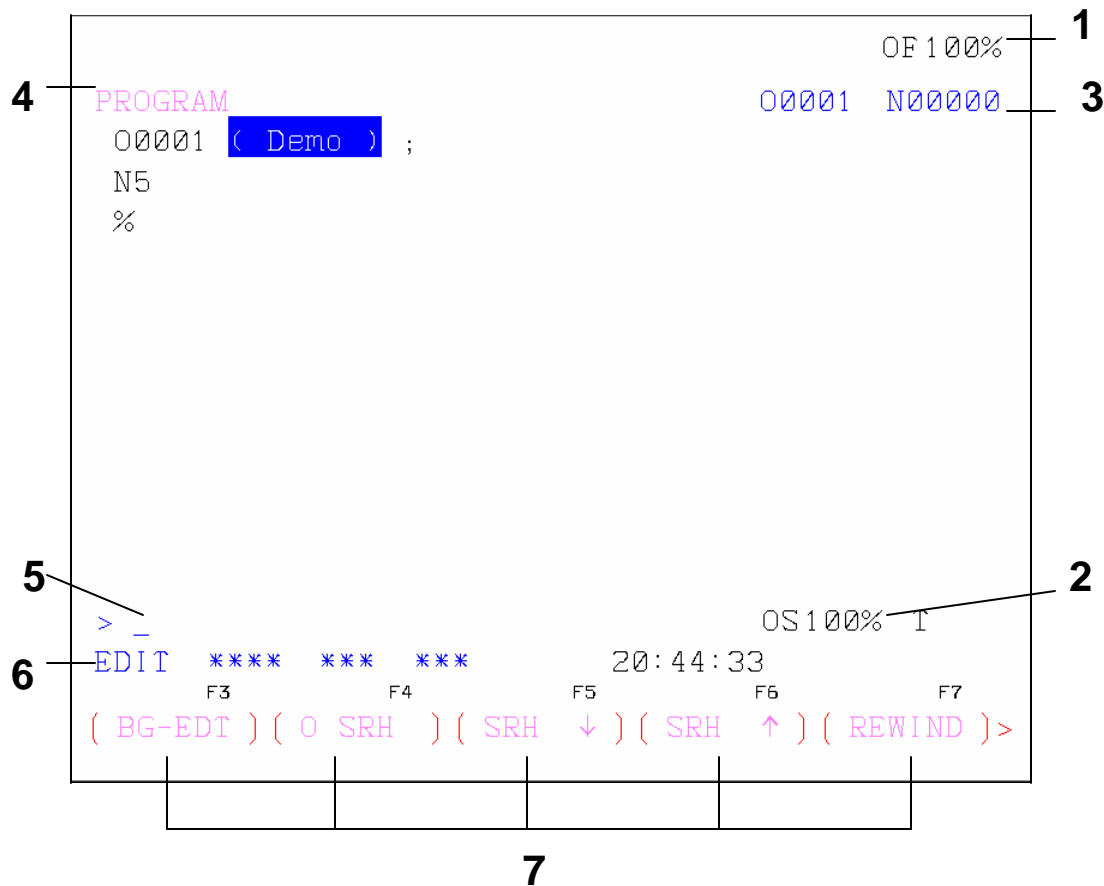
# FANUC 21 CONTROL



# 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

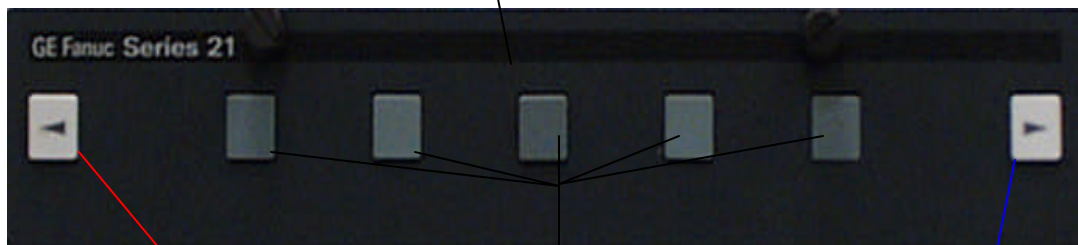


**MESSAGE** = displays operator & alarm messages



**GRAPH** = displays 2-d graph simulation

## **SOFT KEYS**



**SCROLL BACK**

**SOFT KEYS**

**PAGES OVER**

# MACHINE KEYS

## MACHINE FUNCTION KEYS



= Press skip 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

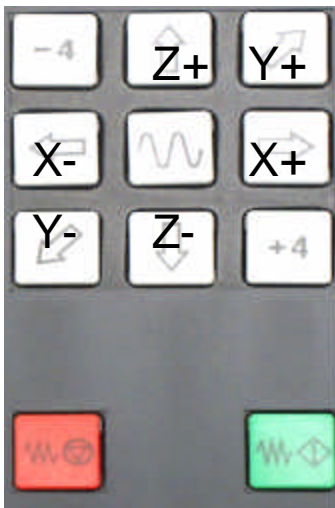


= (Cycle stop) program hold, feed hold



= (Cycle start) program start





## DIRECTION KEYS

These keys control axis directional movements

+4 & -4 = Additional axis

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



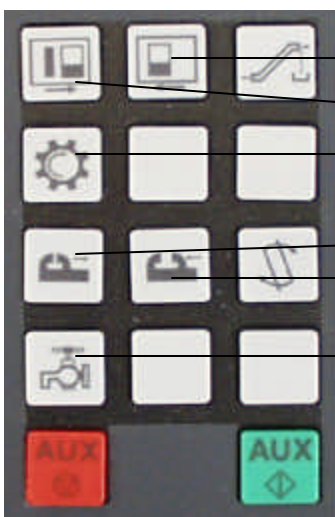
## SPINDLE OVERRIDE KEYS

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



## ACCESSORY FUNCTIONS

Arrow right door open

Arrow left door closed

Press for Rotary axis Indexing

Press once vise closed

Press once vise open

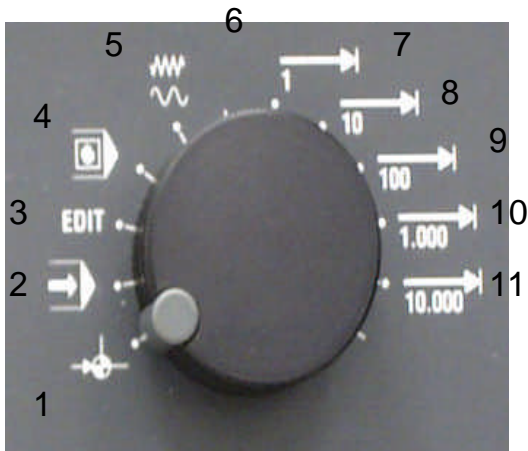
Press once coolant on

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, 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

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

Turret	Air	Rotary	Jog	Jog	Spindle	Spindle	Vise	Vise	Door
1	2	3	4	5	6	7	8	9	0
Tab	Q	W	E	R	T	Y	U	I	O
Backspace	Cancel								
Cap Lock	A	S	D	F	G	H	J	K	L
Shift	Z	X	C	V	B	N	M		
Ctrl	Alt	Space Bar						Alt	Ctrl

Alter	End	Page Up	Page Down
-------	-----	---------	-----------

<	^	>
---	---	---

Num Lock	Dig Run	Op Stop	Stop	Start
Z+	Y+	X+	N+	+
REF ALL				
Y-	Z-			
Reset	Nc	Stop	Start	or

- 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 automotive 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 F11 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

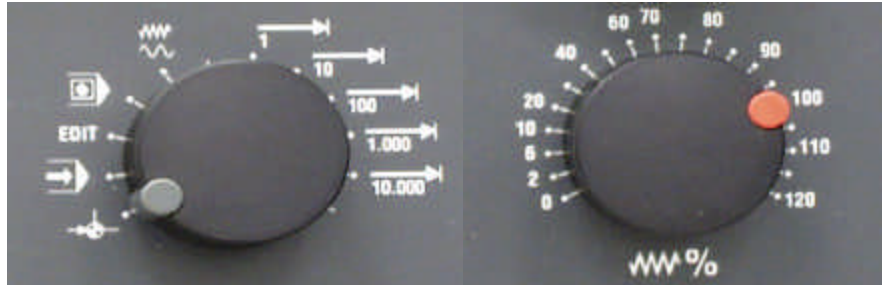
The machine functions are active only with NUM LOCK on

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

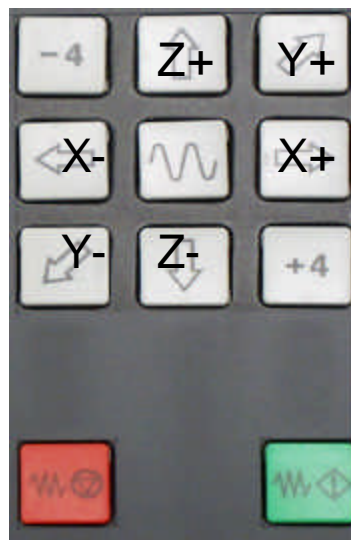
# Turning the Machine On/Entering Fanuc Software

## Referencing the Machine

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



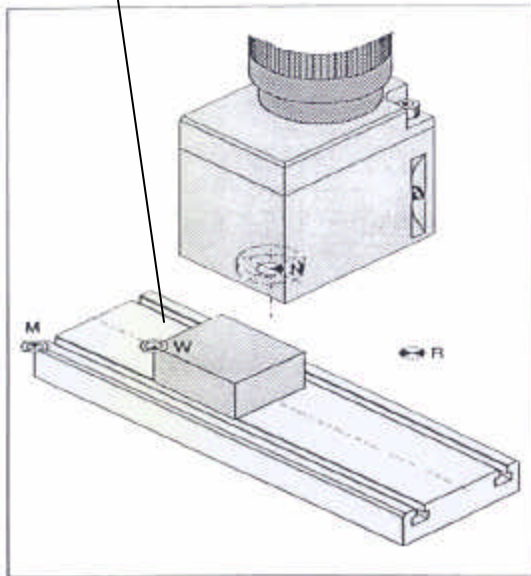
2. Make sure the Door is Closed
3. Press the Z+ (arrow pointing up) this references the Z axis. (Wait until Z is fully reference)
4. Press the X- (arrow pointing left) this references the X axis
5. Press the Y- (arrow pointing left) this references the Y axis



**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".

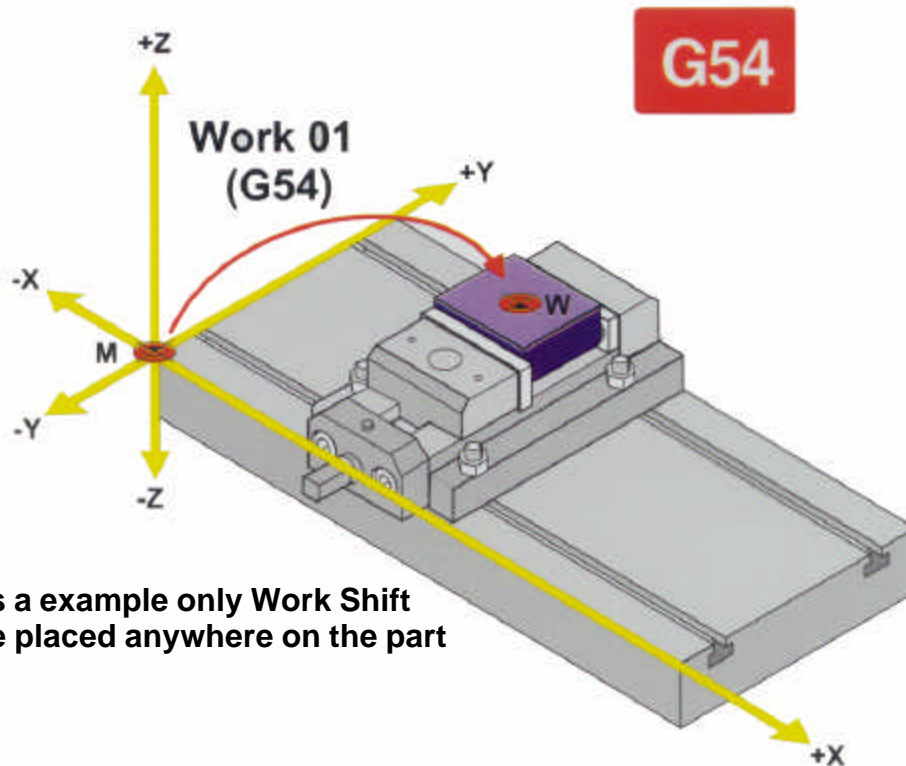
Required after every power failure.

### 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.



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

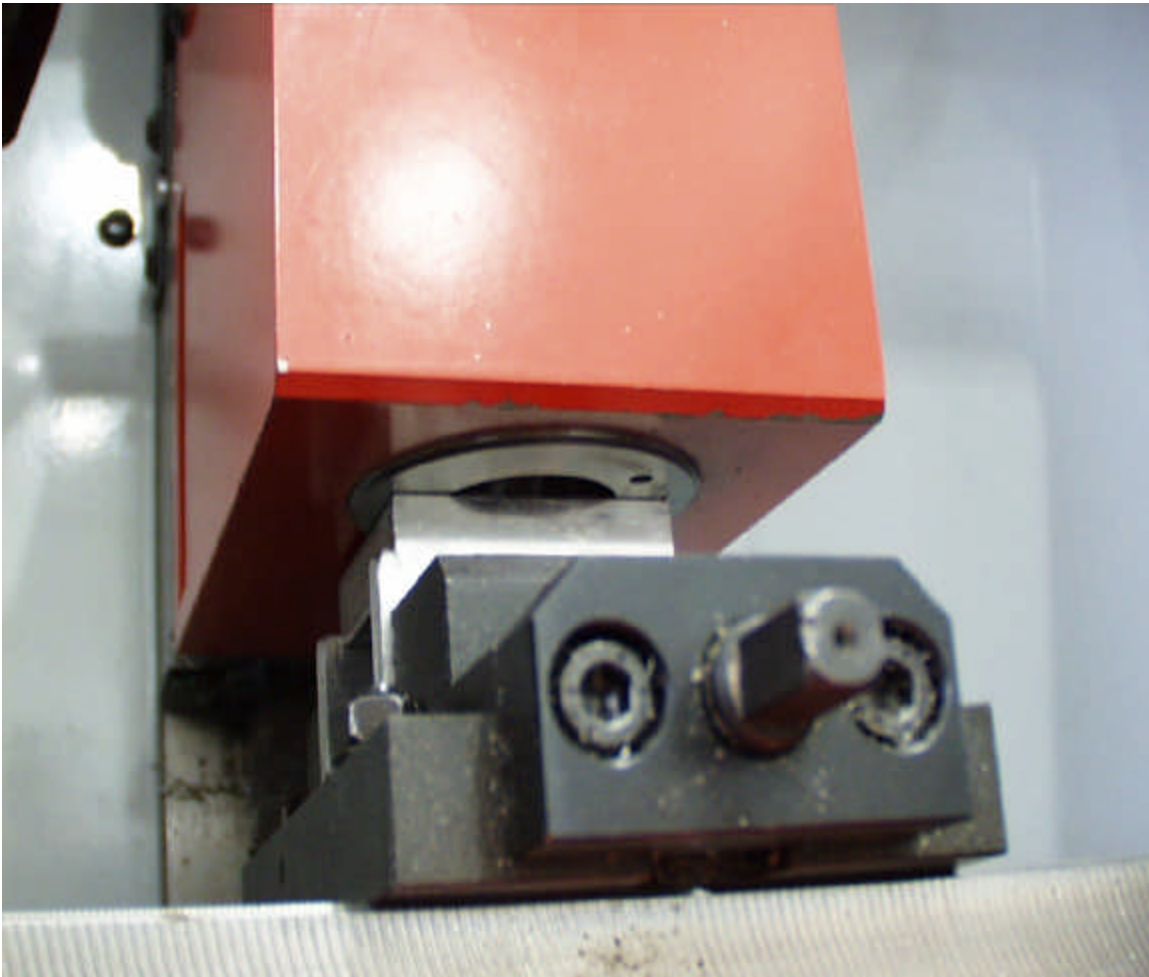


## Work Shift:

1. Move the MODE dial to JOG position
2. Jog the Spindle nose 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



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

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%

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
\*\*\*\*
\*\*\*\*
\*\*\*\*
20:49:22
OS100% T


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

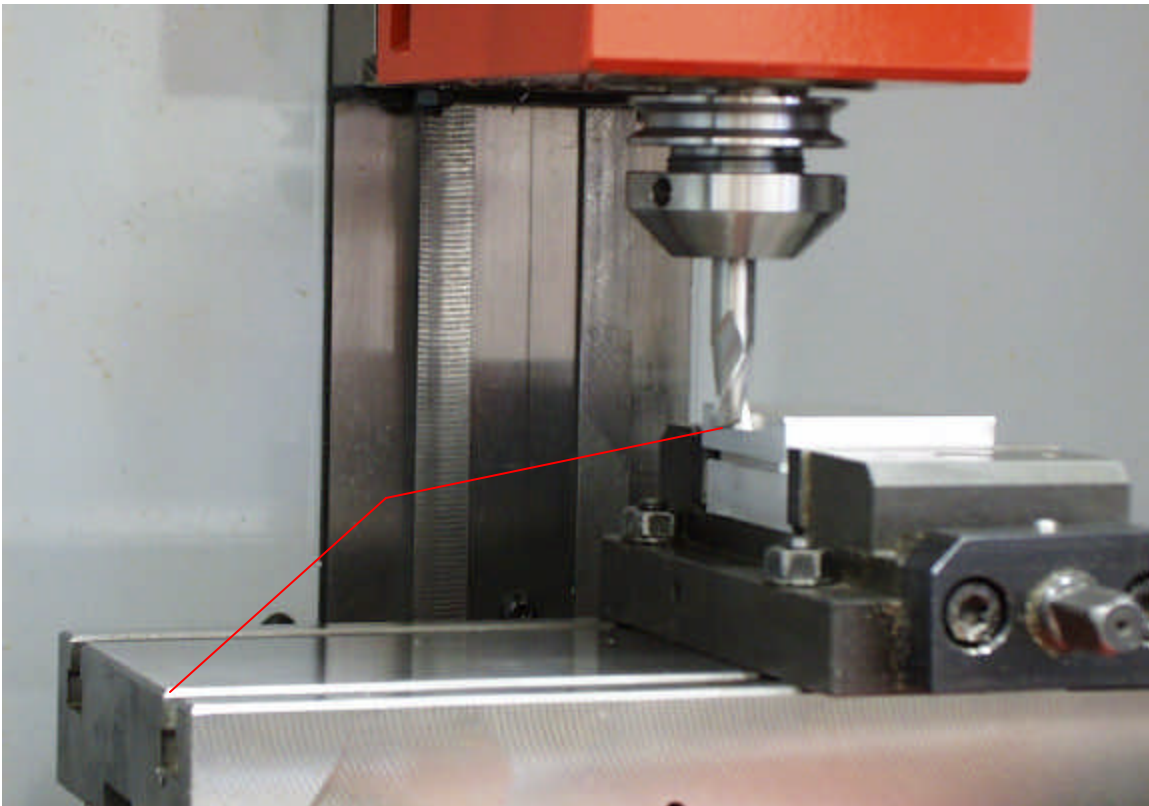
9. Jog Spindle up away from WORK PIECE using Z+
10. Place a edge finder or tool in the Spindle (Example uses 3/8 end mill)
11. Either follow step 12 or follow step 13 when finished go on to step 14
12. Jog the Tool to the left side of the Work Piece & touch using the Direction keys. (Use Feed Dial or Steps to approach at a slower feed)
13. For Scratching move MODE Dial to MDI

- Press the PROGRAM display button  until top of the screen shows MDI (Program)

- Type S1000, M03 

S = RPM M03 = Spindle on Clockwise

- 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.



14. Press the OFFSET/SETT button

OFFSET  
SETT.

- Example 1

- Record the value in the Actual Position Relative X

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

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

17. 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

INPUT

Example 1

OF 100%

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
20:46:16

F3
F4
F5
F6
F7

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

Example 2

OF 100%

00001 N00000

TES

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

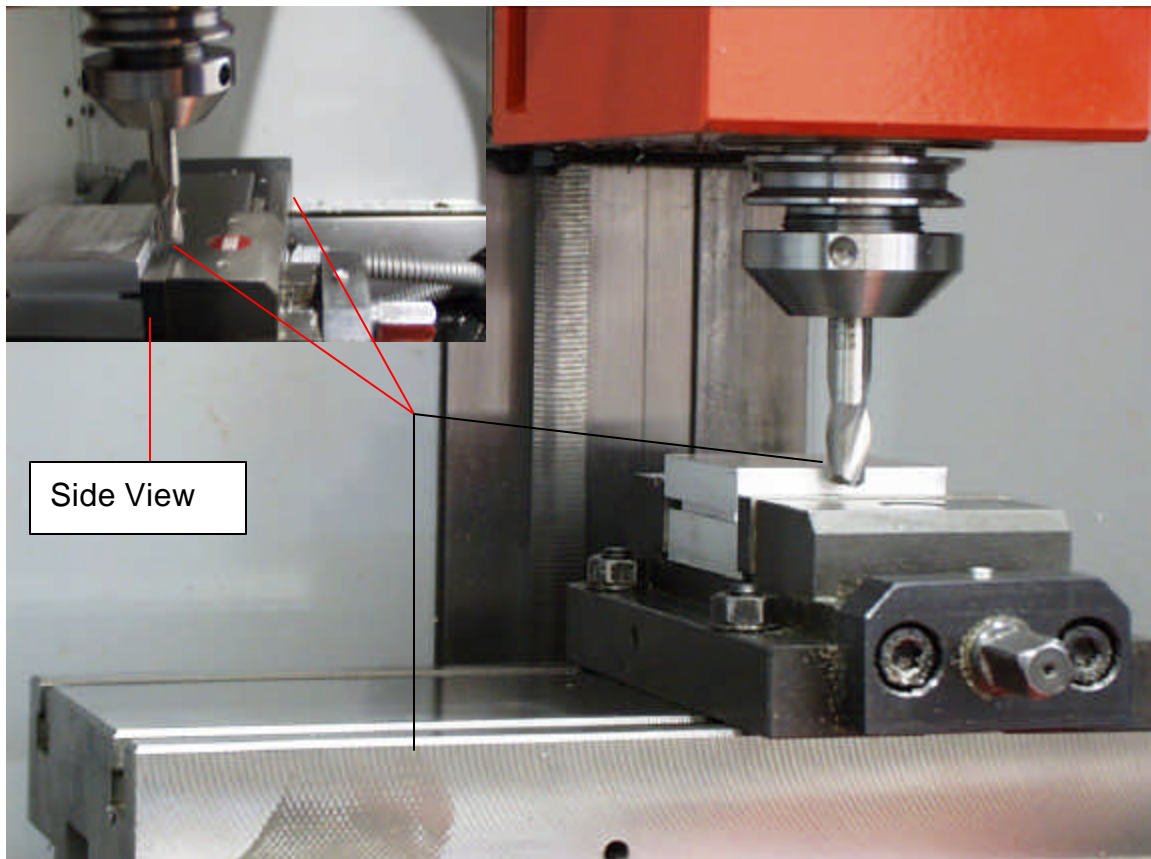
> \_ JOG
20:47:33

F3
F4
F5
F6
F7

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

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


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

- Record the value in the Actual Position Relative Y

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

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

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

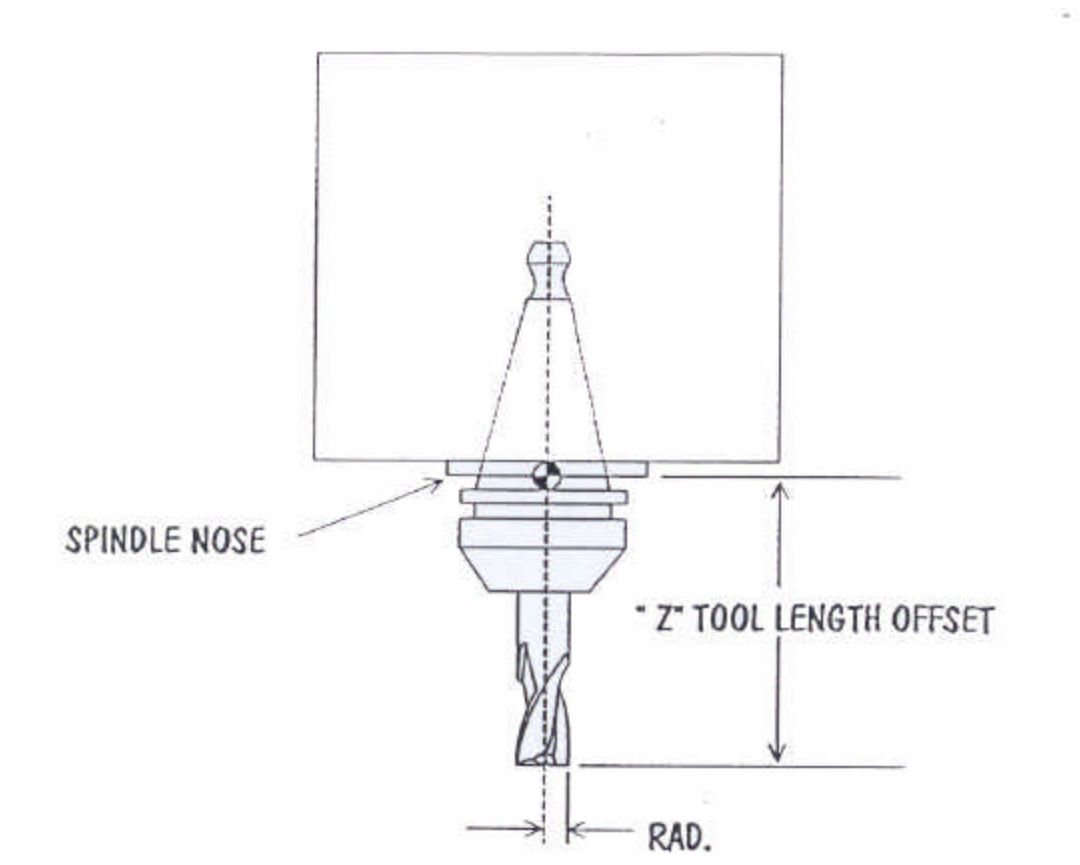
Example: Type 8.463 press Input button 

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


<b>Example 1</b>		OF 100%	
		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 F6 F7			
( OFFSET ) ( SETING ) ( W.SHFT ) ( ) ( OPRT )			
<b>Example 2</b>		OF 100%	
		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 F6 F7			
( OFFSET ) ( SETING ) ( W.SHFT ) ( ) ( OPRT )			

24. 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) }					

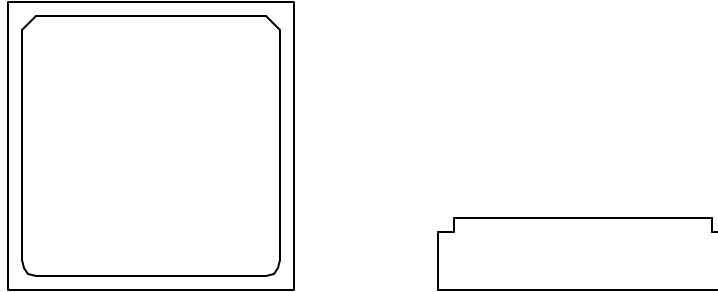
---

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

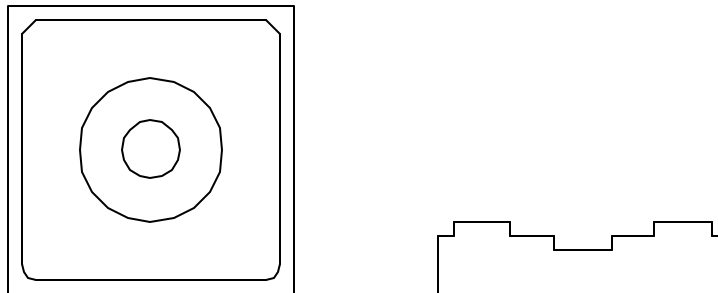
When you use a G41 or G42 the H = Radius

# Program Training

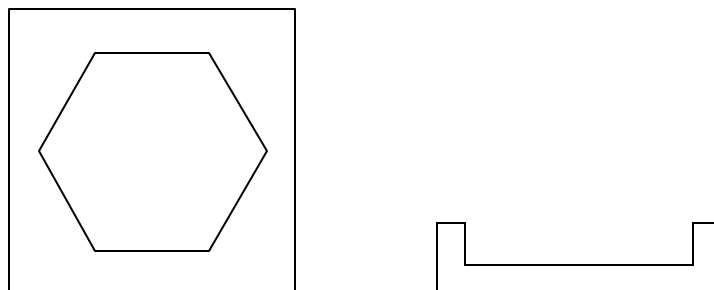
## Program O0001



## 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**

<b>G00</b>	<b>Rapid motion</b>
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
<b>G17</b>	<b>Selection of plane X-Y</b>
G18	Selection of plane Z-X
G19	Selection of plane Y-Z
G20	Dimension in inch
G21	Dimension in millimeter
G28	Approach reference point
<b>G40</b>	<b>Cancel cutter compensation</b>
G41	Cutter compensation left
G42	Cutter compensation right
G43	Tool length compensation positive
G44	Tool length compensation negative
<b>G49</b>	<b>Cancel tool length compensation</b>
G53	Machine coordinate system (00)
<b>G54</b>	<b>Zero point shift 1 (01)</b>
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
<b>G80</b>	<b>Cancel drilling cycle (ALL Drilling Cycles)</b>
G81	Spot or chamfer drilling cycle
G83	Deep hole drilling cycle
<b>G90</b>	<b>Absolute value programming</b>
G91	Incremental value programming
<b>G94</b>	<b>Feed in inch/min</b>
G95	Speed with feed in inch/revolution
<b>G97</b>	<b>Spindle speed per minute</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>
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

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

#### Program screen & Edit mode

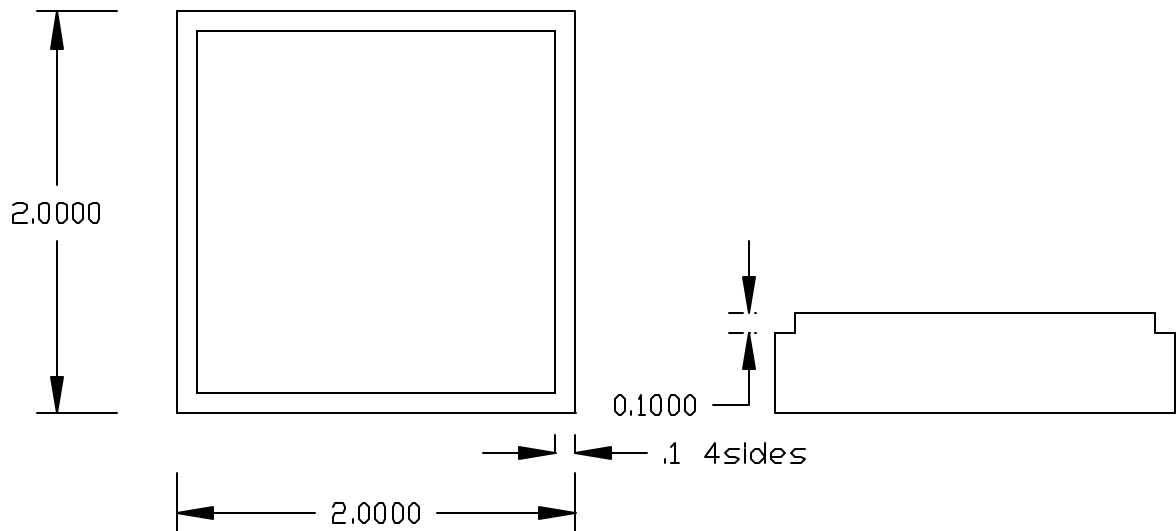
- To edit / change a program / insert new programs & input or output excising programs & offsets

#### 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 1) (2 X 2 X .5 Alum.)

N5 G00 G17 G40 G80

N10 G90 G94 G98

N15 G54

N20 G43 T1 H1 M0 (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

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

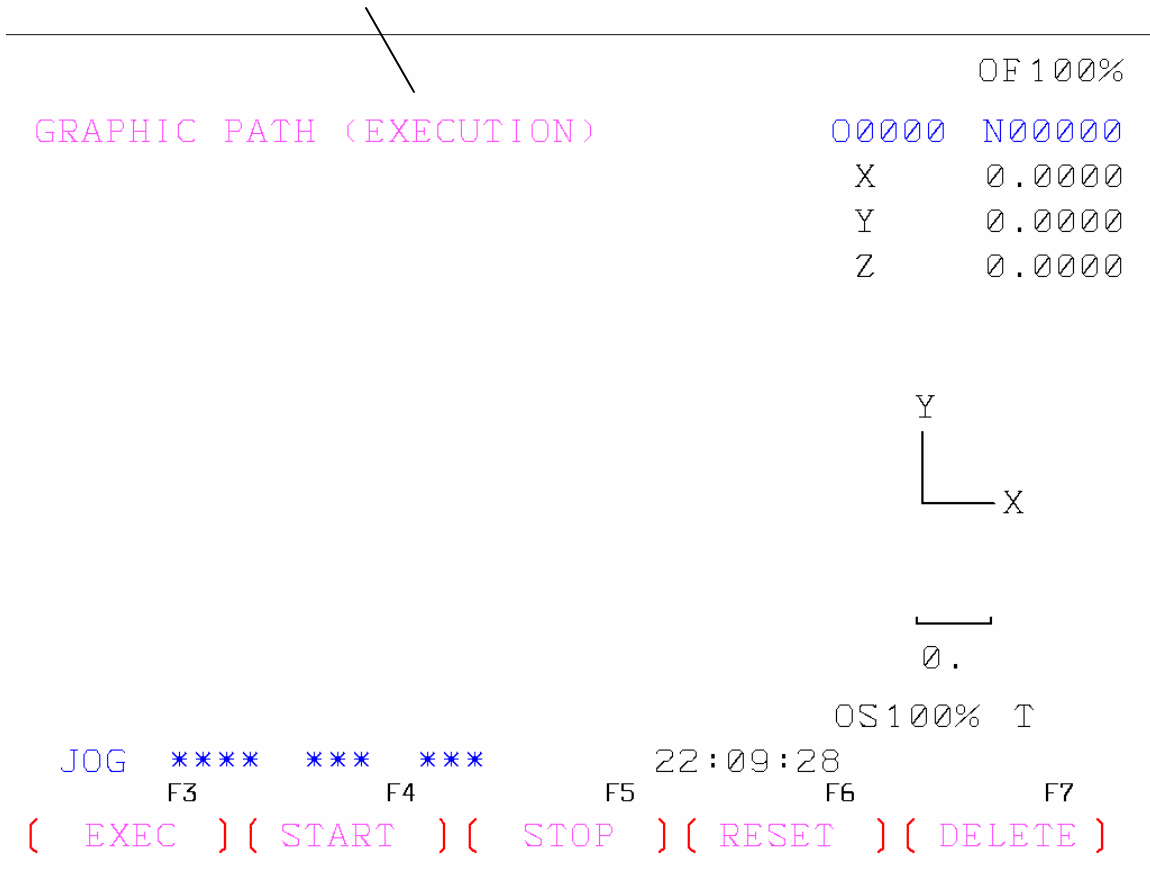
JOG \*\*\*\* \* F3 F4 F5 21:41:41 F6 F7

[ 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



Technical drawing of a square plate with a square hole and a rectangular slot. The plate has a width of 2.0000 and a height of 2.0000. The hole has a side length of 0.1000 and rounded corners with a radius of R0.1000. The slot is 0.1000 wide and 0.1000 high. The drawing includes dimension lines and labels for the hole and slot.

N5 G00 G17 G40 G80

**N15 G54**

N25 S1800 M3

N35 X-1 Y1

N45 G1 G41 H11 X.1 F7

## N55 X1.9 C.1

**N60 Y.1 R.1**

**N65 X.1 R.1**

N70 Y1

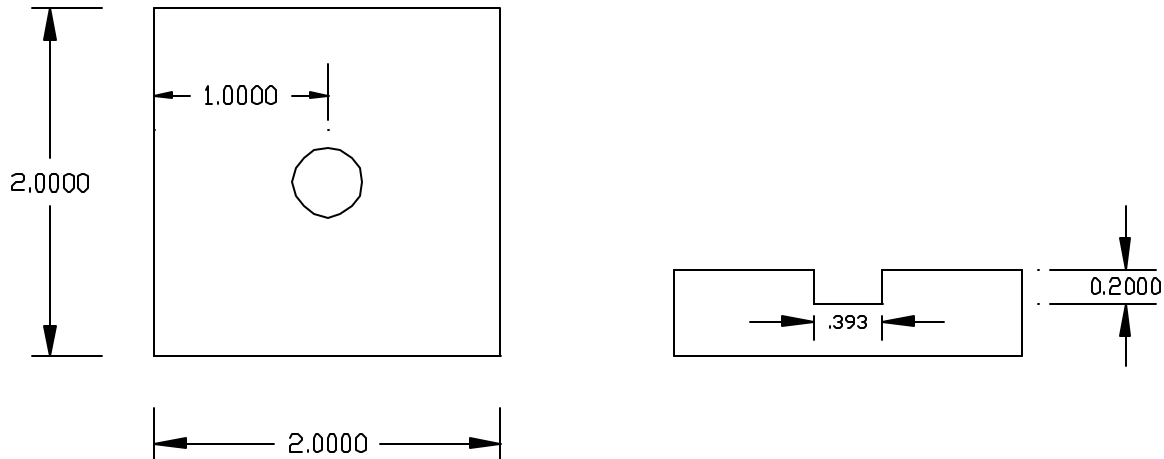
N75 G0 G40 X-1

N80 G28 Z3

N85 G28 X2.5 Y2.5

N90 M30

## Program 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 M0 (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**

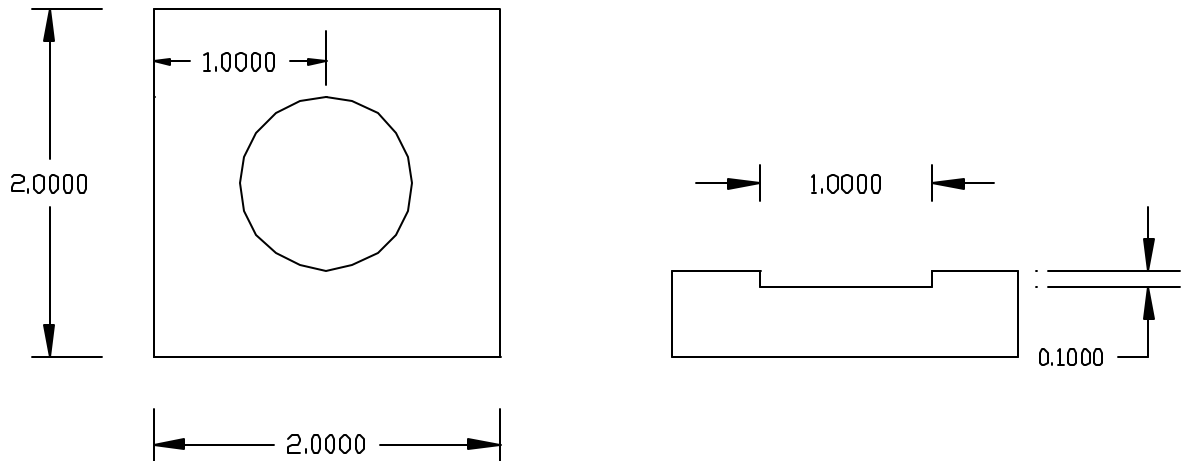
**N40 G80**

N45 G28 Z3

N50 G28 X2.5 Y2.5

N55 M30

## Program Q0003 (I & J)



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

N5 G54

N10 G43 T1 H1 M0 (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)**

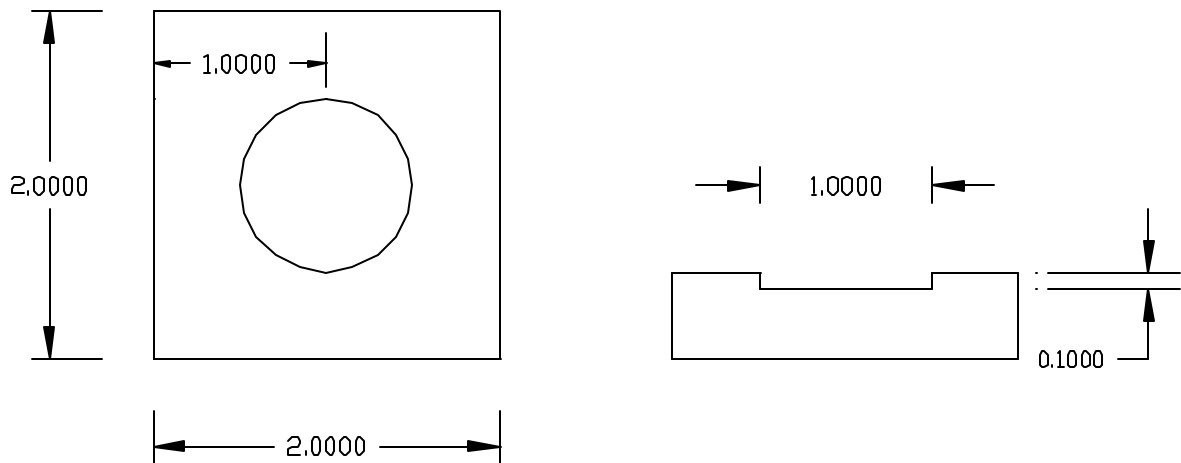
**N55 G0 G40 X1**

N60 G28 Z3

N65 G28 X2.5 Y2.5

N70 M30

## Program O0003 (R)



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

N5 G54

N10 G43 T1 H1 M0 (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 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

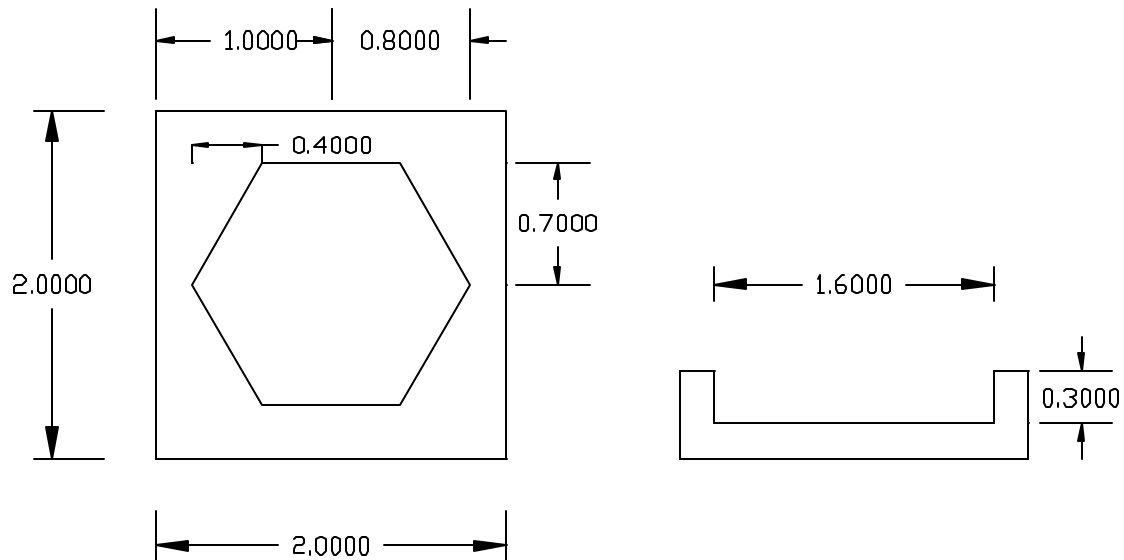
N10 M98 P010001

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 M0 (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 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